Interactive Manual Data Input Function

ONE-TOUCH IGF

OSP5020L CNC SYSTEM

OPERATION MANUAL -APPLICATION-

(3rd Edition)

OKUMA

PART NO. KPC-0089-0(

PUBLICATION NO. 3441-E-R2
(AUGUST 199(

INTRODUCTION

All useful functions of the one-touch IGF are described in this manual.

The table of contents of this manual is itemized so that the user can use it like a dictionary, for example locating the necessary explanation from the table of contents on which names of functions, screens, or function keys are used.

Please make the most of this manual to use the one-touch IGF more effectively.

TABLE OF CONTENTS

PAGE

PAGE

Speeds

PAGE

PAGE

PART I APPLICATION

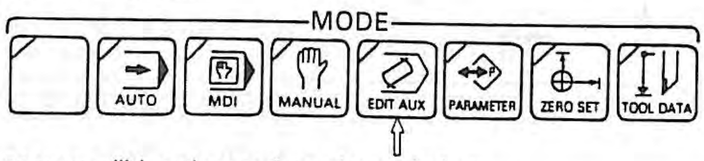
SECTION 1 STARTING-UP ONE-TOUCH IGF SYSTEM

1. Selection of One-touch IGF Mode

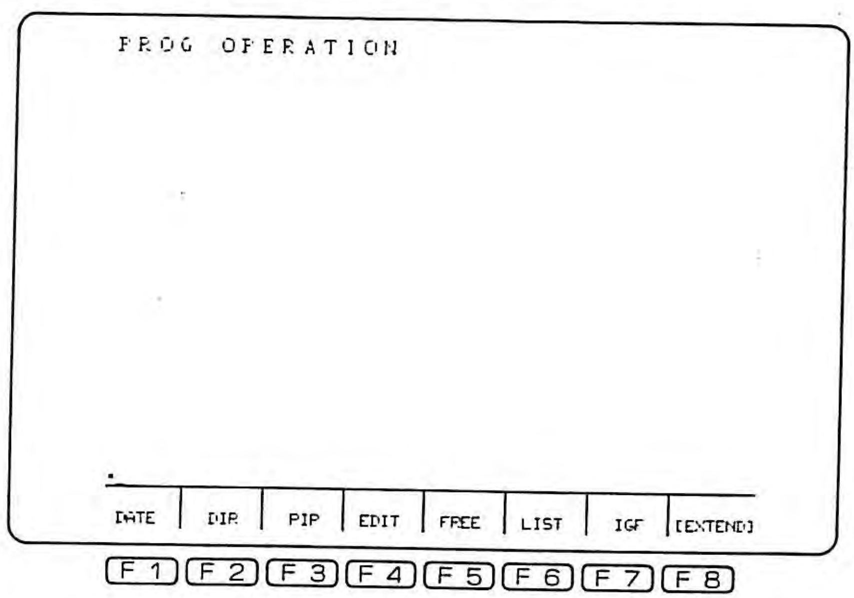
First establish the one-touch IGF mode following the procedure below.

Operating Procedure:

1) Press the **EDIT AUX** key.



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



PART I APPLICATION

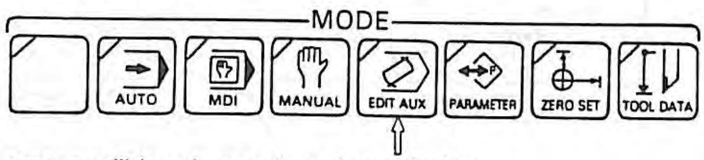
SECTION 1 STARTING-UP ONE-TOUCH IGF SYSTEM

1. Selection of One-touch IGF Mode

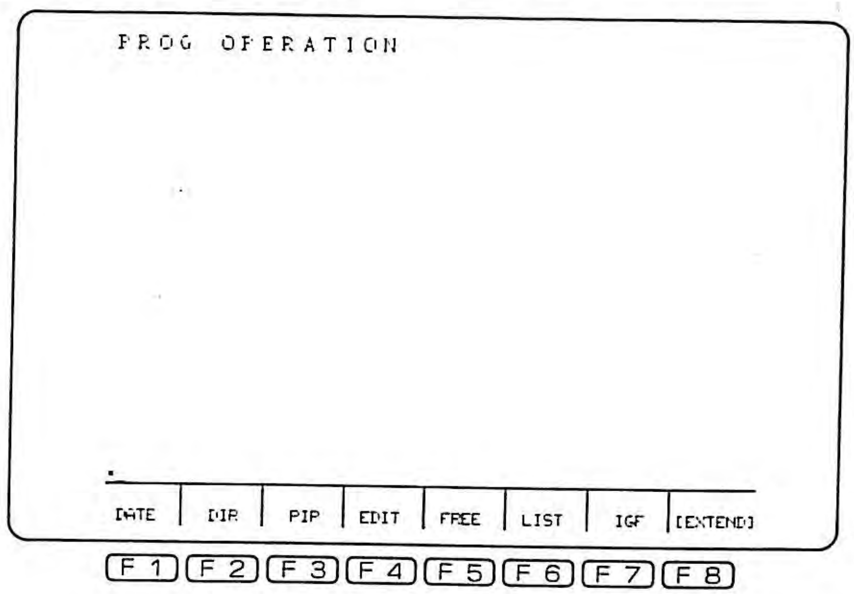
First establish the one-touch IGF mode following the procedure below.

Operating Procedure:

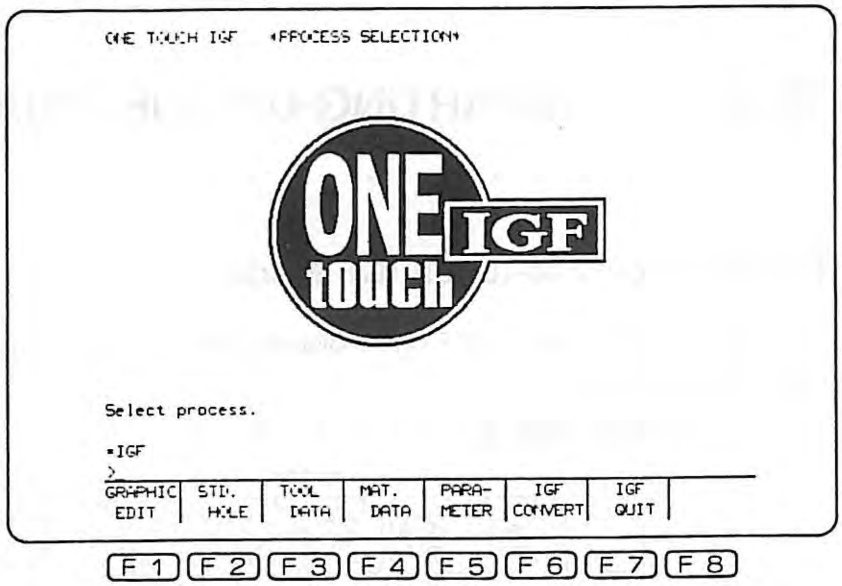
1) Press the **EDIT AUX** key.



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



3) Press the function key [F7] (IGF). The CRT screen will be changed as shown below. This screen is called "one-touch IGF start-up page" and the one-touch IGF mode is established.



(One-touch IGF start-up page)

2. Turning On Graphic Edit Function

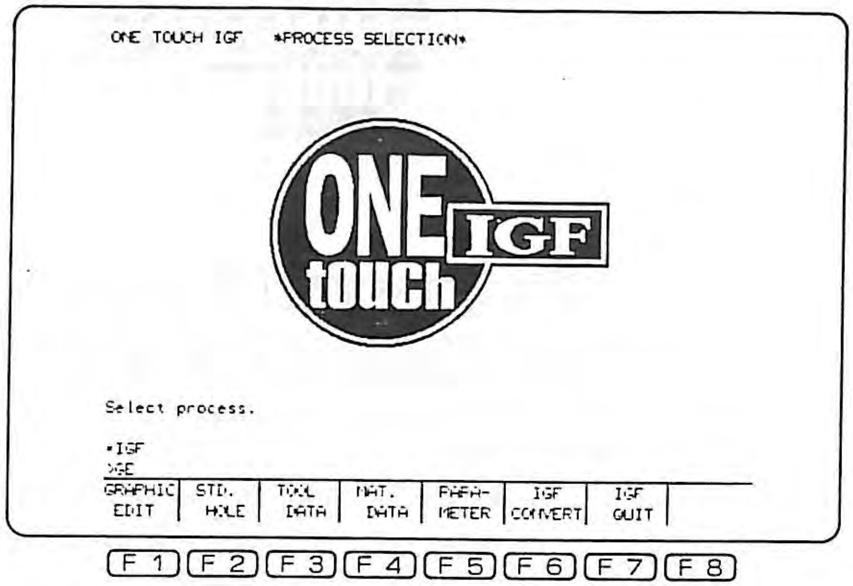
To create a part program, select the GRAPHIC EDIT function.

Operating Procedure:

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

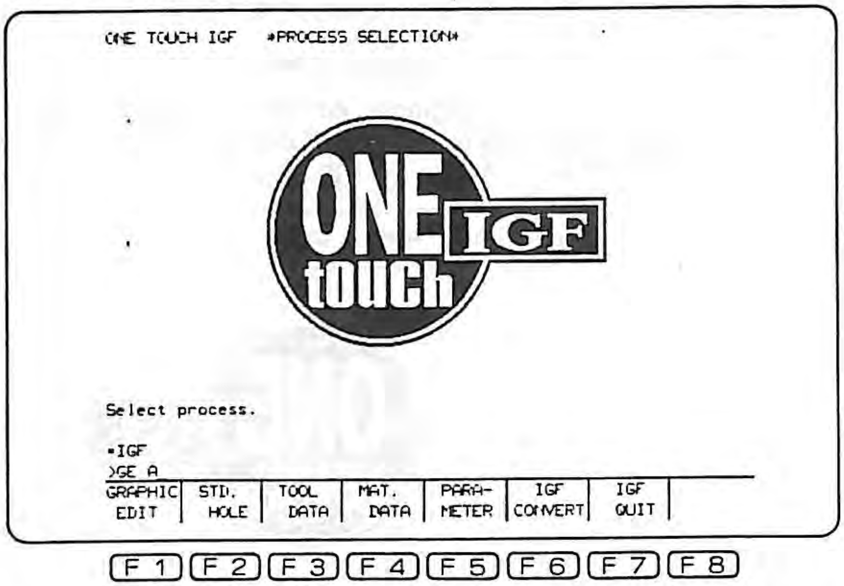
2) The CRT screen will be changed as shown below.

   The data entered in the graphic edit process, such as material, blank shape, or finished product shape (these data is called "IGF data"), will be stored in the computer as a unit. This unit is called "IGF file". Assign the IGF file a file name.



3) To assign the IGF file a file name "A.IGF", for example, key in [A] using the character key, and press the **WRITE** key. If the entry of file name is omitted, the file name "I.IGF" will be automatically assigned.

Here, "A" of "A.IGF" or "I" of "I.IGF" is called "main file name", and ".IGF" of "A.IGF" or "I.IGF" is called "extension".



Note 1: *In addition to newly creating an IGF file, it is also possible to create a new IGF file from the IGF file stored in the memory.*

i) *To newly create an IGF file named "ABC.IGF"*

Key in [A], [B], [C], and press the ***WRITE*** *key.*

ii) *To correct an existing IGF file named "A.IGF"*

Key in [A] and press the ***WRITE*** *key.*

iii) *To correct an existing IGF file named "A.IGF" and give it a new name "B.IGF"*

Key in [A], [,], [B], and press the ***WRITE*** *key.*

Note 2: *When correcting the existing IGF file, it is possible to use the directory screen.*

Display the directory screen and move the cursor to the file name to be selected using the page keys ***PAGE ↓***, ***PAGE ↑*** *and cursor control keys* **↑**, **↓**.

i) *To select an IGF file from all the IGF files stored in the bubble memory*

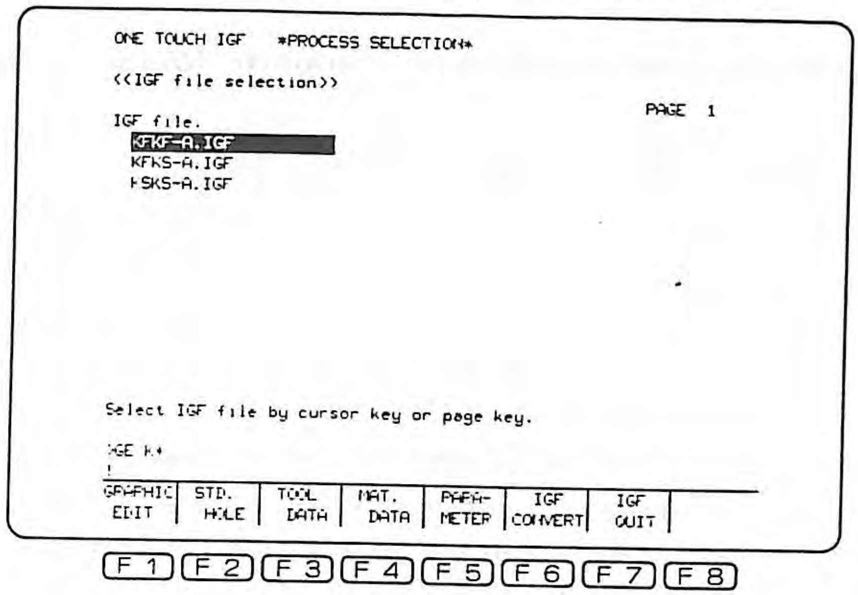
Key in [*] and press the ***WRITE*** *key.*

ii) *To select an IGF file from the IGF files whose name begins with "K"*

Key in [K], [*], and press the ***WRITE*** *key.*

iii) *To select an IGF file from the IGF files whose main file name consists of three characters and begins with "KA"*

Key in [K], [A], [?], and press the ***WRITE*** *key.*



SECTION 2 STANDARD HOLE DATA SETTING

1. What is the Function of Standard Hole Data Setting?

This is the function to register the hole data for tapping in turning operation, and for drilling, boring, tapping, or reaming in multiple machining operation.

For example, when the nominal diameter of tap is designated in the one-touch IGF specification, the data necessary to execute tapping, for example center drill diameter, pre-drilled diameter, chamfer size, or pitch is automatically determined using the standard hole data.

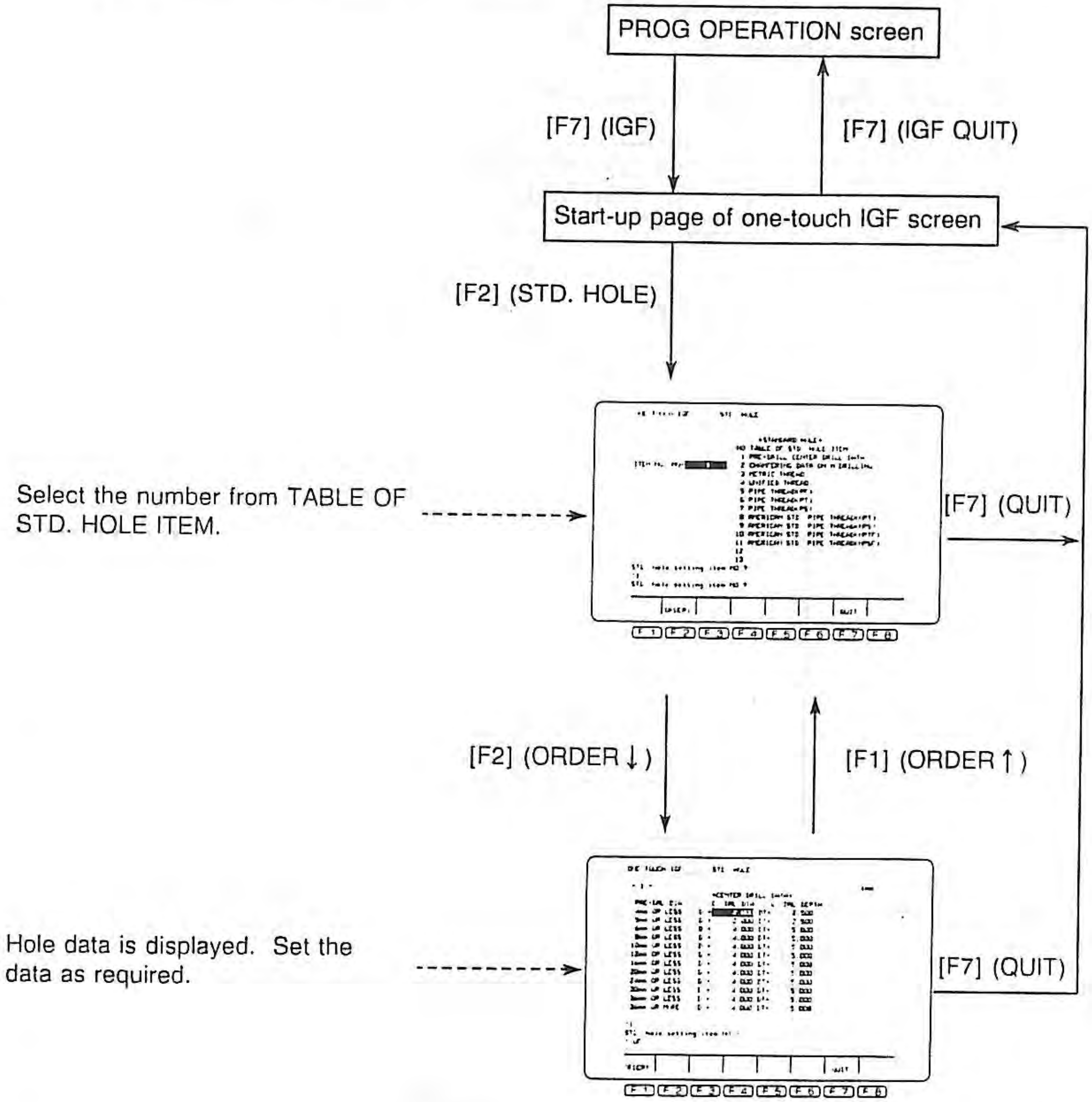
Standard values are factory-set for standard hole data. These values can be changed by user as needed.

Standard hole data includes the following data:

- Diameter and depth of center drill with regard to drilled hole
- Chamfer size with regard to drilled hole
- Pre-drilled diameter, pitch (number of threads), and chamfer size with regard to the nominal diameter of thread

Input Procedure and Screen Transfer:

The STD. HOLE screen is called from the PROG OPERATION screen as diagramed below.



Note: *Data setting screen for No. 3 METRIC THREAD and No. 4 UNIFIED THREAD has two pages.*

Change the page using the function keys [F2] (ORDER↓) and [F1] (ORDER↑).

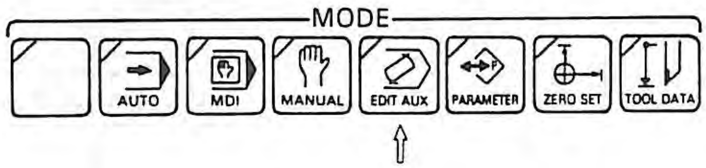
2. Standard Hole Data Setting Procedure

The standard hole data setting procedure is explained using an example.

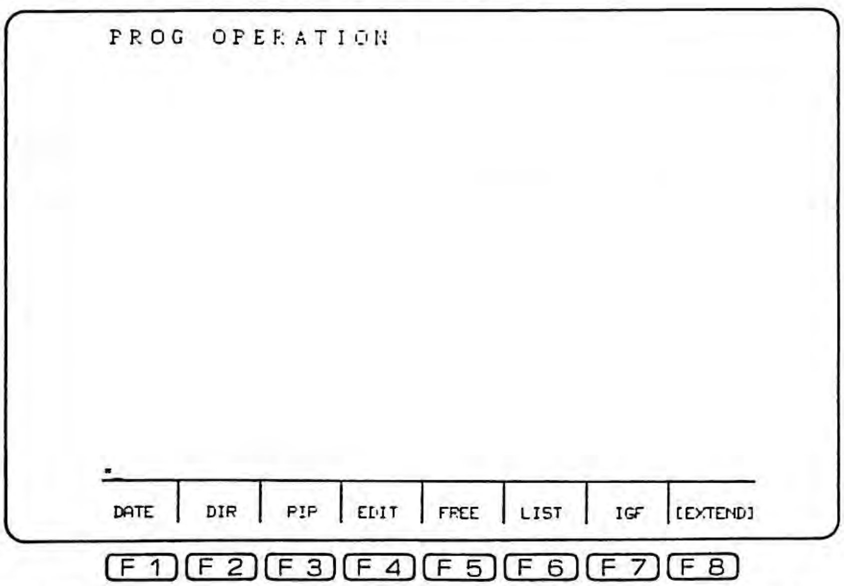
Example: Designation of chamfer size with regard to M-drill hole

Operating Procedure:

1) Press the **EDIT AUX** key to select the program operation mode.

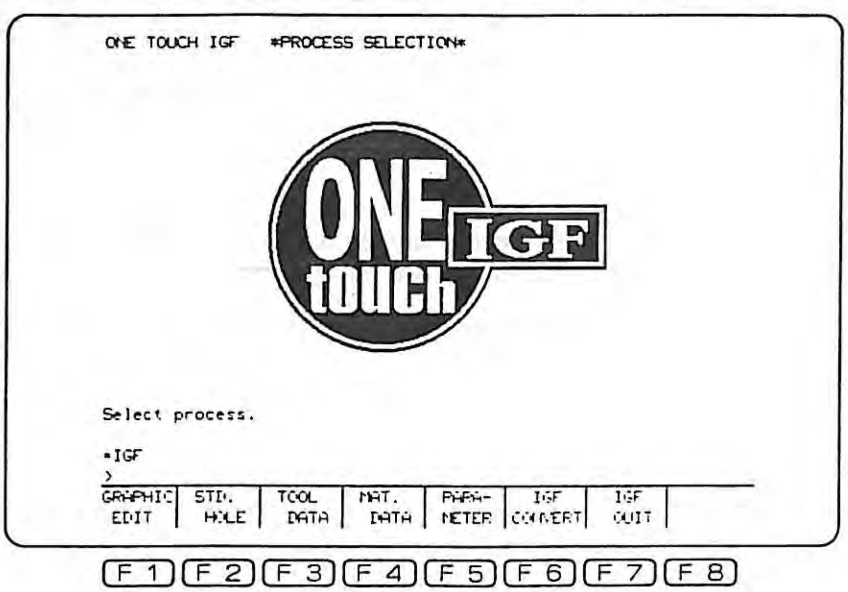


The CRT screen will be changed as shown below.

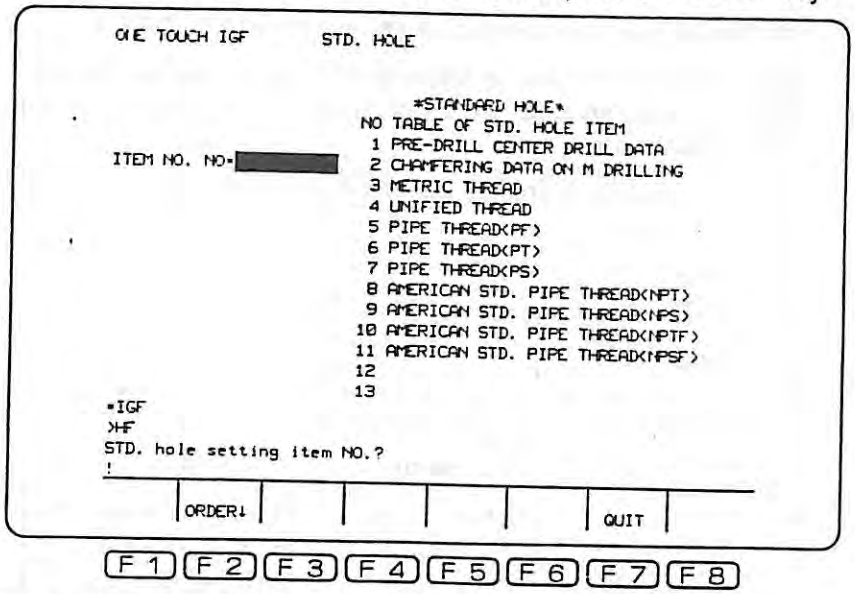


2) Press the function key [F7] (IGF) to establish the one-touch IGF mode.

3) Press the function key [F2] (STD. HOLE) to enable the standard hole data setting.



4) Select the item number from the TABLE OF STD. HOLE ITEM. In this example, select "1. PRE-DRILL CENTER DRILL DATA". Key in [1] and press the **WRITE** key.

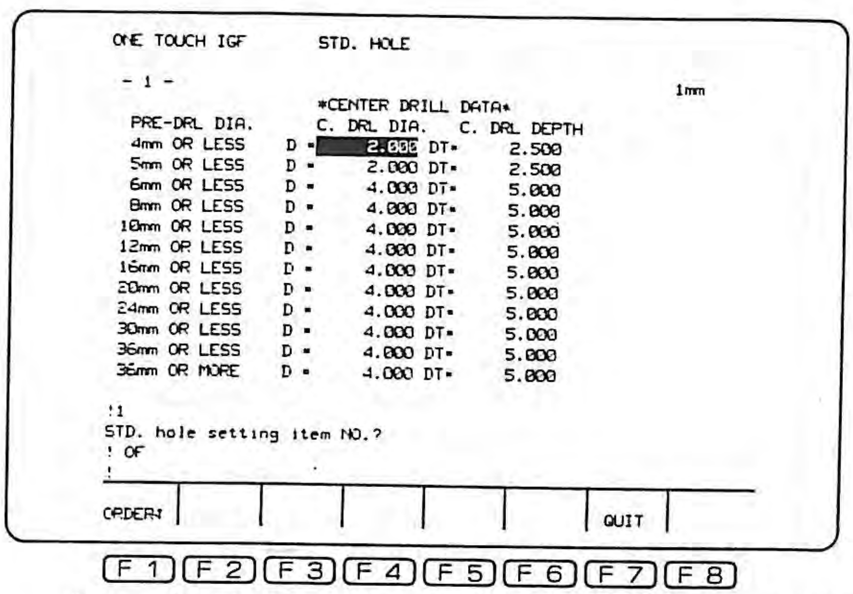


5) Press the function key [F2] (ORDER↓).

6) Move the cursor to the data column using the cursor control keys and enter the new data.

In this example, no data setting is required as the CENTER DRILL DATA is factory-set by Okuma.

Refer to 3. "Items to be Set as Standard Hole Data" for details of types of data which can be set by user.



Note: *Data setting screen for No. 3 METRIC THREAD and No. 4 UNIFIED THREAD has two pages.*

Change the page using the function keys [F2] (ORDER↓) and [F1] (ORDER↑).

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

The start-up page of the one-touch IGF screen will be displayed.

3. Items to be Set as Standard Hole Data

The following are the items to be set as standard hold data.

When the metric system is selected as the unit system, screens in the metric specification will be displayed, while screens in the inch specification will be displayed when the inch system is selected as the unit system.

For the procedure to change the unit system, refer to the Operation Manual of OSP5020L (Publication No. 3272-E).

Item No.	Title/Content	Item/Data		
1	PRE-DRILL CENTER DRIIL DATA * Diameter and depth of center drill with regard to the pre-drill hole diameter are set.	Pre-drill hole diameter Metric 4 mm or less 5 mm or less 6 mm or less 8 mm or less 10 mm or less 12 mm or less 16 mm or less 20 mm or less 24 mm or less 30 mm or less 36 mm or less 36 mm or more	 Inch (5/32 in. or less) (3/16 in. or less) (15/64 in. or less) (5/15 in. or less) (25/64 in. or less) (15/32 in. or less) (5/8 in. or less) (35/32 in. or less) (15/16 in. or less) (1-11/64 in. or less) (1-13/32 in. or less) (1-13/32 in. or more)	Center drill diameter D Center drill depth DT
2	CHAMFERING DATA ON M DRILLING * Chamfer size with regard to the drill hole diameter is set. * Data is set for this item with the one-touch IGF for the multi-machining specification.	Drill hole diameter Metric 4 mm or less 5 mm or less 6 mm or less 8 mm or less 10 mm or less 12 mm or less 16 mm or less 20 mm or less 24 mm or less 30 mm or less 36 mm or less 36 mm or more	 Inch (5/32 in. or less) (3/16 in. or less) (15/64 in. or less) (5/15 in. or less) (25/64 in. or less) (15/32 in. or less) (5/8 in. or less) (35/32 in. or less) (15/16 in. or less) (1-11/64 in. or less) (1-13/32 in. or less) (1-13/32 in. or more)	Chamfer size D2

<table>
<tr><th>Item No.</th><th>Title/Content</th><th colspan="2">Item/Data</th></tr>
<tr><td rowspan="2"></td><td rowspan="2">METRIC THREAD

* Pre-drilled diameter, pitch, and chamfer size with regard to the nominal diameter of thread are set.</td><td>* Nominal diameter of thread
<u>Common to metric and inch specifications</u>
M2
M2.5
M3
M4
M5
M6
M8</td><td>Pre-drilled diameter D1
Pitch F
Chamfer size D2</td></tr>
<tr><td>M10
M12
M16
M20
M24
M30
M36</td><td>Note:
Data setting screen for metric thread has two pages. To set the data for M10 - M36 thread, change the display page using the function key [F2] (ORDER↓). Pressing the function key [F1] (ORDER↑) returns the screen to the data setting page for M2 - M8 thread.</td></tr>
<tr><td rowspan="2">4</td><td rowspan="2">UNIFIED THREAD

* Pre-drilled diameter, number of threads, and chamfer size with regard to the nominal diameter of thread are set.</td><td>* Nominal diameter of thread
<u>Common to metric and inch specifications</u>
No. 1
No. 2
No. 3
No. 4
No. 5
No. 6
No. 8
No. 10
No. 12</td><td>Pre-drilled diameter D1
Number of threads F (per one inch)
Chamfer size D2</td></tr>
<tr><td>U1/4
U3/8
U1/2
U3/4
U1
U1-1/4
U1-1/2
U2</td><td>Note:
Data setting screen for unified thread has two pages. To set the data for U1/4 - U2 thread, change the display page using the function key [F2] (ORDER↓). Pressing the function key [F1] (ORDER↑) returns the screen to the data setting page for No.1 - No. 12 thread.</td></tr>
</table>

Item No.	Title/Content	Item/Data	
5	PIPE THREAD (PF) * Pre-drilled diameter, number of threads, and chamfer size with regard to the nominal diameter of thread are set.	* Nominal diameter of thread <u>Common to metric and inch specifications</u> PF1/8 PF1/4 PF3/8 PF1/2 PF3/4 PF1 PF1-1/4 PF1-1/2 PF2	Pre-drilled diameter D1 Number of threads F (per one inch) Chamfer size D2
6	PIPE THREAD (PT) * Pre-drilled diameter, number of threads, and chamfer size with regard to the nominal diameter of thread are set.	* Nominal diameter of thread <u>Common to metric and inch specifications</u> PT1/8 PT1/4 PT3/8 PT1/2 PT3/4 PT1 PT1-1/4 PT1-1/2 PT2	Pre-drilled diameter D1 Number of threads F (per one inch) Chamfer size D2
7	PIPE THREAD (PS) * Pre-drilled diameter, number of threads, and chamfer size with regard to the nominal diameter of thread are set.	* Nominal diameter of thread <u>Common to metric and inch specifications</u> PS1/8 PS1/4 PS3/8 PS1/2 PS3/4 PS1 PS1-1/4 PS1-1/2 PS2	Pre-drilled diameter D1 Number of threads F (per one inch) Chamfer size D2

Item No.	Title/Content	Item/Data	
8	AMERICAN STD. PIPE THREAD (NPT) * Pre-drilled diameter, pitch, and chamfer size with regard to the nominal diameter of thread are set.	* Nominal diameter of thread Common to metric and inch specifications NPT1/8 NPT1/4 NPT3/8 NPT1/2 NPT3/4 NPT1 NPT1-1/4 NPT1-1/2 NPT2	Pre-drilled diameter D1 Number of threads F Chamfer size D2
9	AMERICAN STD. PIPE THREAD (NPS) * Pre-drilled diameter, number of threads, and chamfer size with regard to the nominal diameter of thread are set.	* Nominal diameter of thread Common to metric and inch specifications NPS1/8 NPS1/4 NPS3/8 NPS1/2 NPS3/4 NPS1 NPS1-1/4 NPS1-1/2 NPS2	Pre-drilled diameter D1 Number of threads F (per one inch) Chamfer size D2
10	AMERICAN STD. PIPE THREAD (NPTF) * Pre-drilled diameter, number of threads, and chamfer size with regard to the nominal diameter of thread are set.	* Nominal diameter of thread Common to metric and inch specifications NPTF1/8 NPTF1/4 NPTF3/8 NPTF1/2 NPTF3/4 NPTF1 NPTF1-1/4 NPTF1-1/2 NPTF2	Pre-drilled diameter D1 Number of threads F (per one inch) Chamfer size D2

Item No.	Title/Content	Item/Data	
11	AMERICAN STD. PIPE THREAD (NPSF) * Pre-drilled diameter, number of threads, and chamfer size with regard to the nominal diameter of thread are set.	* Nominal diameter of thread Common to metric and inch specifications NPSF1/8 NPSF1/4 NPSF3/8 NPSF1/2 NPSF3/4 NPSF1 NPSF1-1/4 NPSF1-1/2 NPSF2	Pre-drilled diameter D1 Number of threads F (per one inch) Chamfer size D2

SECTION 3 TOOL DATA SETTING

1. What is the Function of Tool Data Setting?

This is the function to register the information of the tool to be used for cutting.

The tool data mainly consists of following three items:

(1) DATA NO.

A total of 104 tools can be registered for turret A and the data number is used for identifying the individual tools.

For two-saddle models, a total of 38 tools can be registered for turret B.

(2) TOOL CODE

The tools are classified in accordance with the type and direction of cutting. The system has 38 combinations of these items and one code number is assigned to one tool number.

(3) TOOL FORM CODE

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.

* In addition to the above three, TOOL ANGLE, EDGE ANGLE, or HOLDER DIA. is designated as form data.

The one-touch IGF has the function to automatically determine the tool to be used for cutting.

The operator only needs to register the tool data of tools in hand. The one-touch IGF will automatically select a suitable tool from the registered tools. When the tool selected by the one-touch IGF does not meet your requirements, correct the tool data using the condition setting function and the process edit function. If the suitable tool is not registered in the tool data, the one-touch IGF will automatically generate a tool called "non-registered tool".

Input Procedure and Screen Transfer:

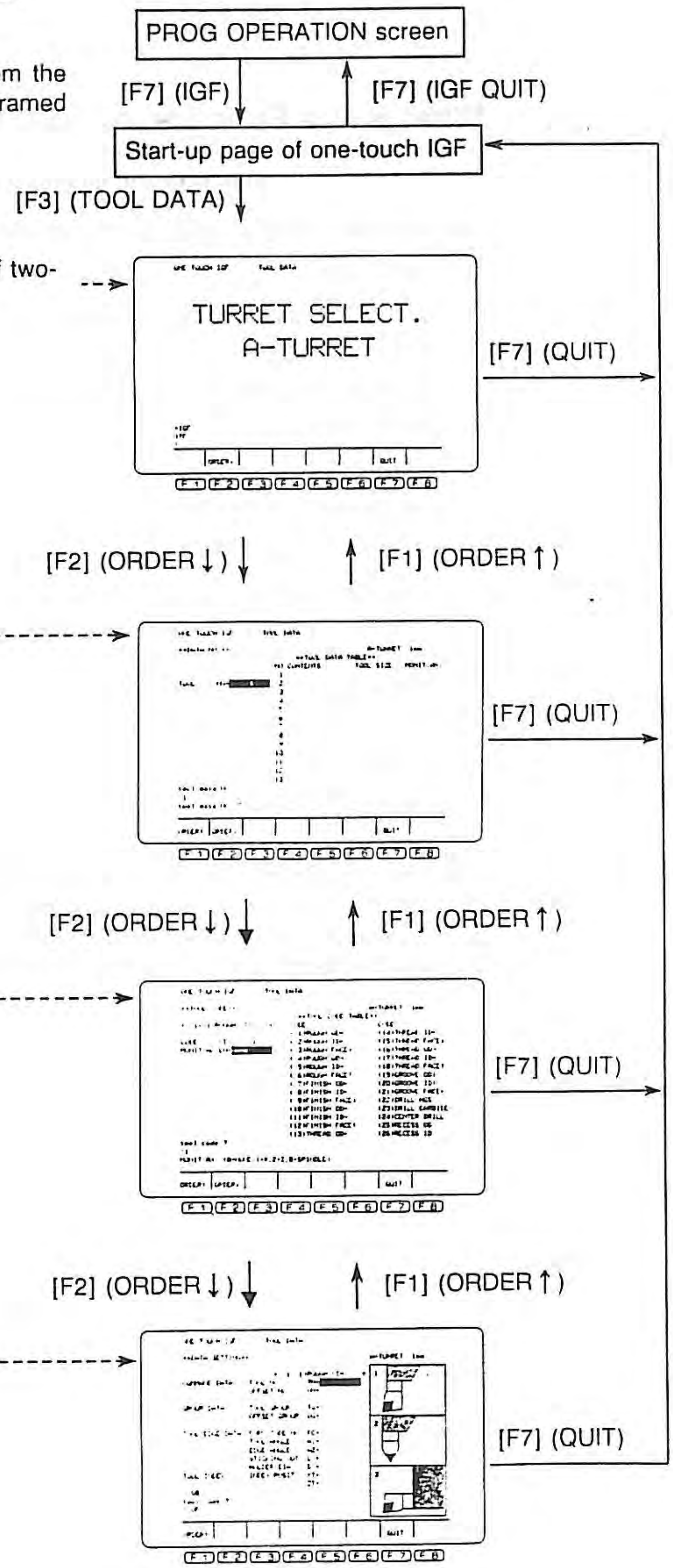
The TOOL DATA screen is called from the PROG OPERATION screen as diagramed to the right.

Designate A- or B-turret in the case of two-saddle models.

Designate the tool data number.

Select the tool code.

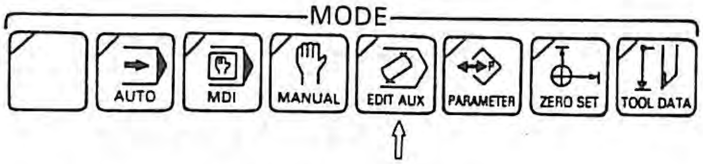
Designate the tool shape data.



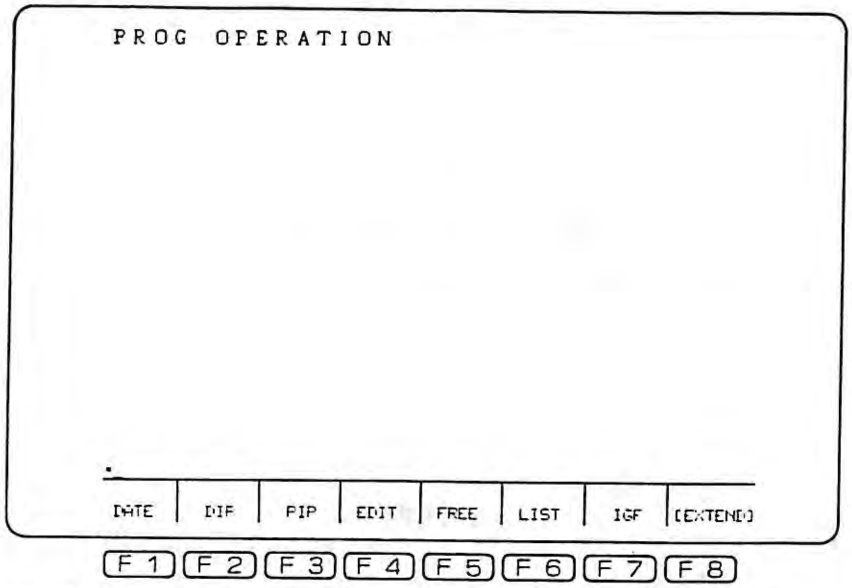
2. Tool Data Setting Procedure

Operating Procedure:

1) Press the **EDIT AUX** key to select the program operation mode.

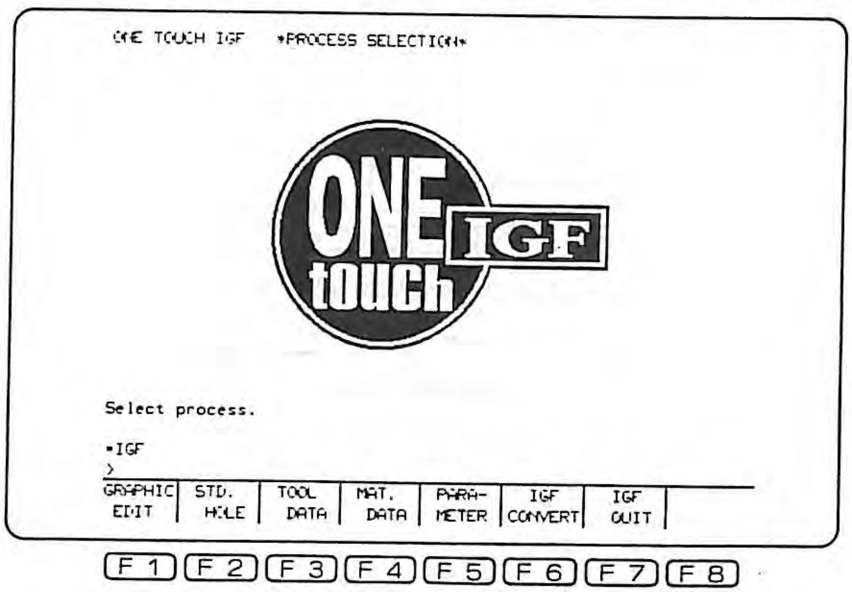


The CRT screen will be changed as shown below.



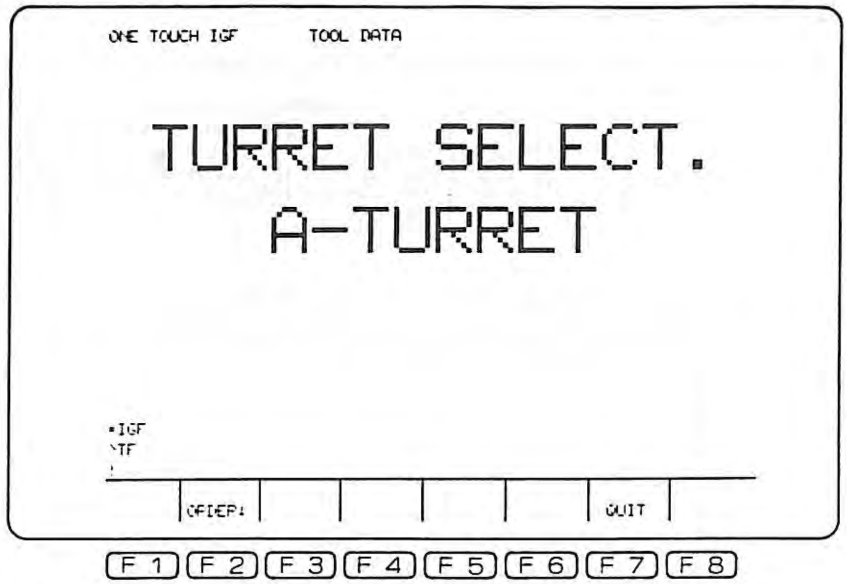
2) Press the function key [F7] (IGF) to establish the one-touch IGF mode.

3) Press the function key [F3] (TOOL DATA) to enable the tool data setting.



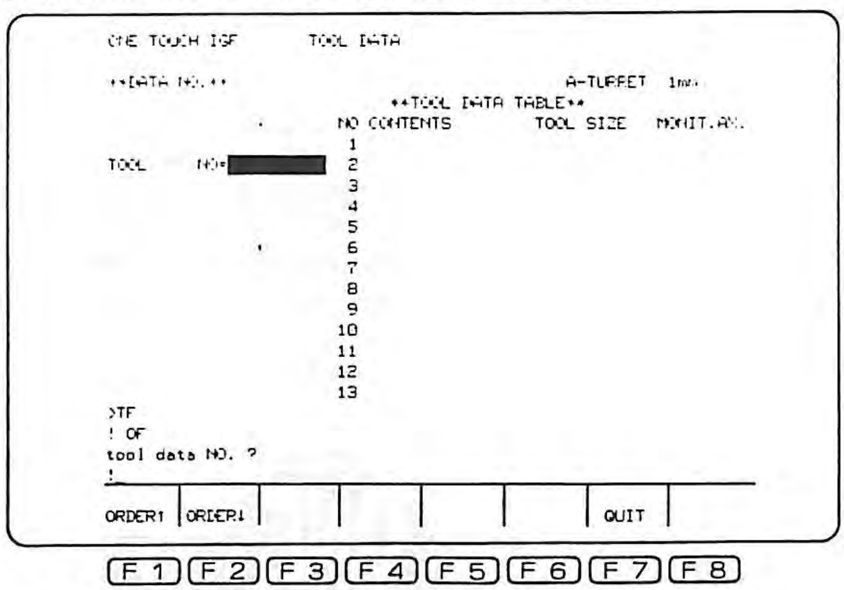
※ In the case of two-saddle models, selection of turret is necessary.

Press the **A** or **B** key on the operation panel, and press the function key [F2] (ORDER ↓).



4) Designate the tool data number.

Designate the number of the tool data table on the screen.



To display tool data numbers from 14 to 104, press the page key.

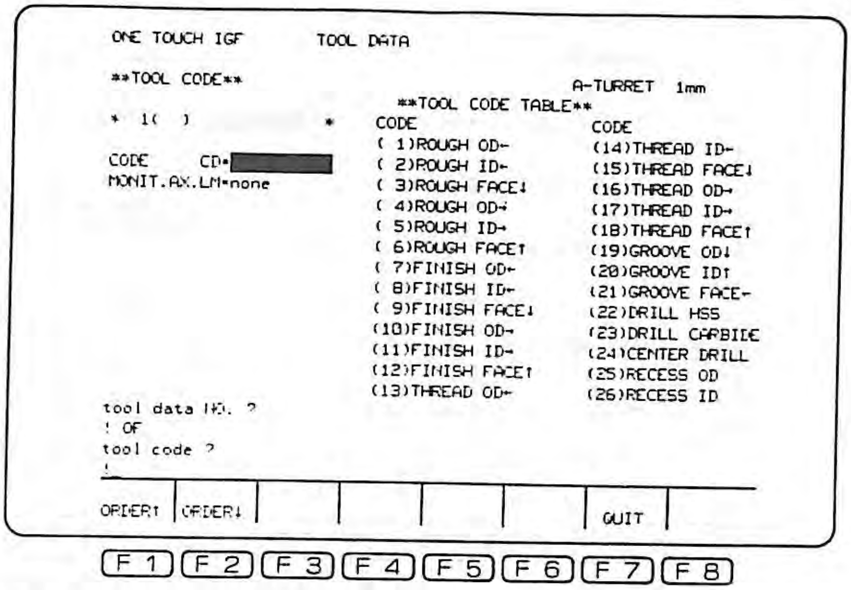
In this example, select tool number "1".

Key in [1] and press the **WRITE** key.

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

6) Set the data for CODE CD.

On the TOOL CODE TABLE, 38 types of tool codes are available and they are displayed on two pages. Press the page key **PAGE ↓**, **PAGE ↑** to display the page covering tool code numbers 27 to 38.



In this example, select (1) ROUGH OD←.

Key in [1] and press the **WRITE** key.

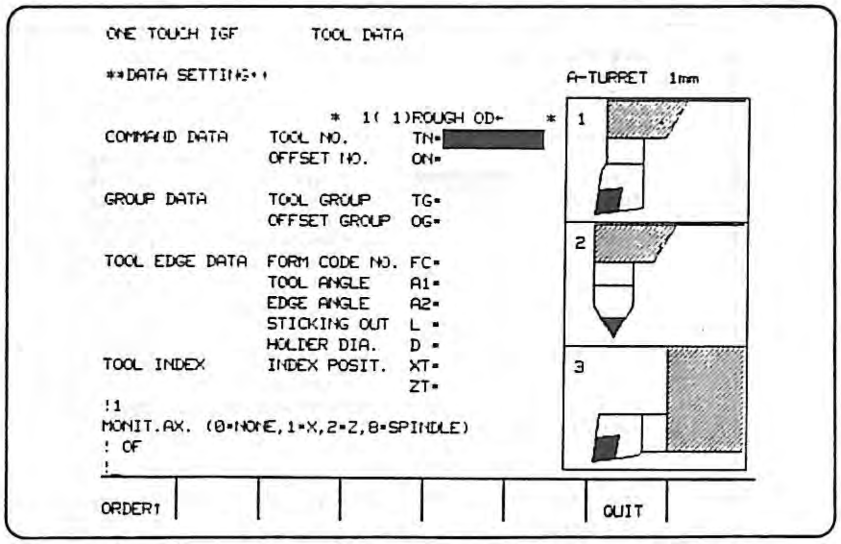
Note: *With the steps above, the tool for "ROUGH OD←" is set to TOOL NO. 1. To set "ROUGH ID←" for TOOL NO. 1, key in [2] and press the* ***WRITE*** *key to select "(2) ROUGH ID←" in step 8).*

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

8) Set the tool data.

Enter the data for TOOL NO., OFFSET NO., and FORM CODE NO.

(For items to be set as the tool data, refer to 3. "Items to be Set as Tool Data".)



F1 F2 F3 F4 F5 F6 F7 F8

Note 1: *Designate the guide drawing number for FORM CODE NO.*

Note 2: *The following values are displayed for the data column of TOOL SIZE on the TOOL DATA TABLE.*

- *Drill/tap tool : tool diameter*
- *Grooving tool : tool width*
- *Other tools : tool edge angle*

9) After the completion of data setting, press the function key [F7] (QUIT). The CRT will return to the start-up page of the one-touch IGF screen.

3. Items to be Set as Tool Data

The following are the items to be set as the tool data.

(1) DATA NO.

A total of 104 tools can be registered for turret A and the data number is used for identifying the individual tools.

For two-saddle models, a total of 38 tools can be registered for turret B.

(2) TOOL CODE

The type of tool is designated. Selection is made from 38 combinations which are classified in accordance with the type and direction of cutting.

(3) COMMAND DATA

TOOL NO. TN

Set the turret position number. This is identical to the T command in the part program.

Set the data for the tool which is used with its tool number designated.

OFFSET NO. ON

Set the tool offset number. This is identical to the T command in the part program.

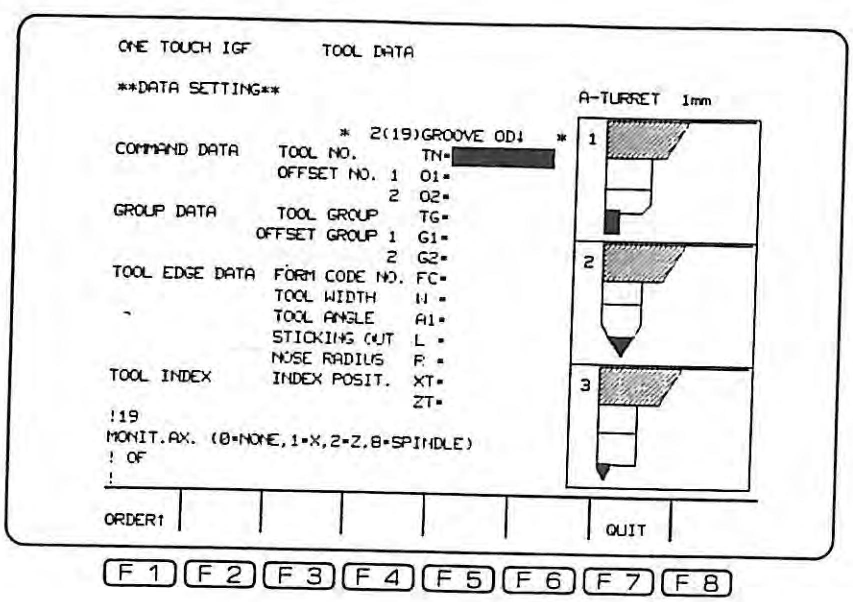
Set the data for the tool which is used with its tool offset number designated.

Note: *OFFSET NO. for (19) GROOVE OD↓, (20) GROOVE ID↑, (21) GROOVE FACE←*

When any of (19) GROOVE OD↓, (20) GROOVE ID↑, and (21) GROOVE FACE← is selected for TOOL NO., the following will be displayed for OFFSET NO. of COMMAND DATA.

OFFSET NO.1 O1 =
2 O2 =

This is to simplify positioning of the grooving tool using two sets of tool offset data and to cut a wide groove.



The Relationship between the Shape of Groove and Tool Offset O1, O2

<u>Single groove cutting</u>

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 (See Fig. 3-1). Positioning of the grooving tool is carried out with the indicated edge taken as the reference and the grove position is determined by this positioning.

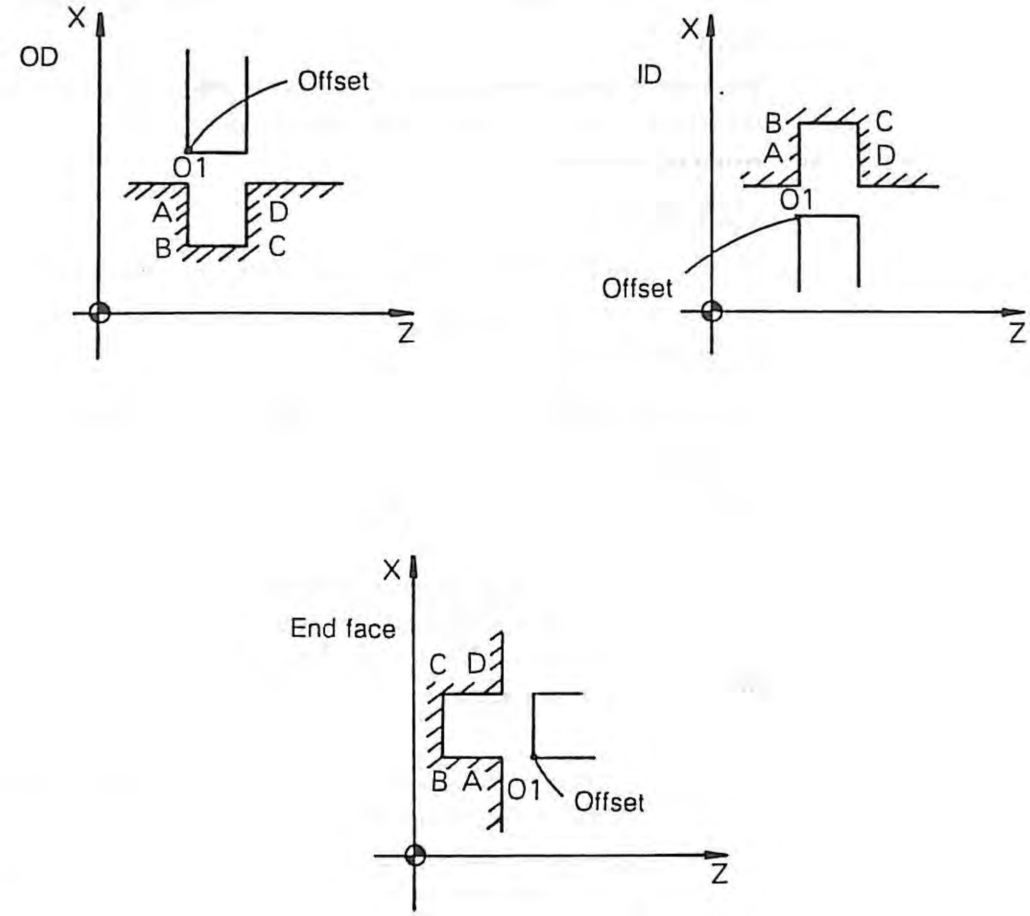


Fig. 3-1

For example, positioning at the reference point is carried out by OFFSET NO. 1 (O1) as shown in Fig. 3-1. If such offset is not carried out, the shape machined can not be accurate.

Wide groove cutting

Offset values used for positioning at the reference point for wide groove cut should be determined in reference to the figures below depending on OD, ID, and face grooving operations. (See Fig. 3-2)

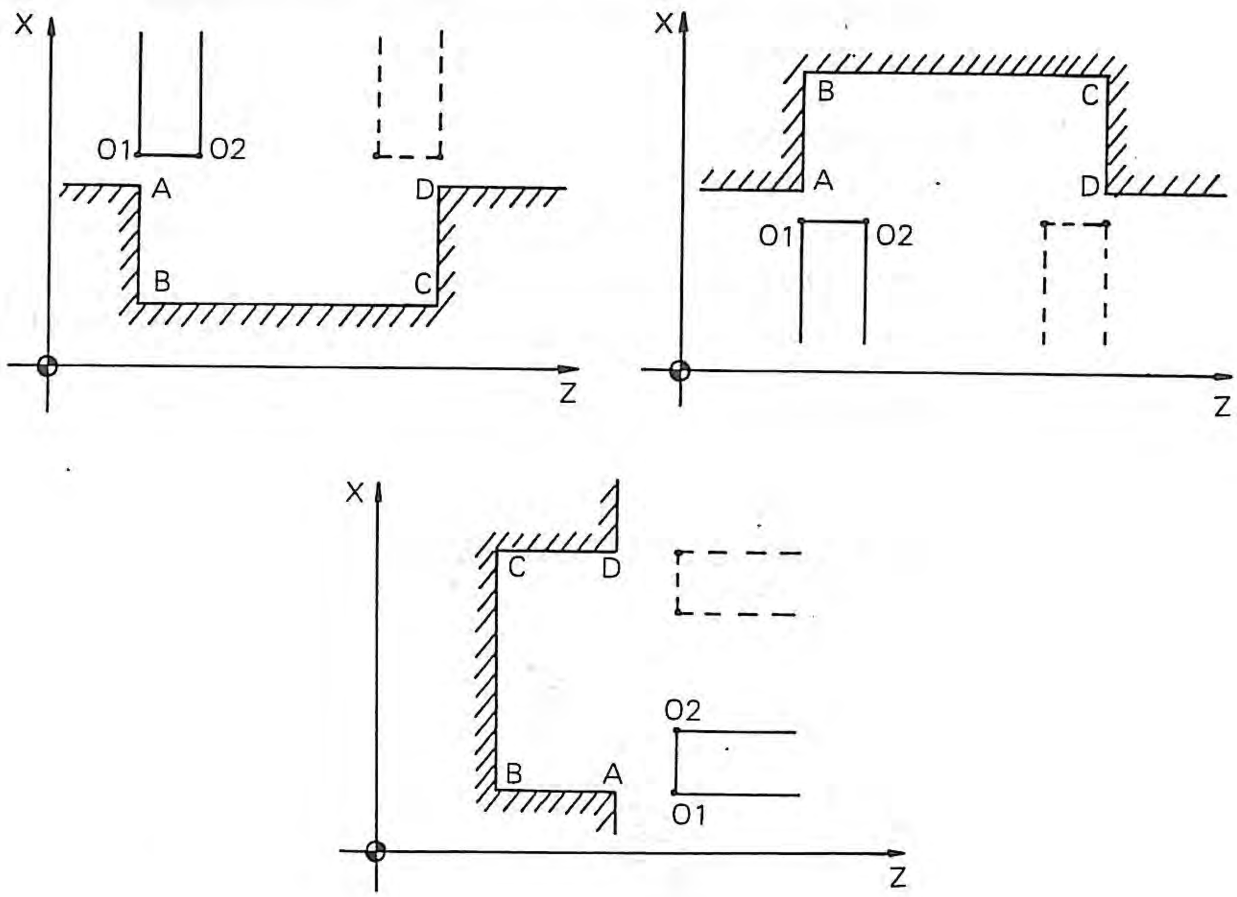


Fig. 3-2

OFFSET NO. 1 O1 =
2 O2 =

Positioning at reference point side (A-B side) for wide groove cut is carried out using the value set at OFFSET NO. 1 (O1), and wide grooving operation is started.

Positioning at end point side (C-D side) is carried out using the value set at OFFSET NO. 2 (O2).

(4) GROUP DATA (displayed when the tool life management specification is selected)

- TOOL GROUP TG

  Set the tool group number of the tool life management specification.

- OFFSET GROUP OG

  Set the offset group number of the tool life management specification.

Refer to OSP5020L Special Functions Manual (Ⅱ) for details of the tool life management function.

(5) TOOL EDGE DATA

- FORM CODE NO. FC

  Set the appropriate number of guide drawings displayed on the screen.

- TOOL ANGLE, EDGE ANGLE, STICKING OUT, etc.

  The data input items meeting the type of tool to be used will be displayed. Enter the data accordingly.

For items to be set as shape data, refer to the table on the next page.

(6) TOOL INDEX

- INDEX POSIT. XT, ZT

  Set the turret position number when changing the cutting tool.

Table 3-1 TOOL CODE (CD) and TOOL EDGE DATA

	Tool Code	Details of shape data to be set
(1)	ROUGH OD←	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 STICKING OUT : L HOLDER DIA. : D
(2)	ROUGH ID←	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 HOLDER LENGTH : L HOLDER DIA. : D
(3)	ROUGH FACE↓	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 STICKING OUT : L HOLDER DIA. : D
(4)	ROUGH OD→	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 STICKING OUT : L HOLDER DIA. : D
(5)	ROUGH ID→	FORM CODE NO. : FC TOOL ANGLE : A1 HOLDER LENGTH : L HOLDER DIA. : D
(6)	ROUGH FACE↑	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 HOLDER LENGTH/ STICKING OUT : L HOLDER DIA. : D
(7)	FINISH OD←	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 STICKING OUT : L HOLDER DIA. : D
(8)	FINISH ID←	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 HOLDER LENGTH : L HOLDER DIA. : D
(9)	FINISH FACE↓	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 STICKING OUT : L HOLDER DIA. : D
(10)	FINISH OD→	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 STICKING OUT : L HOLDER DIA. : D
(11)	FINISH ID→	FORM CODE NO. : FC TOOL ANGLE : A1 HOLDER LENGTH : L2 HOLDER DIA. : D
(12)	FINISH FACE↑	FORM CODE NO. : FC TOOL ANGLE : A1 EDGE ANGLE : A2 HOLDER LENGTH/ STICKING OUT : L HOLDER DIA. : D
(13)	THREAD OD←	FORM CODE NO. : FC TOOL ANGLE : A1 STICKING OUT : L
(14)	THREAD ID←	FORM CODE NO. : FC TOOL ANGLE : A1 HOLDER LENGTH : L HOLDER DIA. : D

Tool Code		Details of shape data to be set
(15)	THREAD FACE ↓	FORM CODE NO. : FC
		TOOL ANGLE : A1
		STICKING OUT : L
(16)	THREAD OD→	FORM CODE NO. : FC
		TOOL ANGLE : A1
		STICKING OUT : L
(17)	THREAD ID→	FORM CODE NO. : FC
		TOOL ANGLE : A1
		DRILL LENGTH : L
		DRILL DIA. : D
(18)	THREAD FACE ↑	FORM CODE NO. : FC
		TOOL ANGLE : A1
		STICKING OUT : L
(19)	GROOVE OD ↓	FORM CODE NO. : FC
		TOOL WIDTH : W
		TOOL ANGLE : A1
		STICKING OUT : L
		NOSE RADIUS : R
(20)	GROOVE ID ↑	FORM CODE NO. : FC
		TOOL WIDTH : W
		HOLDER LENGTH : L
		HOLDER DIA. : D
		NOSE RADIUS : R
(21)	GROOVE FACE←	FORM CODE NO. : FC
		TOOL WIDTH : W
		STICKING OUT : L
		NOSE RADIUS : R
(22)	DRILL HSS	FORM CODE NO. : FC
		TOOL ANGLE : A1
		DRILL LENGTH : L
		DRILL DIA. : D
(23)	DRILL CARBIDE	FORM CODE NO. : FC
		TOOL ANGLE : A1
		DRILL LENGTH : L
		DRILL DIA. : D

Tool Code		Details of shape data to be set
(24)	CENTER DRILL	FORM CODE NO. : FC
(25)	RECESS OD	FORM CODE NO. : FC
		TOOL ANGLE : A1
		EDGE ANGLE : A2
		STICKING OUT : L
		HOLDER DIA. : D
(26)	RECESS ID	FORM CODE NO. : FC
		TOOL ANGLE : A1
		EDGE ANGLE : A2
		HOLDER LENGTH : L
		HOLDER DIA. : D
(27)	M-DRILL ↓	FORM CODE NO. : FC
		TOOL ANGLE : A1
		DRILL LENGTH : L
		DRILL DIA. : D
(28)	M-DRILL←	FORM CODE NO. : FC
		TOOL ANGLE : A1
		DRILL LENGTH : L
		DRILL DIA. : D
(29)	BORING ↓	FORM CODE NO. : FC
		BORING LENGTH : L
		BORING DIA. : D
(30)	BORING←	FORM CODE NO. : FC
		BORING LENGTH : L
		BORING DIA. : D
(31)	TAP ↓	FORM CODE NO. : FC
		TAPING LENGTH : L
		TAPING KIND : No.
		NOMINAL DIA. : D
(32)	TAP←	FORM CODE NO. : FC
		TAPING LENGTH : L
		TAPING KIND : No.
		NOMINAL DIA. : D

Tool Code		Details of shape data to be set
(33)	REAMER ↓	FORM CODE NO. : FC REAMER LENGTH : L REAMER DIA. : D
(34)	REAMER←	FORM CODE NO. : FC REAMER LENGTH : L REAMER DIA. : D
(35)	END MILL ↓	FORM CODE NO. : FC END MILL LENGTH : L END MILL DIA. : D NO. OF FLUTE : N
(36)	END MILL←	FORM CODE NO. : FC END MILL LENGTH : L END MILL DIA. : D NO. OF FLUTE : N
(37)	MILLING ↓	FORM CODE NO. : FC MILL LENGTH : L MILLING DIA. : D NO. OF TOOTH : N
(38)	MILLING←	FORM CODE NO. : FC MILL LENGTH : L MILLING DIA. : D NO. OF TOOTH : N

Supplement 1:

1) TOOL ANGLE (A1) and EDGE ANGLE (A2) of OD, ID, or end face cutting tool are defined as follows:

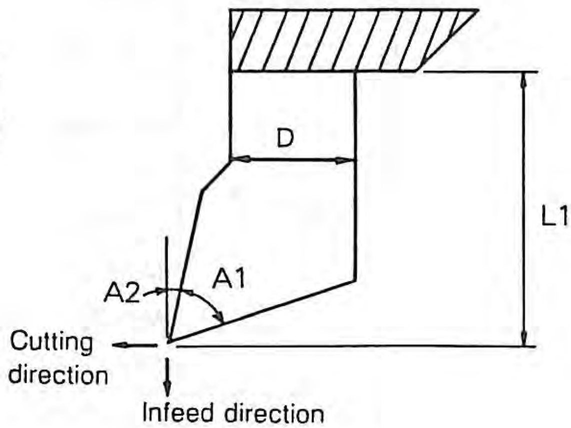
   TOOL ANGLE (A1) : Angle of cutting tool tip or insert tip
   EDGE ANGLE (A2) : An angle made between normal to cutting direction and cutting edge.
   (There are cases where no designation is made.)

2) STICKING OUT (L1) of ID cutting tool is defined as follows:

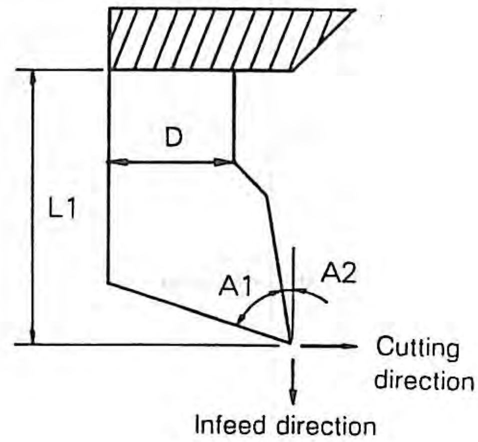
   STICKING OUT (L1) : Length between tool tip and tool holder.
   HOLDER LENGTH (L2): ID cutting tool length or drill length
   TOOL DIA. (D) : Holder diameter
   TOOL WIDTH (W) : Width of tool

Example 1: Normal turning tool (roughing, finishing)

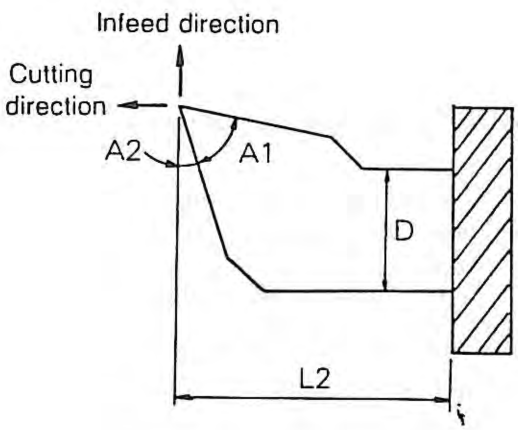
OD ← (1), FACE ↓ (1)



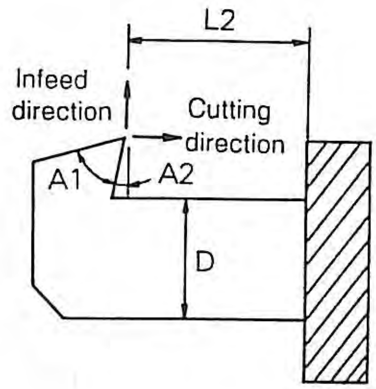
FACE ↓ (4), OD → (1)



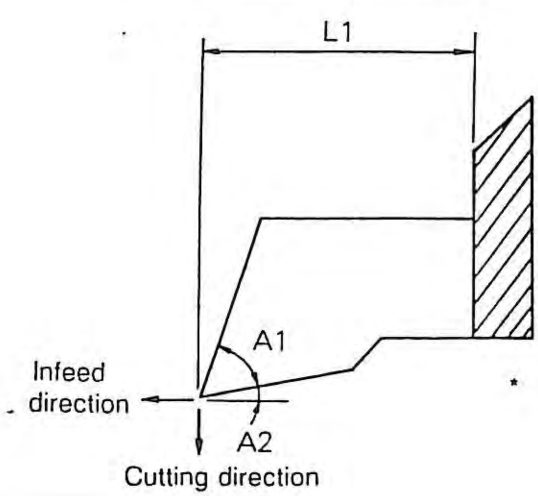
ID ← (1) (2), FACE ↑ (1)



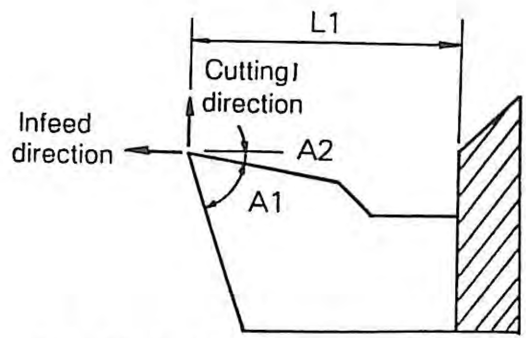
ID → (1)



OD ← (3), FACE ↓ (2)



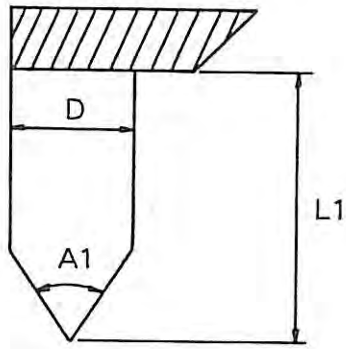
FACE ↑ (2)



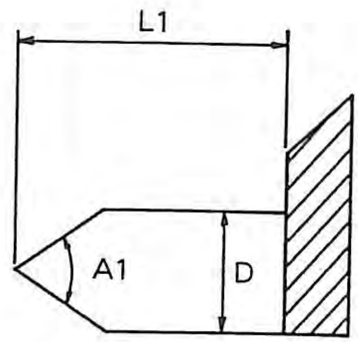
* Numbers in ( ) indicate the shape code numbers of the shape data of each individual tool.

Example 2: No designation is made for EDGE ANGLE

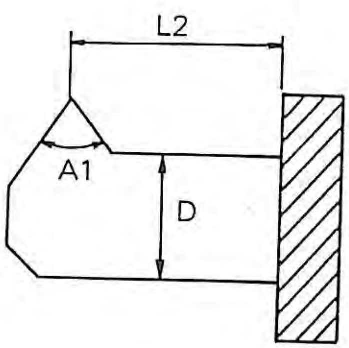
OD ← (2), OD → (2)



FACE ↓ (3), FACE ↑ (3)



ID ← (3), ID → (2)

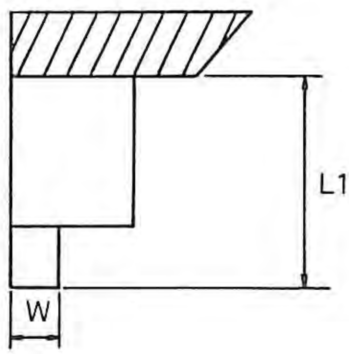


* Numbers in ( ) indicate the shape code numbers of the shape data of each individual tool.

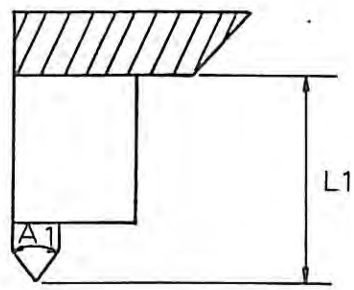
Note: *The cutting tools above without EDGE ANGLE designation are used for displaying the tool shape, and not selected in the automatic process determination. Re-set the tool data for these tools in process edit operation in case they are used.*

Example 3: Groove, thread, drill

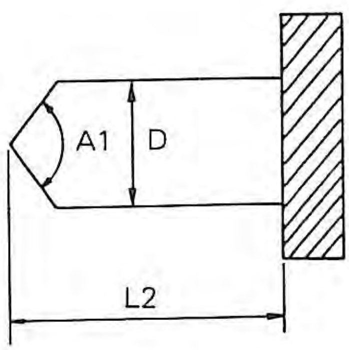
Groove



Thread



Drill



W: Drill width

For the grooving tool, set arbitrary angular data.

Note: *The automatic process determination function does not select the following tools even if they are registered in the tool data.*

- *A drill whose diameter is larger than the value set at the dimension parameter No. 37 DRILL MAXIMUM DIAMETER.*
- *A drill whose diameter is smaller than the value set at the dimension parameter No. 36 CARBIDE DRILL MINIMUM DIAMETER.*
- *A grooving tool whose width is larger than the value set at the dimension parameter No. 25 GROOVING TOOL WIDTH MAX. VALVE.*
- *A grooving tools with shape code No. 2 (groove OD ↓, groove ID ↑, groove END FACE ←) and shape code No. 3 (groove OD ↓). These are used only to display the tool path.*

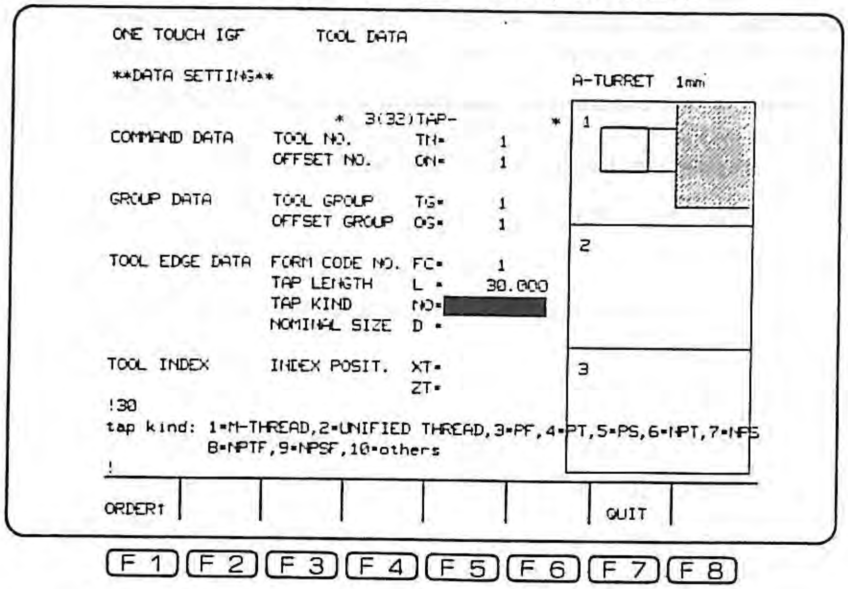
Supplement 2:

When (31) TAP↓ or (32) TAP← is selected, enter the data for TAP KIND NO and NOMINAL SIZE D according to the prompt displayed on the screen.

Example: To enter the shape data of M8 tap

1) The following prompt will be displayed when the cursor is moved to the data column of TAP KIND NO = .

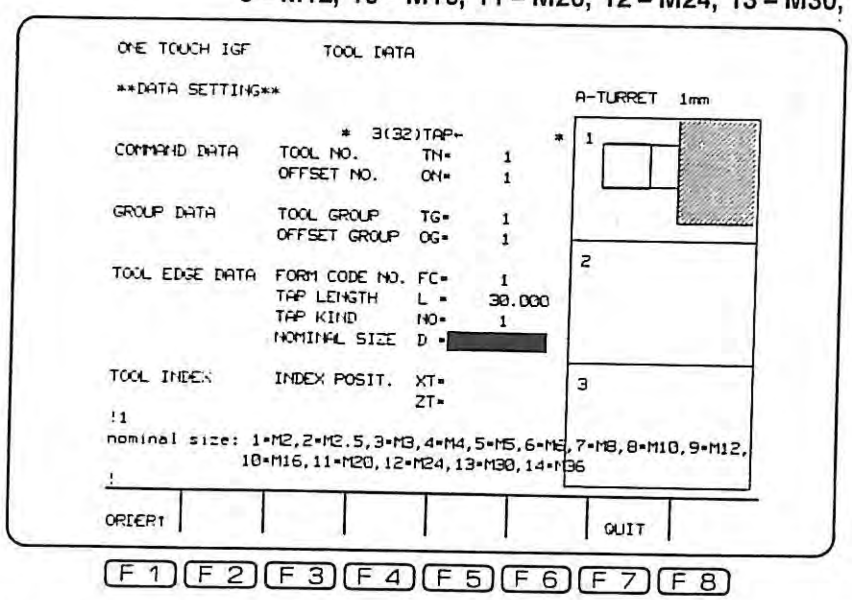
tap kind: 1 = M-THREAD, 2 = UNIFIED THREAD, 3 = PF, 4 = PT, 5 = PS, 6 = NPT, 7 = NPS, 8 = NPTF, 9 = NPSF, 10 = others



2) Key in [1] and press the **WRITE** key.

3) The cursor will move to the data column of NOMINAL SIZE D = and the following prompt will be displayed.

nominal size: 1 = M2, 2 = M2.5, 3 = M3, 4 = M4, 5 = M5, 6 = M6, 7 = M8, 8 = M10, 9 = M12, 10 = M16, 11 = M20, 12 = M24, 13 = M30, 14 = M36



4) Key in [7] and press the **WRITE** key. This completes the shape data setting of M8 tap.

SECTION 4 MATERIAL DATA SETTING

1. What is the Function of Material Data Setting?

This is the function to register cutting conditions for each individual blank material.

The one-touch IGF has the function to automatically determine the cutting conditions meeting the blank material designated. A total of 16 types of material data can be registered and 8 types of material data are factory-set by Okuma as standard blank materials.

To create a part program for machining the blank material other than these eight types of blank materials, it is necessary to set the material data before starting with graphic edit operation. Otherwise automatic process determination is disabled.

The material data consists of the following three items:

(1) NAME NO.

A total of 16 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 alphanumeric characters.

(3) CUTTING CONDITIONS

Cutting conditions for individual tools are registered. This includes CUT. SPEED or CUT. DEPTH.

Input Procedure and Screen Transfer:

The MAT. DATA screen is called from the PROG OPERATION screen as diagramed to the right.

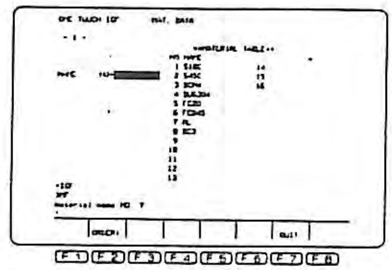
PROG OPERATION screen

[F7] (IGF) ↑ ↓ [F7] (IGF QUIT)

Start-up page of one-touch IGF

[F4] (MAT. DATA) ↓

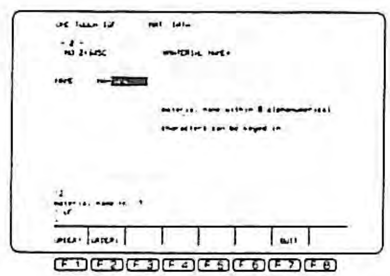
Select the material number from the MATERIAL TABLE. ----→



[F7] (QUIT) →

[F2] (ORDER ↓) ↑ ↓ [F1] (ORDER ↑)

Enter the name of material as needed. ----→

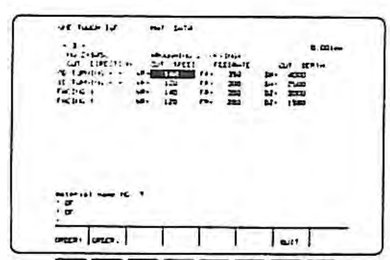


[F7] (QUIT) →

[F2] (ORDER ↓) ↑ ↓ [F1] (ORDER ↑)

Various cutting conditions are displayed. ----→

Change the screen by pressing the function keys [F1] (ORDER ↑) and [F2] (ORDER ↓) and set the necessary data.



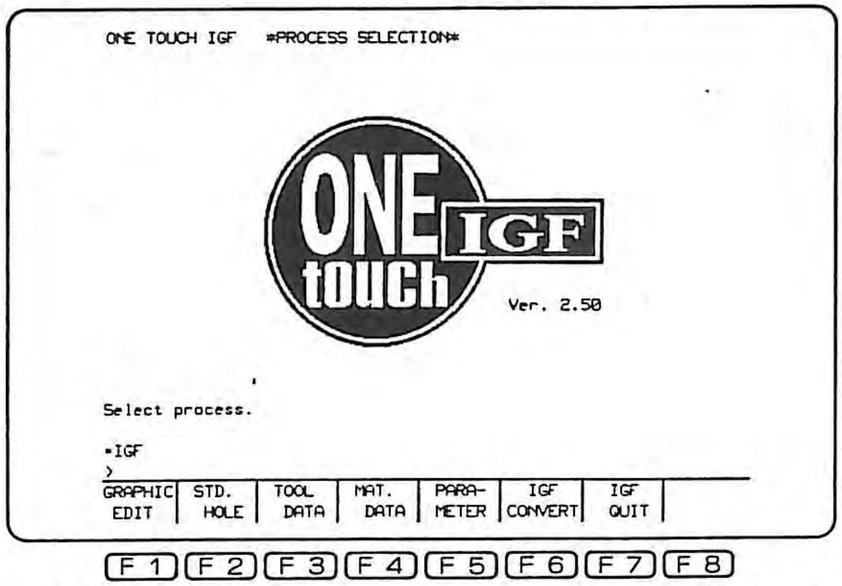
[F7] (QUIT)

2. Material Data Setting Procedure

The procedure to newly set the material data and the procedure to correct the material data are explained below. To newly set the material data, first copy the material data which is already registered in the memory and correct it as needed.

Operating Procedure:

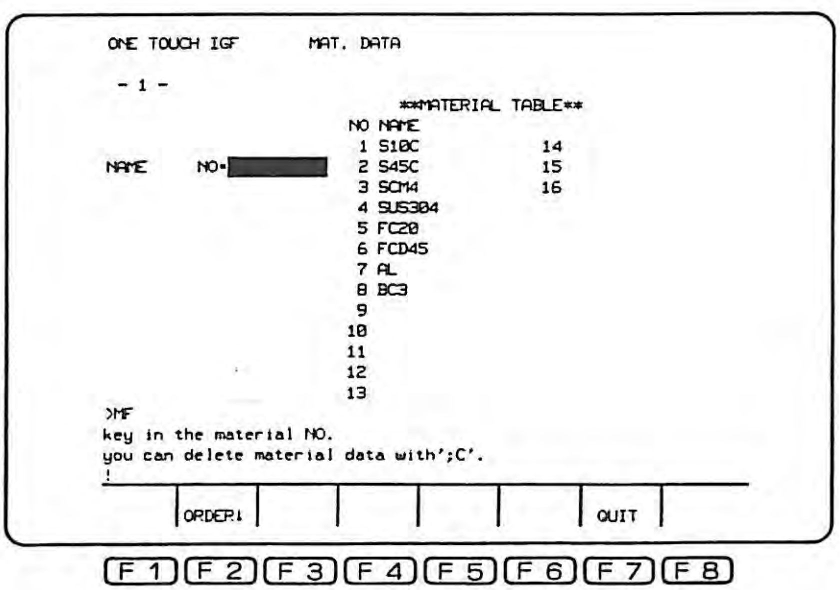
1) Press the function key [F4] (MAT. DATA) on the start-up page of the one-touch IGF screen.



2) Select the material number from No. 1 to No. 16 on the MATERIAL TABLE.

In this example, the material data "S45C" is copied and modified to register "S55C" as new material data under the material No. 9.

Key in "9" and press the **WRITE** key.

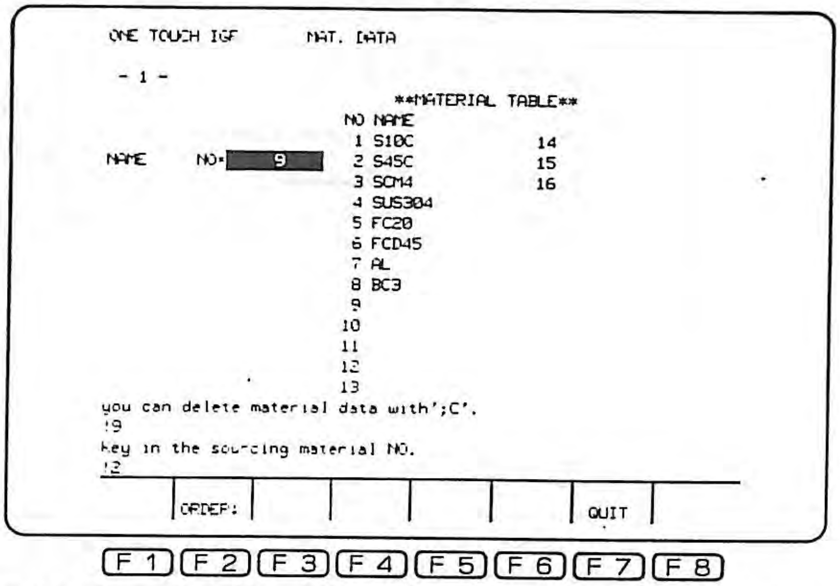


3) The following message will be displayed:

   //key in the sourcing material No.//

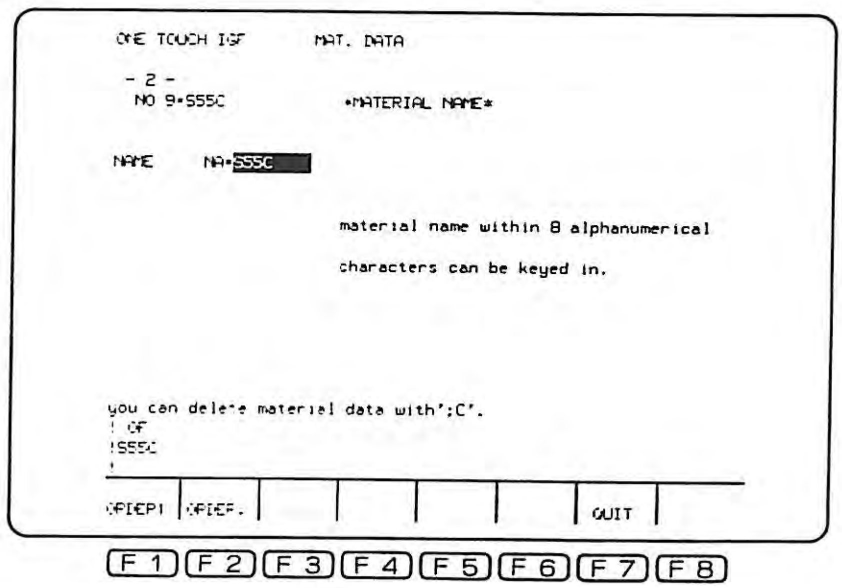
   Key in "2" and press the **WRITE** key.

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



5) Input the material name by pressing [S], [5], [5], and [C] in this order.

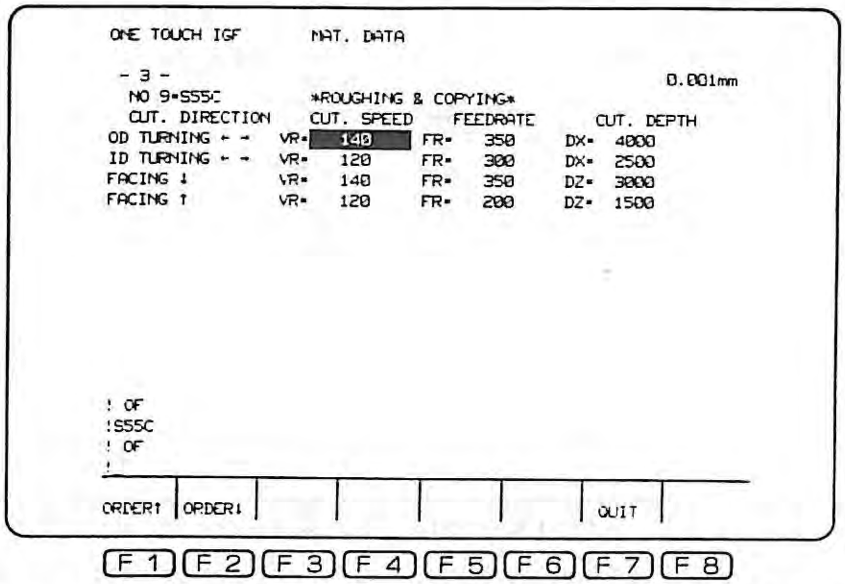
6) Press the function key [F2] (ORDER ↓).



7) Set the cutting conditions.

In this example, the cutting conditions for S55C are set by modifying the cutting conditions for S45C.

Move the cursor to the required data column using the cursor control keys ↑, ↓, →, and ←, and input the cutting condition accordingly. To switch the display page, use the function keys [F1] (ORDER ↑) and [F2] (ORDER ↓).

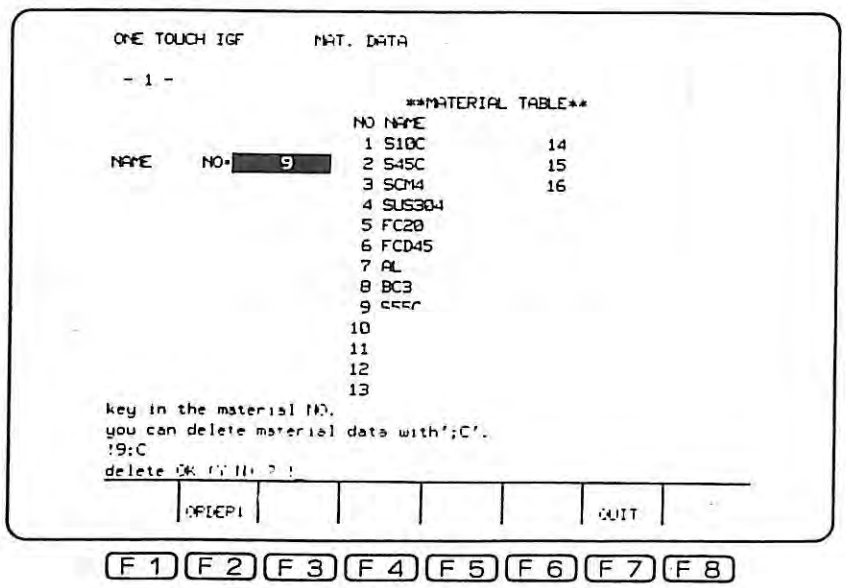


8) After the cutting condition is designated, press the function key [F7] (QUIT). The CRT will return to the start-up page of the one-touch IGF screen.

Supplement:

To modify the material data which is already registered, copy the source data as explained in step 3) above. After the completion of step 2), advance to step 4).

To delete the material data which is already registered, key in the material number, ";", and "C" in this order and press the **WRITE** key. As the prompt "delete OK (Y/N)?" will displayed on the screen, press "Y". This will delete the designated material data.



3. Items to be Set as Material Data

The following data is factory-set by Okuma as standard material data.

Table 4-1

Material Number (NAME NO)	Material
1	S10C
2	S45C
3	SCM4
4	SUS304
5	FC20
6	FCD45
7	AL
8	BC3

* No material data is registered for numbers 9 to 16.

To create a program for machining the material other than the above eight materials, the material data must be set.

Material data is registered according to the table on the following pages. For these data, values ranging from 0 to 64,000 can be set. (decimal point input is not allowed.)

For material and cutting conditions, only typical data is factory-set by Okuma. So there are cases where the data cannot be used as it is.

Note 1: *To set the cutting conditions, press the function keys [F1] (ORDER ↑) and [F2] (ORDER ↓) to change the screen.*

Note 2: *For cutting process Nos. 14 (M-drill cutting) to 22 (contour generating machining), no screen display is given unless the multi-machining specification is selected.*

Note 3: *Screen display is made in the unit system selected for the NC. For the procedure to change the unit system, refer to the OSP5020L Operation Manual.*

Cutting Process	Display Item	Cutting Conditions	Unit System	
			Metric	Inch
1. ROUGHING & COPYING	Common to metric and inch specifications: OD TURNING ID TURNING FACING FACING	CUT. SPEED VR FEEDRATE FR CUT. DEPTH DX	m/min μm/rev μm	feet/min 1/10000 inch/rev 1/10000 inch
2. FINISHING	Common to metric and inch specifications: —	CUTTING SPEED VF FINISHING STOCK X-AXIS LX* FINISHING STOCK Z-AXIS LZ* FINISH__(ROUGH) FEEDRATE F1 = FINISH__(SEMI FINISH) FEEDRATE F2 = FINISH__(FINISH) FEEDRATE F3 = FINISH__(FINE FINISH) FEEDRATE F4 =	m/min μm μm μm/rev μm/rev μm/rev μm/rev	feet/min 1/10000 inch 1/10000 inch 1/10000 inch/rev 1/10000 inch/rev 1/10000 inch/rev 1/10000 inch/rev
3. THREADING	Metric: - THREAD PITCH 1.0 mm OR LESS 1.5 mm OR LESS 2.0 mm OR LESS 2.5 mm OR LESS 3.0 mm OR LESS 4.0 mm OR LESS 5.0 mm OR LESS 5.0 mm OR MORE	CUT. SPEED V CUT. DEPTH D FIN. STOCK L	m/min μm μm	— — —

Cutting Process	Display Item	Cutting Conditions	Unit System	
			Metric	Inch
3. THREADING	Inch:	CUT. SPEED V	—	feet/min
	- THREAD FER INCH	CUT. DEPTH D	—	1/10000 inch
	24 OR MORE	FIN. STOCK L	—	1/10000 inch
	16 OR MORE			
	12 OR MORE			
	10 OR MORE			
	8 OR MORE			
	6 OR MORE			
	5 OR MORE			
	4 OR LESS			
4. TAPPING	Common to metric and inch specifications: —	CUTTING SPEED V	m/min	feet/min
5. GROOVING	Metric:	CUT. SPEED VR	m/min	—
	- TOOL WIDTH	FEEDRATE FR	μm/rev	—
	3 mm OR LESS	CUT. DEPTH D	μm	—
	5 mm OR LESS			
	8 mm OR LESS			
	12 mm OR LESS			
	20 mm OR LESS			
	20 mm OR MORE			
	Inch:	CUT. SPEED VR	—	feet/min
	- TOOL WIDTH (inch)	FEEDRATE FR	—	1/10000 inch/rev
	0.125 OR LESS			
	0.189 OR LESS	CUT. DEPTH D	—	1/10000 inch
	0.312 OR LESS			
	0.472 OR LESS			
	0.787 OR LESS			
	0.787 OR MORE			

Cutting Process	Display Item	Cutting Conditions	Unit System	
			Metric	Inch
6. GROOVE FINISHING	Common to metric and inch specifications:	CUTTING SPEED VF	m/min	feet/min
		FINISHING STOCK X-AXIS LX*	μm	1/10000 inch
		FINISHING STOCK Z-AXIS LZ*	μm	1/10000 inch
	—	FINISH__(ROUGH) FEEDRATE F1 =	μm/rev	1/10000 inch/rev
		FINISH__(SEMI FINISH) FEEDRATE F2 =	μm/rev	1/10000 inch/rev
		FINISH__(FINISH) FEEDRATE F3 =	μm/rev	1/10000 inch/rev
		FINISH__(FINE FINISH) FEEDRATE F4 =	μm/rev	1/10000 inch/rev
7. DRILLING (HSS)	Metric:	CUT. SPEED V	m/min	—
	- DRILL DIA.	FEEDRATE F	μm/rev	—
	5 mm OR LESS	CUT. DEPTH D	μm	—
	8 mm OR LESS			
	14 mm OR LESS			
	23 mm OR LESS			
	32 mm OR LESS			
	50 mm OR LESS			
	50 mm OR MORE			
	Inch:	CUT. SPEED V	—	feet/min
	- DRILL DIA. (inch)	FEEDRATE F	—	1/10000 inch/rev
	3/16 OR LESS			
	5/16 OR LESS	CUT. DEPTH D	—	1/10000 inch
	9/16 OR LESS			
	15/16 OR LESS			
	1-5/16 OR LESS			
	1-15/16 OR LESS			
	1-15/16 OR MORE			

Cutting Process	Display Item	Cutting Conditions	Unit System	
			Metric	Inch
8. DRILLING (CARBIDE)	Metric: - DRILL DIA. 30 mm OR LESS 40 mm OR LESS 50 mm OR LESS 60 mm OR LESS 60 mm OR MORE	CUT. SPEED V FEEDRATE F CUT. DEPTH D	m/min μm/rev μm	— — —
	Inch: - DRILL DIA. (inch) 1-3/16 OR LESS 1-9/16 OR LESS 1-15/16 OR LESS 2-3/8 OR LESS 2-3/8 OR MORE	CUT. SPEED V FEEDRATE F CUT. DEPTH D	— — —	feet/min 1/10000 inch/rev 1/10000 inch
9. DRILLING (POSITION-ING)	Common to metric and SPINDLE inch specifications: —	SPINDLE SPEED S FEEDRATE F CUT. DEPTH D	rpm μm/rev μm	rpm 1/10000 inch/rev 1/10000 inch
10. CENTERING	Common to metric and SPINDLE inch specifications: —	SPINDLE SPEED S FEEDRATE F	rpm μm/rev	rpm 1/10000 inch/rev
11. RECESSING	Common to metric and SPINDLE inch specifications: —	SPINDLE SPEED S FEEDRATE F	m/min μm/rev	feet/min 1/10000 inch/rev
12. END MILL ROUGHING / M END MILL	Metric: - TOOL DIAM. 5 mm OR LESS 8 mm OR LESS 14 mm OR LESS 23 mm OR LESS 32 mm OR LESS 50 mm OR LESS 50 mm OR MORE	CUT. SPEED V. FEEDRATE F CUT. DEPTH D	m/min μm//flute μm	— — —

Cutting Process	Display Item	Cutting Conditions	Unit System	
			Metric	Inch
12. END MILL ROUGHING / M END MILL	Inch: - TOOL DIAM. 3/16 OR LESS 5/16 OR LESS 9/16 OR LESS 15/16 OR LESS 1-5/16 OR LESS 1-15/16 OR LESS 1-15/16 OR MORE	CUT. SPEED V FEEDRATE F CUT. DEPTH D	— — —	feet/min 1/10000 inch/flute 1/10000 inch
13. END MILL FINISHING	Common to metric and inch specifications: —	CUTTING SPEED VF FINISHING STOCK DIA. LD FINISHING STOCK DEPTH LL FINISH__(ROUGH) FEEDRATE F1 = FINISH__(SEMI FINISH) FEEDRATE F2 = FINISH__(FINISH) FEEDRATE F3 = FINISH__(FINE FINISH) FEEDRATE F4 =	m/min μm μm μm//flute μm//flute μm//flute μm//flute	feet/min 1/10000 inch 1/10000 inch 1/10000 inch/flute 1/10000 inch/flute 1/10000 inch/flute 1/10000 inch/flute
14. M DRILLING (multi-machining specification)	Metric: - DRILL DIA. 5 mm OR LESS 8 mm OR LESS 14 mm OR LESS 23 mm OR LESS 32 mm OR LESS 50 mm OR LESS 50 mm OR MORE	CUT. SPEED V FEEDRATE F CUT. DEPTH D	m/min μm/rev μm	— — —

Cutting Process	Display Item	Cutting Conditions	Unit System	
			Metric	Inch
14. M DRILLING (multi-machining specification)	Inch: - DRILL DIA. (inch) 3/16 OR LESS 5/16 OR LESS 9/16 OR LESS 15/16 OR LESS 1-5/16 OR LESS 1-15/16 OR LESS 1-15/16 OR MORE	CUT. SPEED V FEEDRATE F CUT. DEPTH D	— — —	feet/min 1/10000 inch/rev 1/10000 inch/rev
15. M DRILLING (POSITION-ING) (multi-machining specification)	Common to metric and inch specifications: —	M SPINDLE SPEED SB FEEDRATE F CUT. DEPTH D	rpm μm/rev μm	rpm 1/10000 inch/rev 1/10000 inch
16. M CENTERING (multi-machining specification)	Common to metric and inch specifications: —	M SPINDLE SPEED SB FEEDRATE F	rpm μm/rev	rpm 1/10000 inch/rev
17. M BORING	Common to metric and inch specifications: —	CUTTING SPEED V FEEDRATE F	m/min μm/rev	feet/min 1/10000 inch/rev
18. M BORING FINISH (multi-machining specification)	Common to metric and inch specifications: —	CUTTING SPEED VF FINISHING STOCK L FINISH__(ROUGH) FEEDRATE F1 = FINISH__(SEMI FINISH) FEEDRATE F2 = FINISH__(FINISH) FEEDRATE F3 = FINISH__(FINE FINISH) FEEDRATE F4 =	m/min μm μm/rev μm/rev μm/rev μm/rev	feet/min 1/10000 inch 1/10000 inch/rev 1/10000 inch/rev 1/10000 inch/rev 1/10000 inch/rev
19. M TAPPING (multi-machining specification)	Common to metric and inch specifications: —	CUTTING SPEED V	m/min	feet/min

Cutting Process	Display Item	Cutting Conditions	Unit System	
			Metric	Inch
20. M REAMING (multi-machining specification)	Common to metric and inch specifications: —	CUTTING SPEED V FEEDRATE F FINISHING STOCK L	m/min μm/rev μm	feet/min 1/10000 inch/rev 1/10000 inch
21. GENERATING ROUGHING (multi-machining specification)	Metric: - TOOL DIA. 5 mm OR LESS 8 mm OR LESS 14 mm OR LESS 23 mm OR LESS 32 mm OR LESS 50 mm OR LESS 50 mm OR MORE	CUT. SPEED V FEEDRATE F CUT. DEPTH D	m/min μm/rev μm	— — —
	Inch: - TOOL DIA. 3/16 OR LESS 5/16 OR LESS 9/16 OR LESS 15/16 OR LESS 1-5/16 OR LESS 1-15/16 OR LESS 1-15/16 OR MORE	CUT. SPEED V FEEDRATE F CUT. DEPTH D	— — —	feet/min 1/10000 inch/rev 1/10000 inch
22. GENERATING FINISHING (multi-machining specification)	Common to metric and inch specifications)	CUTTING SPEED VF FINISHING STOCK DIA. LD FINISH_(ROUGH) FEEDRATE F1 = FINISH_(SEMI FINISH) FEEDRATE F2 = FINISH_(FINISH) FEEDRATE F3 = FINISH_(FINE FINISH) FEEDRATE F4 =	m/min μm μm//flute μm//flute μm//flute μm//flute	feet/min 1/10000 inch 1/10000 inch/flute 1/10000 inch/flute 1/10000 inch/flute 1/10000 inch/flute

Supplement:

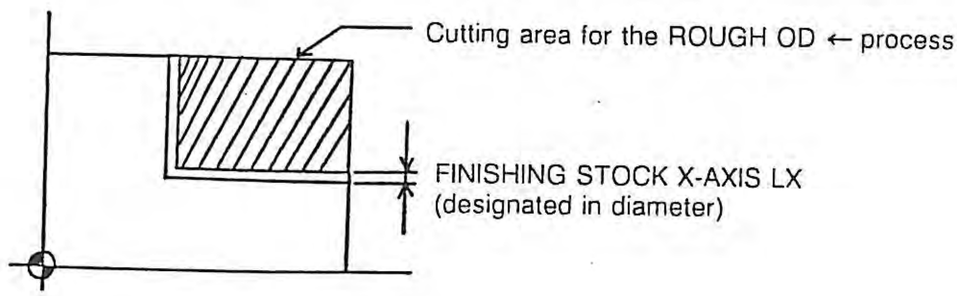
To enable automatic process determination, enter the value for FINISHING STOCK Z-AXIS or FINISHING STOCK X-AXIS to designate the finishing stock for the machining processes indicated below:

FINISHING STOCK X-AXIS

ROUGH OD ←, ROUGH ID ←, ROUGH M. OD ←, ROUGH M. ID ←, ROUGH M. OD →, ROUGH M. ID →, ROUGH OD →, ROUGH ID →, COPY OD ←, COPY ID ←, COPY M. OD ←, COPY M. ID ←, COPY M. OD →, COPY M. ID →, COPY OD →, COPY ID →

FINISHING STOCK Z-AXIS

RUGH O. FACE ↓, RUGH I. FACE ↓, RUGH M. FACE ↓, RUGH M. FACE ↑; COPY O. FACE ↓, COPY I. FACE ↑, COPY M. FACE ↓, COPY M. FACE ↑



SECTION 5 ONE-TOUCH IGF PARAMETER SETTING

1. What is the Function of an IGF Parameter?

With the one-touch IGF, the tools to be used, the cutting conditions, and the machining process are determined automatically; part programs are created by merely entering shape definition data and setup data. However, there are cases where may automatically determined conditions may need to be changed. In such cases, change the setting of the one-touch IGF parameter as necessary.

Setting the one-touch IGF parameter is also necessary to output the various special M codes to the part program.

The following parameters are one-touch IGF parameters.

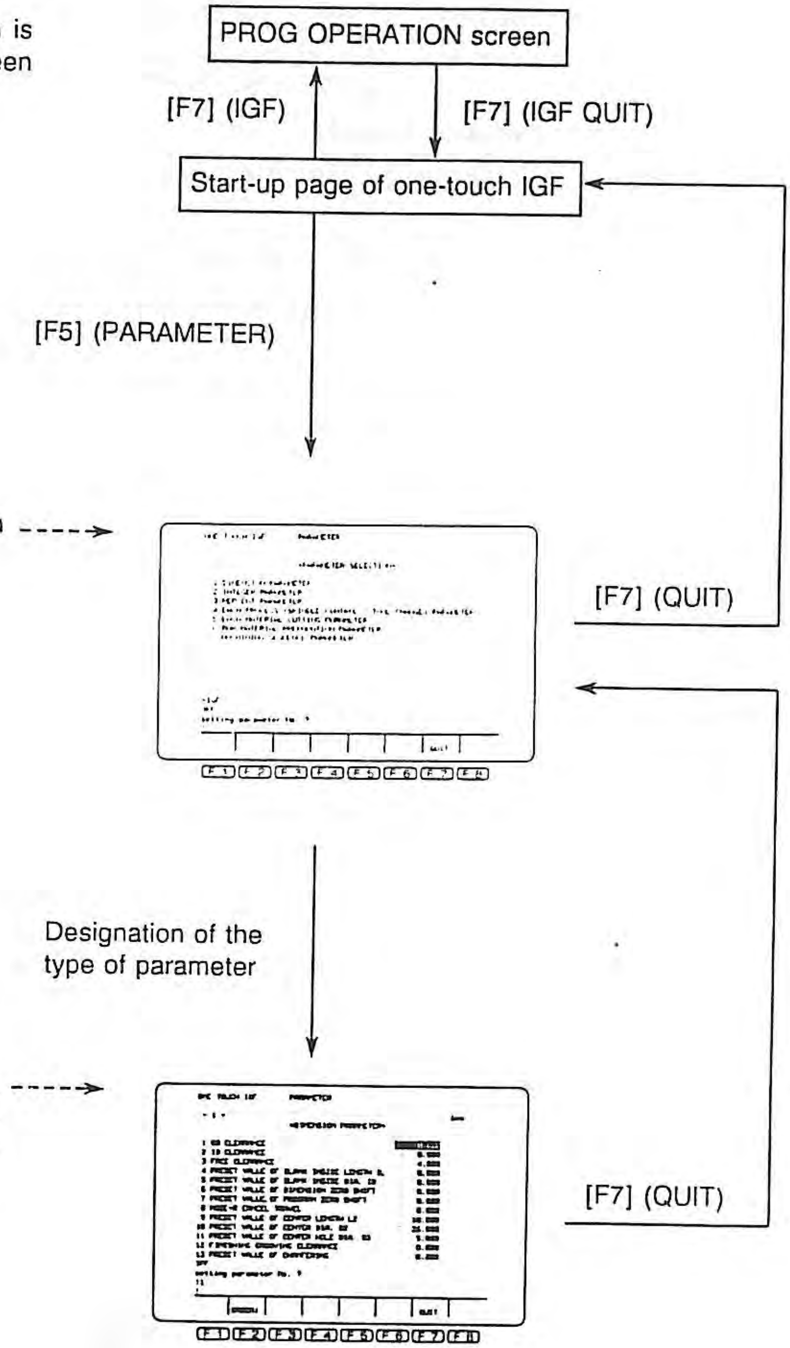
- DIMENSION PARAMETER
- INTEGER PARAMETER
- PERCENT PARAMETER
- EACH PROCESS (SPINDLE CONTROL / TOOL CHANGE) PARAMETER
- EACH MATERIAL CUTTING PARAMETER
- BAR MATERIAL PREPARATION PARAMETER
- MACHINING SEQUENCE PARAMETER

Input Procedure and Screen Transfer:

The one-touch IGF PARAMETER screen is called from the PROG OPERATION screen as diagramed to the right.

Select the type of parameter for which data is to be set.

Various types of parameter are displayed. Change the screen by pressing the function keys [F1] (ORDER ↑) and [F2] (ORDER ↓) and set the necessary data.



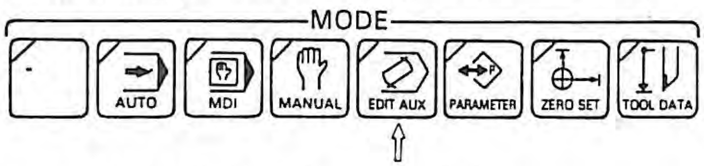
2. Parameter Setting Procedure

The parameter setting procedure is explained using an example.

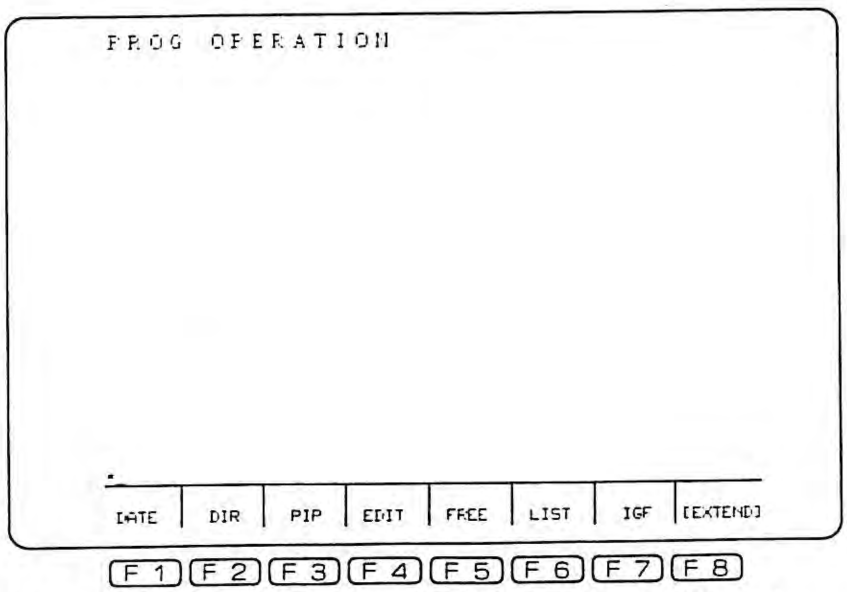
Example: To change the setting of 1_SEQUENCE_NO._INCREMENT_IN_PROCESS of INTEGER PARAMETER to sequential number

Operating Procedure:

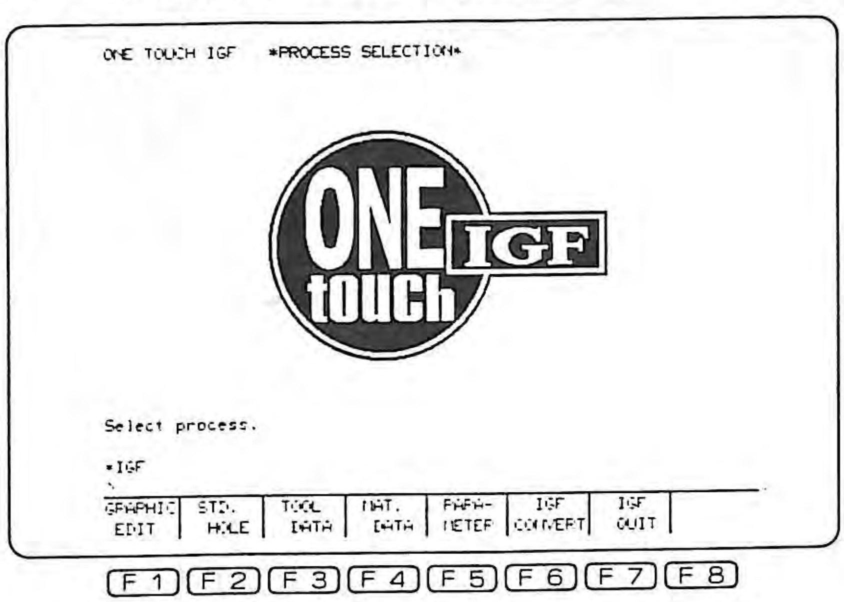
1) Press the **EDIT AUX** key to select the program operation mode.



The CRT screen will be changed as shown below.



2) Press the function key [F7] (IGF) to establish the one-touch IGF mode.
3) Press the function key [F5] (PARAMETER) to enable the material data setting.



4) Designate the type of parameter.

In this example, key in [2] and press the **WRITE** key to select "2. INTEGER PARAMETER".

```
ONE TOUCH IGF        PARAMETER

                     *PARAMETER SELECTION*

  1 DIMENSION PARAMETER
  2 INTEGER PARAMETER
  3 PERCENT PARAMETER
  4 EACH PROCESS (SPINDLE CONTROL / TOOL CHANGE) PARAMETER
  5 EACH MATERIAL CUTTING PARAMETER
  6 BAR MATERIAL PREPARATION PARAMETER
  7 MACHINING SEQUENCE PARAMETER

•IGF
>PF
setting parameter No. ?
!
|      |      |      |      |      |      | QUIT |      |
 [F 1] [F 2] [F 3] [F 4] [F 5] [F 6] [F 7] [F 8]
```

5) The INTEGER PARAMETER screen will be displayed.

Move the cursor to the data column of 1 SEQUENCE NO. INCREMENT IN PROCESS.

Key in [0] and press the **WRITE** key.

```
ONE TOUCH IGF        PARAMETER

 - 1 -
                     *INTEGER PARAMETER*

 1 SEQUENCE NO. INCREMENT IN PROCESS              10
 2 PRESET VALUE OF MATERIAL CODE                   2
 3 M41 MINIMUM SPINDLE SPEED                      75
 4 M42 MINIMUM SPINDLE SPEED                    1051
 5 M43 MINIMUM SPINDLE SPEED                       0
 6 M44 MINIMUM SPINDLE SPEED                       0
 7 M41 MAXIMUM SPINDLE SPEED                    1050
 8 M42 MAXIMUM SPINDLE SPEED                    4200
 9 M43 MAXIMUM SPINDLE SPEED                       0
10 M44 MAXIMUM SPINDLE SPEED                       0
11 MAXIMUM SPINDLE SPEED                        4200
12 PRESET VALUE OF THREADS PER INCH. J             1
13 PRESET VALUE OF THREAD LEADS NO. NT             1
>PF
setting parameter No. ?
!2
!
|      |ORDER↓|      |      |      |      | QUIT |      |
 [F 1] [F 2] [F 3] [F 4] [F 5] [F 6] [F 7] [F 8]
```

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

The CRT will return to the start-up page of the one-touch IGF screen.

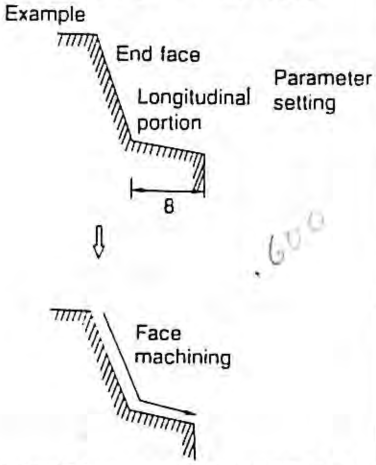
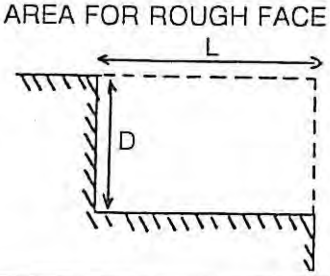
3. One-touch IGF Parameters

The one-touch IGF parameters are listed in the table below.

The unit system used for the initial setting of the DIMENSION PARAMETER is "mm". When "inch" is selected as the unit system, the inch values converted from metric values are displayed.

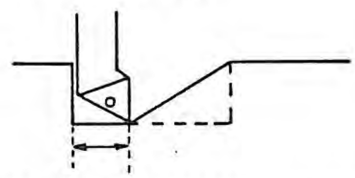
Parameters with indication "(multi-machining specification)" or "(in-process gauging specification)" are available when such specification is selected.

Name	Initial Value	Explanation	Reference
DIMENSION PARAMETER			
1. OD CLEARANCE	8	Clearance is added to the blank material shape and the clearance boundary is set.	Refer to Section 12, 8. "Tool Change Control".
2. ID CLEARANCE	8		
3. FACE CLEARANCE	4		
4. PRESET VALUE OF BLANK INSIDE LENGTH IL	0	Used for setting the default of individual input items on the BASIC BLANK SHAPE screen.	
5. PRESET VALUE OF BLANK INSIDE DIA. ID	0		
6. PRESET VALUE OF DIMENSION ZERO SHIFT	0		
7. PRESET VALUE OF PROGRAM ZERO SHIFT	0		
8. NOSE-R CANCEL TRAVEL	0	Used for eliminating uncut portion in taper and arc cuts. Identical to the I and K command values in the G40 (tool nose radius compensation cancel) mode.	
9. PRESET VALUE OF CENTER LENGTH L2			Refer to Section 8, "Setup Input".
10. PRESET VALUE OF CENTER DIA. D2			
11. PRESET VALUE OF CENTER HOLE DIA. D3			
12. FINISHING GROOVING CLEARANCE	0.05		
13. PRESET VALUE OF CHAMFERING	0.2	Default of chamfering which is automatically generated	

Name	Initial Value	Explanation	Reference
DIMENSION PARAMETER			
14. ROUGHING X-AXIS CLEARANCE	8	Clearance for calculating the cycle reference point is set.	Dimension parameter No. 69 and No. 70
15. ROUGHING Z-AXIS CLEARANCE	4		
16. FINISHING X-AXIS CLEARANCE	4		
17. FINISHING Z-AXIS CLEARANCE	2		
18. GROOVING CLEARANCE	2		
19. THREADING CLEARANCE	5		
20. RECESSING CLEARANCE	2		
21. DRILLING CLEARANCE	3		
22. TAPPING CLEARANCE	10		
23. LONGITUDINAL LENGTH FOR ROUGH / FIN. FACE Example  .600	10	Designation of longitudinal distance for combining end face machining elements Even if the shape which is tangent to the end face is recognized to be a longitudinal portion judging from the setting of integer parameters No. 31, 37, and 38, the percent parameter No. 35, and the dimension parameter No. 24, face machining is carried out for this portion when the longitudinal dimension L is smaller than the set value of this parameter.	Integer parameter No. 31 Integer parameter No. 37 Integer parameter No. 38 Percent parameter No. 35 Dimension parameter No. 24
24. MAX. WIDTH OF CUTTING AREA FOR ROUGH FACE 	20 1.00	Designation of maximum stock removal in the Z-axis direction to be judged as end face machining element	Integer parameter No. 31 Integer parameter No. 37 Percent parameter No. 35 Dimension parameter No. 23

Name	Initial Value	Explanation	Reference
DIMENSION PARAMETER			
24. MAX. WIDTH OF CUTTING AREA FOR ROUGH FACE		In rough machining, the area of machining, either face or longitudinal portion, varies depending on the relationship between the dimension L in the Z-axis direction of the blank material and the set value of this parameter: L < Set value – Face machining L > Set value – Machining in the longitudinal direction	
25. GROOVING TOOL WIDTH MAX. VALUE	7	Maximum width of grooving tool to be determined automatically	
26. GROOVING WIDTH MAX. VALUE	7	Maximum width for single groove cut	
28. BLIND HOLE MIN. DIA. VALUE	20	For drilling a blind hole, the minimum blind hole diameter for which an ID turning tool can be used is designated. For thru-hole drilling operation, the half of this value is the minimum thru-hole diameter. To drill a hole whose diameter is smaller than the set value of this parameter, an end mill is used for both roughing and finishing.	
29. DRILL & END MILL ALLOWANCE WIDTH (CUTTING)	2	The drill whose diameter ØD satisfies the following conditions is selected from the tool data for drilling: ØDm ≥ ØD ≥ ØDm × (set value of percent parameter No. 29) Here, ØDm is the diameter of the ideal drill and is calculated using the equation below: ØDm = (machining diameter entered in shape definition) – (set value of this parameter)	Percent parameter No. 29

Name	Initial Value	Explanation	Reference
DIMENSION PARAMETER			
30. END MILL ALLOWANCE WIDTH (NON-CUTTING)	0.5	The diameter of end mill which is used to cut the uncut portion of the drill hole is determined taking the set value of this parameter into account. For ID turning on the diameter which is larger than the set value of the dimension parameter No. 28, an end mill is used to cut the uncut portion left after drilling. The end mill whose diameter ØD satisfies the following conditions is selected from the tool data: ØDm ≧ ØD ≧ ØDm × (set value of percent parameter No. 32) Here, ØDm is the diameter of the ideal end mill and is calculated using the equation below: ØDm = (drill diameter) − (set value of this parameter)	Percent parameter No. 32
31. THRU HOLE EXCESS DEPTH	3	The excess depth when drilling a through hole is designated.	
32. TAPPING INCMP THREADS PITCH	3	Number of threads in the incomplete thread portion. The tap is infed exceeding the depth of tap designated by the amount calculated through (pitch) × (set value of this parameter).	
33. TAPPING PRE-DRILL EXCESS PITCH	4	When the tapping process is automatically generated, the pre-drilling tool is infed by the amount calculated through (pitch) × (set value of this parameter).	

Name	Initial Value	Explanation	Reference
DIMENSION PARAMETER			
35. DRILLING MINIMUM HOLE DEPTH	10	For blind hole drilling, the minimum value of stock on the end face (cut depth) is designated to judge whether or not drilling should be carried out.	
36. CARBIDE DRILL MINIMUM DIAMETER	18	The minimum diameter of the carbide drill set in automatic process determination is designated.	
37. DRILL MAXIMUM DIAMETER	35	The maximum diameter of the drill used is designated.	
38. TOOL HOLDER DIA. (凹 TURNING INTERF. CHECK)	25	The lower limit value for the inversed tool to approach to cut the concave shape. 	
39. COEFFICIENT OF CAL. IN CMP THREADS PITCH	25	Used for calculating the incomplete thread portion length.	
40. TOOL INDEX POSITION X-AXIS 41. TOOL INDEX POSITION Z-AXIS		The tool indexing position of the tool for which the tool indexing position has been not designated in the tool data setting mode is designated.	
42. REAR MACHINING Z-AXIS CLEARANCE	4	Used to calculate the cycle reference point for rear end face machining.	
43. THRU-HOLE CUTTING DEPTH	3	The infeed amount at a lower feedrate for drilling a thru-hole.	Refer to 4. (2) Data Setting in Section 11, "Process Edit".

Name	Initial Value	Explanation	Reference
DIMENSION PARAMETER			
44. ROUGHING NOSE-R MAXIMUM VALUE 45. FINISHING NOSE-R MAXIMUM VALUE	1.2 0.8	To eliminate the portions left uncut due to the nose R of the cutting tool, compensation is made at the cutting starting point and the cutting end point on the basis of these values. Enter the maximum nose R values.	
46. MAX. AIR CUT DISTANCE FOR RECESSED SHAPE	10		
48. GAUGE PT TO APPROACH PT1 DISTANCE (APP) 49. 1ST GAUGE PT TO APPROACH PT2 DIST (APPS) 50. GAUGE PT TO IMAGINARY PT DISTANCE (IMP) 51. NG LIMIT (DNG) 52. OK LIMIT (DOK)	2 1 2 0.04 0.02		Refer to Section 21, "One-touch IGF on In-process Gauging Function Specification". (In-process gauging specification)
53. M MACHIN. / CONTOUR GENERAT. OD CLEARANCE 54. M MACHIN. / CONTOUR GENERAT. FACE CLEARANCE 55. M MACHINING CLEARANCE	60 70 3	Clearance for calculating the reference point for the multi-machining cycle is set.	(multi-machining specification)
56. M MACHINING PRE-DRILL ALLOWA. WIDTH	2	The drill whose diameter ØD satisfies the following conditions is selected from the tool data to pre-drill a hole for boring or reaming. ØDm ≧ ØD ≧ ØDm × (set value of percent parameter No. 53) Here, ØDm is the diameter of the ideal drill and is calculated using the equation below: ØDm = (machining diameter entered in shape definition) – (set value of this parameter)	Percent parameter No. 53 (multi-machining specification)

Name	Initial Value	Explanation	Reference
DIMENSION PARAMETER			
57. M MACHINING REAMING PR-DRL ALLOWA. DEPTH	1	The excess depth of the pre-drill hole for reaming is set. Pre-drilling is executed exceeding the depth designated in shape definition by the amount set here.	(multi-machining specification)
58. M MACHINING BORING PT-DRL ALLOWA. DEPTH	1	The excess depth of the pre-drill hole for boring is set. Pre-drilling is executed exceeding the depth designated in shape definition by the amount set here.	(multi-machining specification)
59. GENERATING CLEARANCE	3	Clearance for calculating the reference point for the multi-machining cycle is set.	(contour generation machining specification)
60. M MACHINING DRILLING THRU-HOLE CUT. DEPTH	3	The depth of cut which is used to machine a thru-hole M-drill hole at a lower feedrate.	Refer to Section 11, 5. (2) Data Setting.
61. M MACHINING CHAMFERING ALLOWA. DIA	5	Chamfering is executed using a drill with the diameter calculated using the formula below: Hole diameter entered in shape definition + chamfer size + set value of this parameter	(multi-machining specification)
66. STICKING OUT CLEARANCE ALONG X-AXIS	10	Used for designating the margin for tool projection.	
67. STICKING OUT CLEARANCE ALONG Z-AXIS	10		

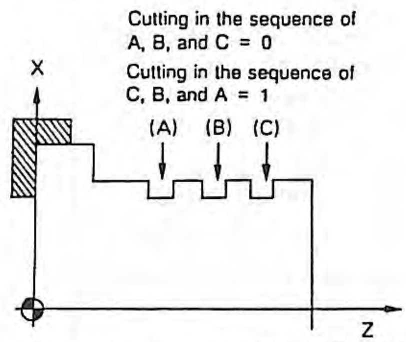
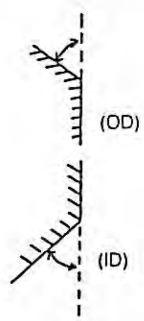
Name	Initial Value	Explanation	Reference
DIMENSION PARAMETER			
68. ID PROCESS ID CLEARANCE	3	Clearance for setting the X-coordinate of the process starting point for the ID longitudinal machining process.	
69. ID ROUGHING X-AXIS CLEARANCE	3	Clearance in the X-axis direction to set the cycle starting point for the ID longitudinal rough machining process.	Dimension parameter No. 16 and No. 17
70. ID FINISHING X-AXIS CLEARANCE	3	Clearance in the X-axis direction to set the cycle starting point for the ID longitudinal rough machining process.	

Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
1. SEQUENCE NO. INCREMENT IN PROCESS	10	1/10 of increment for assigning sequence numbers for each cutting process. When "0" is set, sequence numbers are assigned with consecutive numbers. "5" causes the first sequence of each process to be assigned as follows: N0000 1st process N0050 2nd process N0100 3rd process N0150 4th process : :	
2. PRESET VALUE OF MATERIAL CODE	2	Used for setting the default of material name in shape definition. The material whose number is designated for MATERIAL NO. is preset.	
3. M41 MINIMUM SPINDLE SPEED 4. M42 MINIMUM SPINDLE SPEED 5. M43 MINIMUM SPINDLE SPEED 6. M44 MINIMUM SPINDLE SPEED		The lowest spindle speed (rpm) in each range is set. The initial setting varies according to machine models.	
7. M41 MAXIMUM SPINDLE SPEED 8. M42 MAXIMUM SPINDLE SPEED 9. M43 MAXIMUM SPINDLE SPEED 10. M44 MAXIMUM SPINDLE SPEED		The lowest spindle speed (rpm) in each range is set. The initial setting varies according to machine models.	

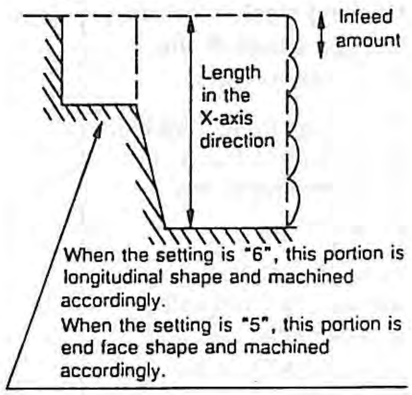
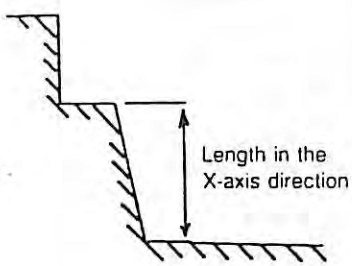
Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
11. MAXIMUM SPINDLE SPEED		The allowable maximum spindle speed is set meeting the setup selected. The designated spindle speed is output to the program with the G50 code. The initial setting varies according to machine models.	
12. PRESET VALUE OF THREADS PER INCH. J	1	Used for setting the default of each item of shape element data in shape definition.	
13. PRESET VALUE OF THREAD LEADS NO. NT	1		
14. PRESET VALUE OF GROOVES NO. NC	1		
15. PRESET VALUE OF SUR. RGH.	2	Used for setting the default of the finish symbol indicating the surface roughness of each shape element.	
16. IGF FILE STORAGE	1	Used for designating whether or not the IGF file is to be stored. 0: Not stored 1: Stored	
17. THREAD INFEED PATTERN (M CODE)	73	Used for designating tool infeed pattern for thread cutting. 73: M73 74: M74 75: M75	Refer to the section covering compound fixed cycle in the Programming Manual.
18. THREAD CUTTING MODE (M CODE)	32	Used for designating cutting pattern of thread cutting operation. 32: Straight infeed along thread face (left face) M32 33: Zigzag infeed M33 34: Reverse straight infeed along thread face (right face) M34	Refer to the section covering compound fixed cycle in the Programming Manual.

Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
19. TOOL OFFSET CANCEL OUTPUT (0 = YES, 1 = NO)	0	Used for designating whether or not the tool offset mode is canceled when the turret is returned to the turret indexing position. 0: Cancel command (T0100 or TG = 01, OG = 0) is output. 1: Cancel command is not output.	
20. SPACE DELETION IN PROGRAM	2	Used for designating whether or not spaces are output when creating a NC program. 0: Spaces are output. 1: Spaces are not output. 2: One space is output.	
21. TOOL EDGE DATA CHECK MARGIN (ANGLE)	3	Used for setting margin angle (0° - 360°) for tool edge data.	
22. TAILSTOCK BARRIER CHECK M-CODE OUTPUT	0	Used for designating whether or not the tailstock barrier check M code is output. 0: M code is not output. 1: M code is output.	
23. CHUCK BARRIER CHECK M-CODE OUTPUT OR NOT	0	Used for designating whether or not the chuck barrier check M code is output. 0: M code is not output. 1: M code is output.	
24. AUTO COMBINED 4-AXIS PROGRAMMING (A + B)	1	Used for designating that the simultaneous 4-axis control processes are automatically combined. 0: Simultaneous 4-axis control processes are not combined. 1: Simultaneous 4-axis control processes are combined.	

Name	Initial Value	Explanation	Reference
INTEGER PARAMETER 25. BALANCED CUTTING	2	Used for designating whether or not balance cut is executed. 0: Balance cut is not executed. 1: Balance cut is executed if the tool which can be used for balance cut is registered in the tool data. 2: Balance cut is executed only for the rough turning process if the tool which can be used for balance cut is registered in the tool data.(except grooving) 3: Balance cut is executed in any case. 4: Balance cut is executed for the rough turning process in any case.	
26. TOOL INTERF. CHECK M-CODE OUTPUT OR NOT	0	Used for designating whether or not the tool interference check M code is output. 0: M code is not output. 1: M code is output.	
27. MODAL M CODE FORCED OUTPUT(EACH PROCESS)	0	0: Modal codes are checked and the modal M code previously output and effective in the process in question is not output. 1: All M codes are output for each of processes disregarding of whether they are modal or not.	
28. M01 OUTPUT OR NOT (EACH PROCESS)	0	0: M01 is not output. 1: M01 is output at the end of each process.	

Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
29. THREAD, TAPPING CYCLE OUTPUT OR NOT (G77)	0	0: Tapping cycle in the turning operation mode is output in blocks of commands. 1: Tapping cycle in the turning operation mode is output in the G77 mode.	Refer to Section 6, (10) THREAD.
30. CONTINUOUS GROOVING DIR. (0 = → ↑ , 1 = ← ↓)  	0	The sequence for continuous groove cutting is designated. 0: Grooving is started from the position nearest to the chuck, or from the position nearest to the center line. 1: Grooving is started from the position nearest to the tailstock center, or from the position farthest from the center line.	
31. MAX. ANGLE OF INCLINATION FOR FACE  	20	The angle by which the portion is judged to be an end face. If the angle is equal to or smaller than the set value of the parameter, the portion is judged to be an end face.	Integer parameter No. 37 Integer parameter No. 38 Dimension parameter No. 24 Dimension parameter No. 23 Percent parameter No. 35

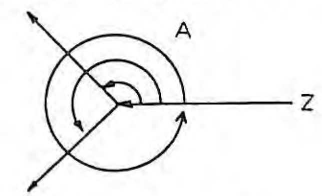
Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
32. ROUGH SHAPE OUTPUT (0 = OFFSET, 1 = SHIFT)	0	Used for designating whether or not finishing stock is added to the finished shape of the rough machining process. 0: Finishing stock is added, outputting neither U nor W command to the program. 1: Finishing stock is not added. Finishing stock is designated using U and W words in the program.	
33. CENTERING TOOL SELECTION	1	Designation of centering tool for drilling operation in the turning operation mode 0: Center drill 1: Drill tool is used as center drill as well.	
34. FORMATION ANGLE CHAMFERING	130	Chamfering process is generated if the angle of the shape is less than the value set by this parameter.	
35. RUGH. CYCLE SELECT. (0 = ROUND BAR, 1 = COPYING)	0	Used for designating which LAP cycle is used in creating an NC program. 0: High-speed round bar machining cycle 1: Blank material profile machining cycle	
36. ROUND BAR/COPYING M85 OUTPUT OR NOT	1	Used to designate whether or not M85 (no cycle reference point return) is output in the high-speed bar turning cycle or copy turning cycle in the LAP-4 mode. 0: M85 is not output 1: M85 is output	Instruction Manual for LAP-4

Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
37. FACE LENGTH DECIDE ON LONG ROUGH MACHIN.  	3 1	When the length in the X-axis direction of the shape element which was judged to be an end face at the integer parameter No. 31 is smaller than the value obtained through "infeed amount × parameter setting", that shape element is recognized to be a longitudinal shape element and machined accordingly. The infeed amount refers to the infeed amount of the roughing ←, → in the material data.	Integer parameter No. 31 Dimension parameter No. 24 Dimension parameter No. 23 Percent parameter No. 35
38. FACE LENGTH DECIDE ON LONG FIN. MACHIN.  	6 60	When the length in the X-axis direction of the shape element which was judged to be an end face at the integer parameter No. 31 is smaller than the value obtained through "infeed amount × parameter setting", that shape element is recognized to be a longitudinal shape element and machined accordingly. The infeed amount refers to the infeed amount of the roughing ←, → in the material data.	Integer parameter No. 31 Dimension parameter No. 23
40. M241 MINIMUM SPINDLE SPEED 41. M242 MINIMUM SPINDLE SPEED 42. M241 MAXIMUM SPINDLE SPEED 43. M242 MAXIMUM SPINDLE SPEED		The allowable maximum/minimum M-spindle speed (rpm) is set. The initial setting varies according to machine models.	(multi-machining specification)

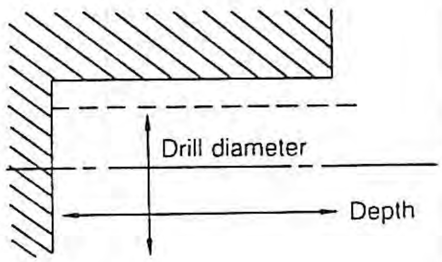
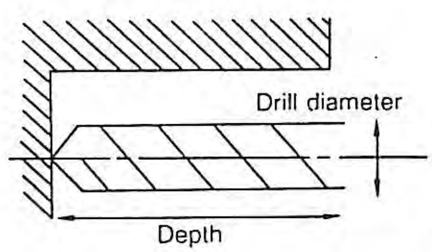
Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
44. PRESET VALUE OF SUR. RGH, (GENERATING)	2	Used for setting the default of the finish symbol indicating the surface roughness of each shape element in graphic edit. (designation of surface roughness)	(contour generation machining specification)
47. GENERAT. ROUGH. RECIPROC .(0 = ONE, 1 = BOTH)	0	0: One directional roughing 1: Reciprocating roughing	(contour generation machining specification)
48. COORDINATE SYSTEM TYPE (0 = X-C, 1 = X-Y)	0	Used for setting the default of coordinate system for multi-machining shape definition. 0: X-C coordinate 1: X-Y coordinate	(multi-machining specification)
46. CUTTER RADIUS COMP. IN FACE GENERATING	1	Used for designating whether or not cutter radius compensation is made in face contour generation machining. 0: Cutter radius compensation in NC is not made. The one-touch IGF outputs the tool center path. 1: Cutter radius compensation in NC is made. The one-touch IGF outputs both the finished shape path and the cutter radius compensation G code.	(contour generation machining specification)
49. M ARC DEFINE DIRECTION (0 = CW, 1 = CCW)	1	Used for setting the default of hole position direction in multi-machining shape definition. 0: CW 1: CCW	(multi-machining specification)

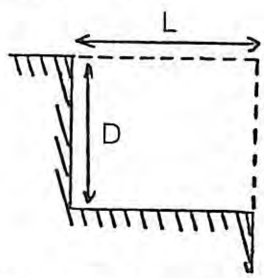
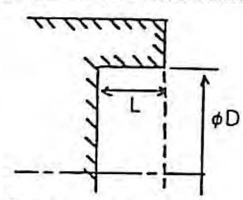
Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
49. M ARC DEFINE DIRECTION (0 = CW, 1 = CCW)	1	Used for setting the default of hole position direction in multi-machining shape definition. 0: CW 1: CCW	(multi-machining specification)
50. KEY WAY CUT. DEPTH PATTERN (M CODE)	213	Used for designating the infeed pattern of keyway cutting. 213: M213 infeed by designated amount 214: M214 equal infeed	Refer to the Programming Manual for Multi-machining Models. (multi-machining specification)
51. KEY WAY CUTTING MODE (M CODE)	211	Used for designating the infeed direction of keyway cutting. 211: M211 one directional 212: M212 reciprocating	Refer to the Programming Manual for Multi-machining Models. (multi-machining specification)
52. TN CODE LOCATION IN PROGRAM (for ATC)	0	Used for designating the output position of next tool command with the ATC specification. 0: Tool is indexed during the execution of the previous process. 1: Tool is indexed after the completion of the previous process.	(multi-machining ATC specification)
53. HOME POSIT. MOVE COMMAND EXIST (for ATC)	0	0: Turret is rotated at the tool change position. 1: Turret is rotated at Home Position 1 (HP-1).	(multi-machining ATC specification)
54. ATC HOME POSIT. AREA (0 = A, 1 = B, 2 = C)	1	Used for designating in which area, A, B, or C in the illustration in Section 12 the home position is present. Air-cutting cycle is generated following the area set by this parameter in the one-touch IGF.	Refer to Section 12, "PART PROGRAM CREATION". (multi-machining specification)

Name	Initial Value	Explanation	Reference
INTEGER PARAMETER			
55. M MACHINING HOLE PROCESS WAY	2	Used for designating the machining method of each hole machining. 0: With centering process 1: With centering motion 2: With centering process including chamfering For the machining process which is actually generated by the automatic process determination function, refer to the explanation for shape definition of each hole machining.	Refer to Section 6, 4. Finished Shape Definition for Multiple Machining (optional) (multi-machining specification)
56. KEY WAY CUTTING (0 = ROUGH, 1 = ROUGH / FINISH)	1	Used for designating whether or not both roughing and finishing are executed in keyway cutting operation. 0: Roughing only 1: Both roughing and finishing	(multi-machining specification)

Name	Initial Value	Explanation	Reference
PERCENT PARAMETER			
1. ROUND BAR CYCLE ↖ DIRECTION	150	Factors (%) for determining the feedrate to be used when cutting is made along the shape in round bar and copy turning operations. Factors are assigned for three different cutting directions such as ↖, ←, ↙, in reference to the main cutting direction.  ↖: 0° ≦ A < 135° ←: 135° ≦ A ≦ 225° ↙: 225° < A < 360°	
2. ROUND BAR CYCLE ← DIRECTION	100		
3. ROUND BAR CYCLE ↙ DIRECTION	100		
4. COPYING CYCLE ↖ DIRECTION	150		
5. COPYING CYCLE ← DIRECTION	100		
6. COPYING CYCLE ↙ DIRECTION	100		
7. GROOVING SHIFT (*GROOV. TOOL WIDTH)	90	Tool shift factor in grooving operation Shift amount is the tool width × set factor (%).	
8. GROOVING MID RETRACT (*GROOV. TOOL WIDTH)	200	Factor used for determining tool retract and infeed amounts during grooving cycle. Retract amount is the tool width × set factor (%).	
9. GROOVING END DWELL (*RPM)	200	Dwell period at the end point of the grooving is specified in the number of spindle revolutions.	
10. DRILLING MID RETRACT (*DRILL DIA.)	200	Factor used for determining drill retract and infeed amounts during drilling cycle. Retract amount is the drill diameter × factor (%).	
12. DRILLING THRU-HOLE FEEDRATE	50	Feedrate for drilling a through hole is determined by this factor. Drill feedrate × factor (%).	
13. DRILLING END DWELL (*RPM)	200	Dwell period at the end point of drilling is specified in the number of spindle revolutions.	

Name	Initial Value	Explanation	Reference
PERCENT PARAMETER			
14. CENTERING END DWELL (*RPM)	200	Dwell period at the end point of centering is specified in the number of spindle revolutions.	
15. RECESSING END DWELL (*RPM)	200	Dwell period at the end point of recessing is specified in the number of spindle revolutions.	
16. THREAD CHAMFERING DISTANCE (*PITCH)	0	Factor used for determining the chamfering amount in thread cutting operation. Chamfering amount is F/J × factor (%).	
17. HEIGHT OF SCREW THREAD (EXTERNAL) (*PITCH)	65	Factors (%) used for designating the default of HEIGHT	
18. HEIGHT OF SCREW THREAD (INTERNAL) (*PITCH)	50	HEIGHT = LEAD/(TPI × NO. OF THREAD LEADS) × FACTOR (%)/100	
19. HEIGHT OF SCREW THREAD (END FACE) (*PITCH)	87		
20. 4-AXIS CUTTING CONDITIONS (RPM)	120	Setting range: 0 - 200%	
21. 4-AXIS CUTTING CONDITIONS (CUT DEPTH)	50	Setting range: 0 - 100%	
22. 4-AXIS CUTTING CONDITIONS (DRILL FEED)	70	Setting range: 0 - 100%	
23. 4-AXIS CUTTING CONDITIONS (GROOVE FEED)	80	Setting range: 0 - 100%	
24. SPINDLE OVERRIDE FOR SIMULATION	100	Setting range: 0 - 300%	
25. FEEDRATE OVERRIDE FOR SIMULATION	100	Setting range: 0 - 300%	
26. POSSIBLE CUT DEPTH (*GROOVE TOOL WIDTH)	300	Setting range: 0 - 1000%	
27. BALANCED CUTTING DEPTH	100	Setting range: 0 - 300%	
28. BALANCED CUTTING FEEDRATE	200	Setting range: 0 - 300%	

Name	Initial Value	Explanation	Reference
PERCENT PARAMETER			
29. DRILL SELECTION LOWER LIMIT	90	Factor used for setting the lower limit value of the drill diameter when selecting a drill tool	Dimension parameter No. 29
30. HS. DRILL POSSIBLE CUT. DEPTH / DRILL DIA  	250	When generating a drill process, a carbide drill is selected when the value (depth/drill diameter) is smaller than the parameter (%). Note that the drill diameter must be equal to or larger than the setting of the integer parameter No. 36.	Dimension parameter No. 36 Dimension parameter No. 29 for drill diameter Percent parameter No. 29
31. SHAPE NEEDS CENTERING  	50	When a drill process using a HSS drill is generated, centering is executed when the value (depth/drill diameter) is equal to or larger than the parameter (%). How the centering is performed is designated by the integer parameter No. 33.	Integer parameter No. 33
32. END MILL SELECTION LOWER LIMIT	20	Factor used for setting the lower limit value of the endmill diameter for the rough endmilling process. The parameter is applied to the end mill whose diameter is larger than the set value of the dimension parameter No. 28.	50

Name	Initial Value	Explanation	Reference
PERCENT PARAMETER			
33. END MILL MID RETRACT (*END MILL DIA.)	200	Factor used for determining tool retract and infeed amounts during endmilling cycle. Retract amount is the endmill diameter × set factor (%).	
34. SMALL-DIA. END MILL SELECT. LOWER LIMIT	100	To drill the ID of a blind hole larger than the set value of the dimension parameter No. 28, an endmill is used for both roughing and finishing operations. This is the factor used for selecting the lower limit value of the endmill.	100
35. FACE ROUGH CUT. AREA RATIO L : B 	300	End face rough machining is executed when the following conditions are satisfied: Length/width ratio L/D ≧ set value of this parameter (%) Dimension in the Z-axis direction < set value of the dimension parameter No. 24	Integer parameter No. 31 Integer parameter No. 37 Dimension parameter No. 24 Dimension parameter No. 23
36. FACE DECISION CUT. INSIDE SHAPE	1000	 The shape as diagramed above is machined in the face machining process (ID machining), without using a drill, when ØD/L is equal to or larger than the set value of this parameter (%). Note that a drill is also used when L is so large that the uncut portion is left in the face machining process.	
40. M DRILLING MID RETRACT (*END MILL DIA.)	200	Factor used for determining tool retract and infeed amounts during M-drilling cycle. Retract amount is the M-drill diameter × set factor (%).	(multi-machining specification)

Name	Initial Value	Explanation	Reference
PERCENT PARAMETER			
42. M DRILLING THRU-HOLE FEEDRATE	50	Feedrate for drilling a M-drill through hole is determined by this factor. M-drill feedrate × factor (%).	(multi-machining specification)
43. M DRILLING END DWELL (*RPM)	200	Dwell period at the end point of M-drilling is specified in the number of M-drill revolutions.	(multi-machining specification).
44. M CENTERING END DWELL (*RPM)	200	Dwell period at the end point of M-centering is specified in the number of M-centering tool revolutions.	(multi-machining specification)
45. FACE GENERATING CUTTING FEEDRATE	50	Factor used for determining feedrate from cycle reference point to cutting starting point. Face generating machining feedrate × set factor (%).	(contour generation machining specification)
46. SIDE GENERATING CUTTING FEEDRATE	50	Factor used for determining feedrate from cycle reference point to cutting starting point. Side generating machining feedrate × set factor (%).	(contour generation machining specification)
47. KEY WAY CUTTING FEEDRATE	50	Factor used for determining feedrate for groove cutting. Rough generating machining feedrate × set factor (%).	(multi-machining specification)
48. M TAPPING END DWELL (*RPM)	0	Dwell period at the end point of M-tapping is specified in the number of M-tap revolutions.	(multi-machining specification)
49. M END MILL MID RETRACT (*END MILL DIA.)	200	Factor used for determining tool retract and infeed amounts during M-endmilling cycle. Retract amount is the M-endmill diameter × set factor (%).	(multi-machining specification)
50. M DRILLING CENTER SELECTION UPPER LIMIT 51. M DRILLING CENTER SELECTION LOWER LIMIT	110 50	Factor used for setting the lower and upper limit values of center-drill diameter when selecting an center-drill.	(multi-machining specification)
52. M DRILLING CHAMFER SELECTION UPPER LIMIT	150	Factor used for setting the lower limit value of chamfering drill diameter when selecting a chamfering drill.	(multi-machining specification)

Name	Initial Value	Explanation	Reference
PERCENT PARAMETER			
53. M DRILLING PRE-DRILL SELECT. LOWER LIMIT	90	Factor used for setting the lower limit value of the pre-drilled diameter when selecting a pre-drill tool.	Dimension parameter No. 56 (multi-machining specification)
54. M BORING SELECTION LOWER LIMIT	95	Factor used for setting the lower limit value of boring diameter when selecting an M-boring tool.	(multi-machining specification)
55. M END MILL SELECTION LOWER LIMIT	95	Factor used for setting the lower limit value of endmill diameter when selecting an M-endmill for the rough reaming process.	(multi-machining specification)
56. KEY WAY TOOL (END MILL) LOWER LIMIT	90	Factor used for setting the lower limit value of keyway cutting tool diameter when selecting a rough keyway cutting tool.	(multi-machining specification)

Name	Initial Value	Explanation	Reference
SPINDLE CONTROL PARAMETER			
SPINDLE ROT		Used for designating the default of CW/CCW parameter of CONDITION data setting process. 0: Reverse direction, left thread 1: Forward direction, right thread	
CONST SPEED		Used for designating the default of constant speed cutting mode selection. 0: Constant speed cutting OFF 1: Constant speed cutting ON	
TOOL CHANGE PARAMETER			
RETRACT APPROACH		Used for designating the retract mode to the turret indexing position and the approaching mode to the circle reference point. 0: One axis 1: Two axes 2: Gang tooling (for turning only) For the multi-machining specification, the same designation is made also for revolving tools.	
EACH MATERIAL CUTTING PARAMETER			
DRILLING CARBIDE HSS GROOVING		Used for designating the step feed mode for individual materials. 0: No step feed 1: Step feed (Designate the material name using the material number in the material table.)	

Name	Initial Value	Explanation	Reference
EACH MATERIAL CUTTING PARAMETER COOLANT		Used for designating the default of coolant on/off status for individual materials. 0: Coolant OFF 1: Coolant ON (Designate the material name using the material number in the material table.)	
END MILL ROUGH FINISH		Used for designating the step feed mode for individual materials. 0: No step feed 1: Step feed (Designate the material name using the material number in the material table.)	
M DRILL		Used for designating the step feed mode for individual materials. 0: No step feed 1: Step feed (Designate the material name using the material number in the material table.)	(multi-machining specification)
M END MILL		Used for designating the step feed mode for individual materials. 0: No step feed 1: Step feed (Designate the material name using the material number in the material table.)	(multi-machining specification)

Name	Initial Value	Explanation	Reference
BAR MATERIAL PREPARATION PARAMETER			
1. BAR FEEDER / PULLER OPERATION PATTERN	0	Used for designating the type of bar feeder (bar puller).	Refer to Section 20, "Bar Feeder and Bar Fuller Function"
2. BAR FEEDER / PULLER PROGRAM (0 = OFF < 1 = ON)	0	Used for designating whether or not bar feeder (bar puller) controlling M codes are output when creating a program.	
3. TOP CUTTING SPEED	80	Setting range: 1 - 3200 (mm/rev)	
4. TOP CUTTING FEEDRATE	0.1	Setting range: 0.001 - 9999.999 (mm/min)	
5. TOP CUTTING SPINDLE (0 = CCW, 1 = CW)	1	Used for designating the spindle rotation direction for top-cut operation.	
6. TOP CUTTING COOLANT (0 = OFF,1 = ON)	1	Used for designating the on/off status of coolant for top-cut operation.	
7. WORK STOPPER START POSITION FOR TOP CUT	0	Setting range: 0.001 - 9999.999 (mm)	
8. WORK STOPPER START POSIT. BEFORE OPERAT.	0	Setting range: 0.001 - 9999.999 (mm)	
9 WORK STOPPER INTERLOCK FEED	2000	Setting range: 0.001 - 9999.999 (mm/min)	
10. INITIAL BAR STOCK-OUT LENGTH	15	Setting range: 0.001 - 9999.999 (mm)	
11. BAR PULLER STARTING POSITION	20	Setting range: 0.001 - 9999.999 (mm)	
12. BAR PULLER GRIPPING DISTANCE	10	Setting range: 0.001 - 9999.999 (mm)	
13. BAR PULLER CATCHING SPEED	1000	Setting range: 0.001 - 9999.999 (mm/min)	
14. BAR PULLER DRAWING SPED	2000	Setting range: 0.001 - 9999.999 (mm/min)	

Name	Initial Value	Explanation	Reference
MACHINING SEQUENCE PARAMETER			
1. DRILLING	1	These are the auxiliary parameters used to designate the machining sequence. There may be cases in which the machining sequence does not change even if the setting of these parameters has been changed.	
2. OD FACE ROUGHING	2		
3. OD LONGITUDINAL ROUGHING	3		
4. ID FACE ROUGHING	4		
5. ID LONGITUDINAL ROUGHING	5		
6. OD FACE FINISHING	6		
7. OD LONGITUDINAL FINISHING	7		
8. ID FACE FINISHING	8		
9. ID LONGITUDINAL FINISHING	9		
10. FACE GROOVING	10		
11. OD GROOVING	11		
12. ID GROOVING	12		
13. OD THREADING	13		
14. FACE THREADING	14		
15. ID THREADING	15		
16. OD RECESSING	16		
17. ID RECESSING	17		
18. TAPPING	18		
1. GENERATING	1	Used for designating the sequence for contour generation or multi-machining operation.	(multi-machining specification)
2. KEY WAY CUTTING	2		
3. M HOLE DRILLING	3		

SECTION 6 FINISHED SHAPE DEFINITION (SHAPE DEFINITION)

1. What is the Function of "Finished Shape Definition"?

In this section, finished shape definition of the one-touch IGF is explained.

With the one-touch IGF, the machining process is automatically determined through designating the finished shape and blank material shape of workpiece. Therefore, it is not necessary to take the machined shape of workpiece into consideration for each machining process.

Define the finished shape generated through turning and milling operations.

The finished shape definition process consists of the following shape definition steps.

- BASIC BLANK SHAPE definition
- TURNING MACH SHAPE definition
- M MACHINING SHAPE definition (optional)
- FACE GENERATING SHAPE definition (optional)
- SIDE GENERATING SHAPE definition (optional)

The relationship between the sequence of shape definition and the finished shape definition is diagramed below.

1) Basic data of blank material is entered in the BASIC BLANK SHAPE definition step.
2) Designation of finished shape for each of turning, multi-machining, and contour generating processes.
3) Designation of AVE. BLANK SHAPE or BLANK SHAPE.

Startup of shape definition

⇩

1) BASIC BLANK SHAPE definition

⇩

2) TURNING MACH SHAPE definition

⇩

2) M MACHINING SHAPE definition
FACE GENERATING SHAPE definition
SIDE GENERATING SHAPE definition

⇩

3) AVE. BLANK SHAPE definition
BLANK SHAPE definition

⇩

Completion of shape definition
To SETUP and process determination

2. Basic Blank Shape Definition (Designation of blank material, basic blank material shape (outline of blank material shape), and type of blank material shape)

In this process, blank material, basic blank material shape, and type of blank material shape are designated.

Screen Transfer:

The BASIC BLANK SHAPE screen is called from the PROG OPERATION screen as diagramed below.

PROG OPERATION screen

[F7] (IGF) ↓ ↑ [F7] (IGF QUIT)

Start-up page of one-touch IGF

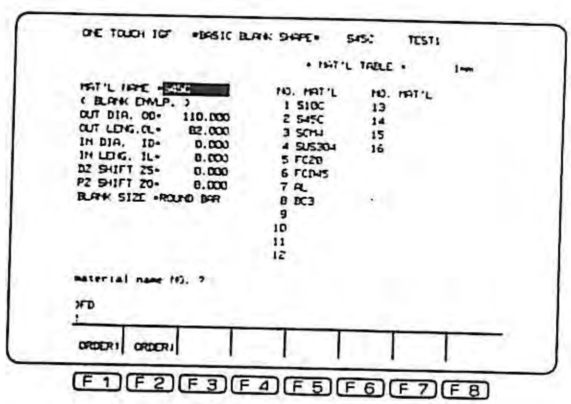
[F1] [GRAPHIC EDIT)
+ IGF file name
+ **WRITE** ↓ ↑ [F7] (QUIT)

Start-up page of graphic edit

[F1] (SHAPE DEFINE) ↓ ↑ [F1] (ORDER ↑)

BASIC BLANK SHAPE screen

Blank material, basic blank material shape, and type of blank material shape are designated. -------



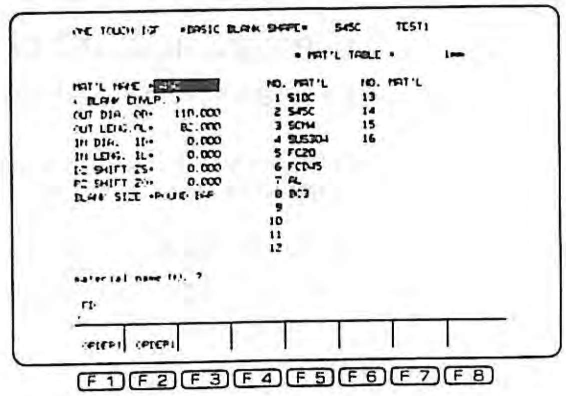
After the completion of data input, press the function key [F2] (ORDER ↓) to advance to the TURNING MACH SHAPE screen. ------ [F2] (ORDER ↓) ↓

This is the TURNING MACH SHAPE screen.

Note: *Detailed designation of blank material shape is made after the completion of finished shape definition. Refer to Section 7. Note that if "ROUND BAR" is designated as the type of blank material shape, the data set for OUT DIA. OD =, OUT LENG. OL =, IN DIA. ID =, and IN LENG. IL = in the BASIC BLANK SHAPE process becomes effective as the detailed data of blank material shape. Therefore, no designation is made after the completion of finished shape definition.*

a) Data to be Designated

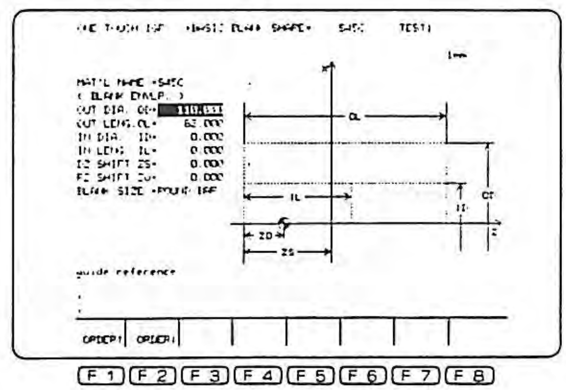
MAT'L NAME	
OUT DIA.	OD
OUT LENG.	OL
IN DIA.	ID
IN LENG.	IL
DZ SHIFT	ZS
PZ SHIFT	ZO
BLANK SIZE	



b) How to Designate

Cursor control keys

Move the cursor to the desired data column using the cursor control keys (follow the prompt if displayed), and enter the data.



MAT'L NAME

Material name of the blank material to be machined is designated.

Material number and material name are displayed on the MAT'L TABLE on the BASIC BLANK SHAPE screen. Enter the material number.

OUT DIA. OD, OUT LENG. OL, IN DIA. ID, IN LENG. IL

Shape of the blank material to be machined is roughly designated. Enter the maximum OD of the blank material for OUT DIA. OD, and the maximum length for OUT LENG. OL, according to the guide drawing displayed on the screen.

For IN DIA. ID and IN LENG. IL, enter the data in the same manner.

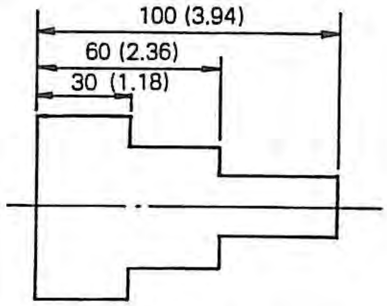
This enables the display of basic blank material shape on the TURNING MACH SHAPE screen with yellow color.

DZ SHIFT ZS

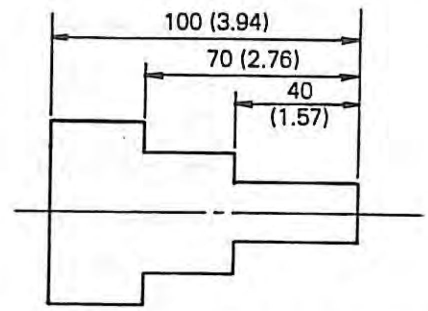
The reference point of the blank material in the longitudinal direction is designated. This point refers to the point from which the shape data in the longitudinal direction is entered, that is Z = 0.

Key in "0" when the dimensions are taken while placing the reference point at the left end of workpiece.

Key in "100" for example when the dimensions are taken while placing the reference point at the right end of workpiece.



Example with the reference point at the left end of workpiece



Example with the reference point at the right end of workpiece

The reference point in the diametrical direction is fixed at the center of diameter, that is X = 0.

PZ SHIFT ZO

The Z-coordinate of program zero is designated. A part program taking this point as program zero (work zero) is created by the one-touch IGF system.

BLANK SIZE

The type of blank material shape is designated. Select from the following three types.

1. ROUND BAR

   Basic blank material shape

2. AVERAGE STICK

   Blank material shape such as casting which has uniform stock in respect to the finished shape. (Refer to Section 7 of this manual.)

3. FREE SHAPE

   Arbitrary-shaped blank material (Refer to Section 7 of this manual.)

c) End of Designation

When all necessary data has been entered, press the function key [F2] (ORDER ↓). The one-touch IGF system will check the entered data and the CRT will advance to the TURNING MACH SHAPE screen when the data has been correctly entered. An error will occur if there is faulty data.

Pressing the function key [F1] (ORDER ↑) will return the CRT to the start-up page of the one-touch IGF screen.

3. TURNING MACH SHAPE Definition

The explanation that follows is the procedure for finished shape definition for turning.

The following functions are provided to define the turning shape.

(1) START PT (START PT, DEFINE DIR., CHAMFERING)

(2) FACE

(3) TAPER

(4) LONG

(5) CCW ARC

(6) CW ARC

(7) R-CHF

(8) CHAMFER

(9) GROOVE

(10) THREAD

(11) RECESS

(12) JUMP

(13) Display of guide drawing

(14) Insertion of shape element

(15) Deletion of shape element

(16) Re-display of shape

In the turning shape definition process, first designate the shape definition starting point, shape definition direction, and chamfer size. Then, enter the shape elements (2) to (12) sequentially according to the finished shape.

Note: *A total of 256 shape elements can be designated. (With the optional multi-machining specification, the total of turning shape elements and multi-machining elements is 256.)*

The relationship between the sequence of shape definition and the turning shape definition is diagramed below.

Startup of shape definition

↓

BASIC BLANK SHAPE definition

↓ [F2] (ORDER ↓)

TURNING MACH SHAPE definition

- START PT
- TAPER
- LONG
- FACE
- CW ARC
- R-CHF
- CCW ARC
- GROOVE
- THREAD
- CHAMFER
- JUMP
- RECESS
- Display of guide drawing
- Insertion of shape element
- Deletion of shape element
- Re-display of shape

When turning shape definition is completed, press the function key [F2] (ORDER ↓). The CRT will advance to the BLANK SHAPE screen or M MACHINING SHAPE screen.

↓ [F2] (ORDER ↓)

M MACHINING SHAPE definition
FACE GENERATING SHAPE definition
SIDE GENERATING SHAPE definition

Optional:
Multi-machining specification
Contour generating machining specification

↓

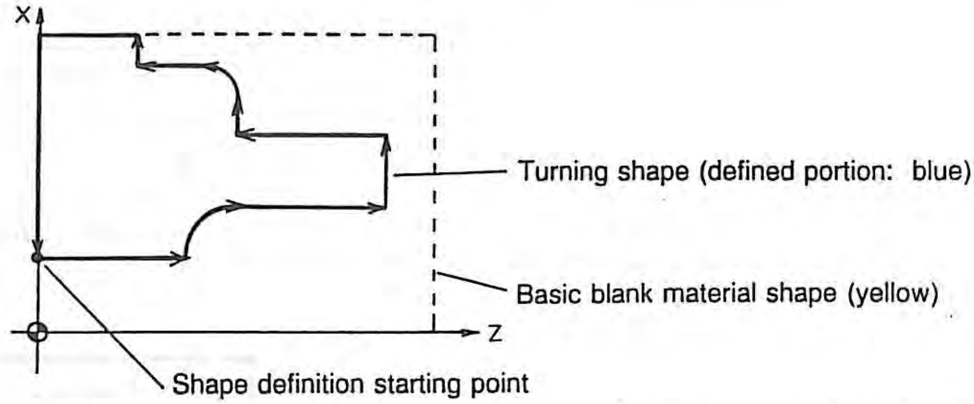
AVE. BLANK SHAPE definition
BLANK SHAPE definition

↓

Completion of shape definition
To SETUP and process determination

Turning shape definition is carried out in the following sequence.

Example:



1) After the completion of basic blank material shape definition, press the function key [F2] (ORDER ↓) to advance to the TURNING MACH SHAPE screen.
2) Enter the data for START PT, DEFINE DIR., CHAMFERING. (Refer to page 77.)
3) Select the shape element and define the turning shape.

   Necessary shape elements: LONG, CW ARC, LONG, FACE, LONG, FACE, R-CHF, LONG, FACE, JUMP

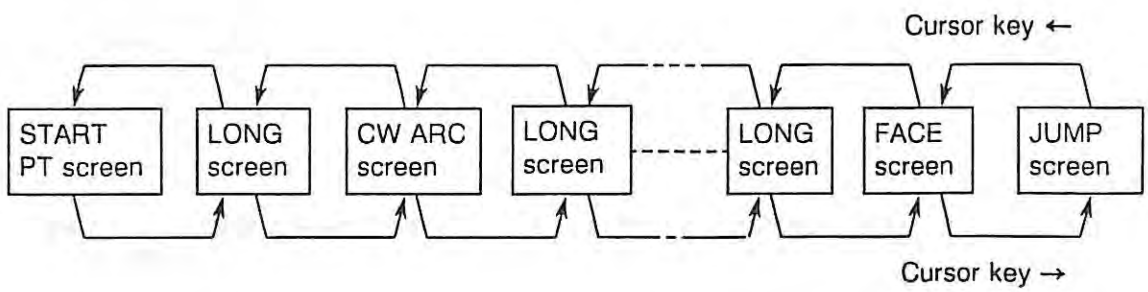
   Start turning shape definition from the starting point and define the turning shape in such a manner that the starting point and end point will agree each other.

   Definition with a single stroke
4) After the completion of turning shape definition, press the function key [F2] (ORDER ↓).

   The one-touch IGF system will check the entered data and the CRT will advance to the BLANK SHAPE screen or M MACHINING SHAPE screen if the data has been correctly entered. An error will occur if there is faulty data.

Note : *Using the cursor control keys ← and →, it is possible to display the shape element designation screen for the turning shape which has been already defined. Change of the set data is also possible.*

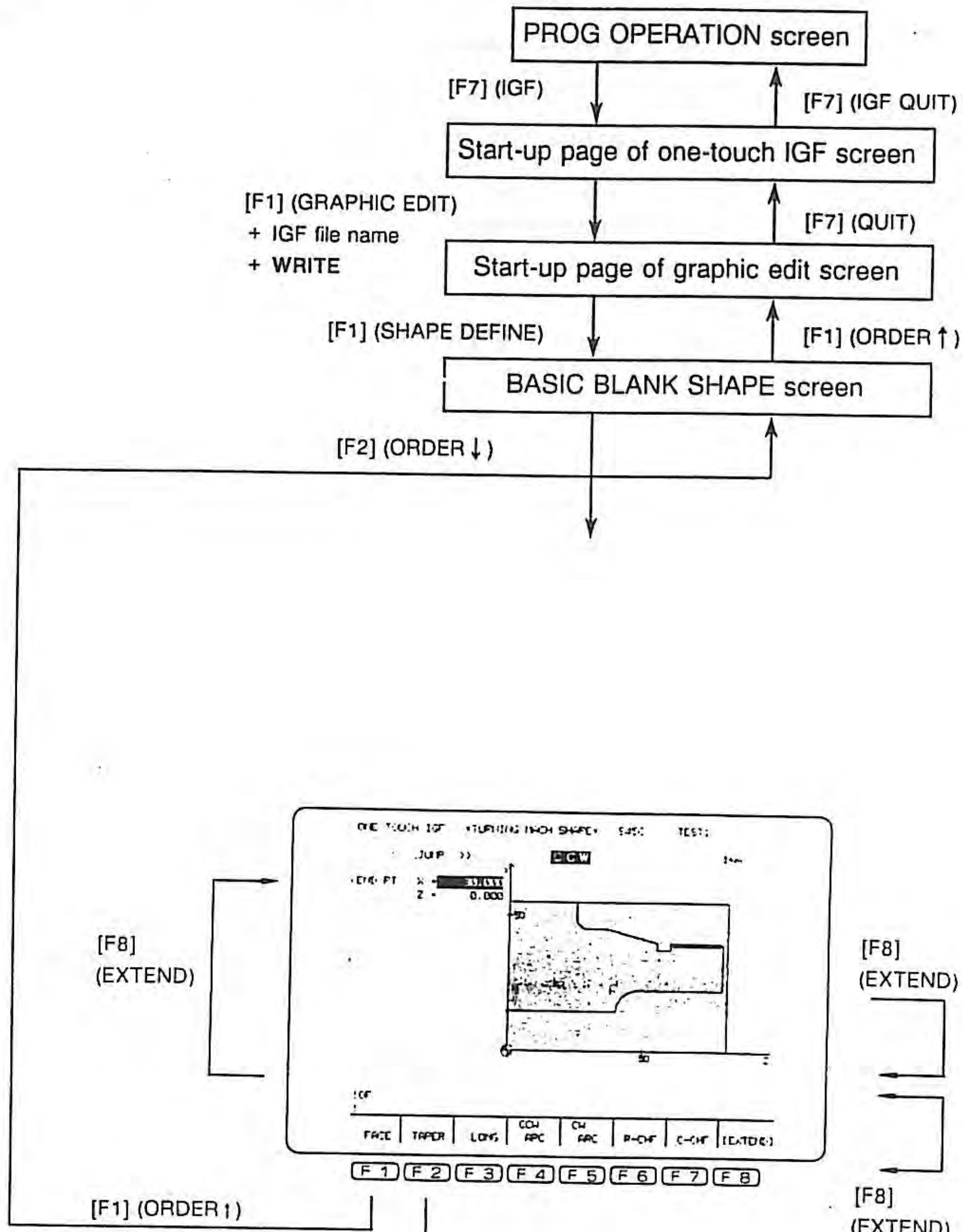
For the example, the following screens can be displayed sequentially.



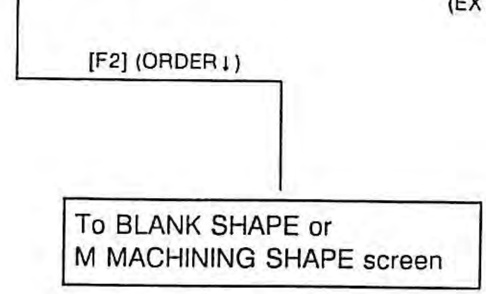
The (magenta) marker indicates where the finished shape (blue) is defined on the displayed shape element designation screen. Use the cursor control keys to move the marker.

Screen Transfer:

The TURNING MACH SHAPE screen is called from the PROG OPERATION screen as diagramed below.



When the TURNING MACH SHAPE definition process is completed, press the function key [F2] (ORDER ↓). The CRT will advance to the BLANK SHAPE screen or M MACHINING SHAPE screen. (When ROUND BAR is set for BLANK SIZE, the start-up page of the graphic edit screen will be displayed.)



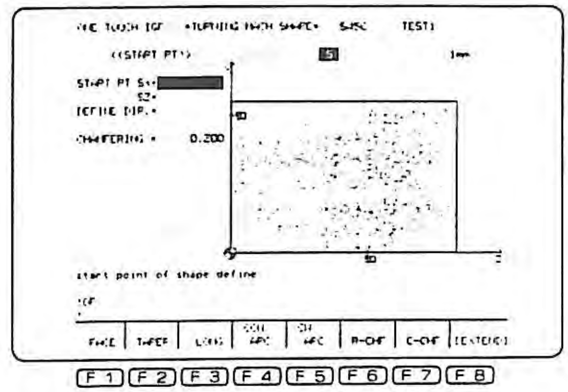
(1) Designation of Starting Point (START PT)

a) Function

START PT, DEFINE DIR., and CHAMFERING for TURNING MACH SHAPE is designated.

b) Data to be Designated

START PT SX
SZ
DEFINE DIR.
CHAMFERING



c) How to Designate

Press the function key [F2] (ORDER ↓) after the completion of BASIC BLANK SHAPE definition. The START PT screen for TURNING MACH SHAPE definition will be displayed.

Move the cursor to the required data column using the cursor control keys ↑ and ↓ (follow the prompt if displayed), and enter the data.

START PT SX, SZ

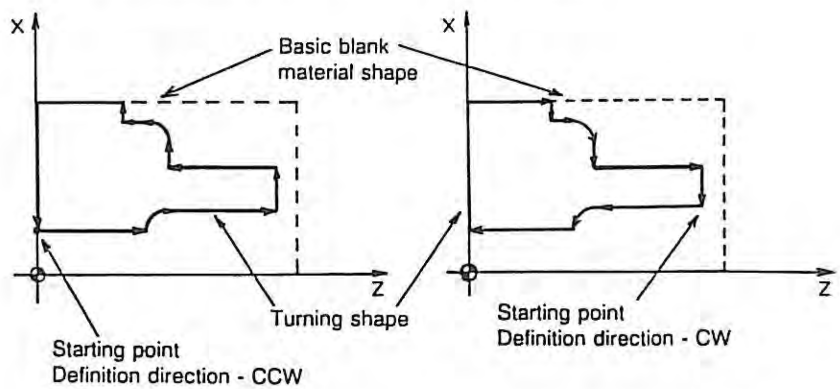
The X- (SX-) and Z- (SZ-) coordinates of the shape definition starting point for TURNING MACH SHAPE are designated.

The shape definition starting point may be located at any place on the circumference and inside of the basic blank material shape.

DEFINE DIR.

Shape definition direction for TURNING MACH SHAPE is designated. As the prompt "1 = CCW, 2 = CW" is displayed, enter "1" or "2" accordingly.

Example:



CHAMFERING

The one-touch IGF system automatically generates CHAMFERING. Designate the necessary chamfer size. On the screen, the value set by the one-touch IGF parameter (dimensions parameter No. 13 CHAMFERING) is set as initial setting.

Note 1: *Automatic Generation of Chamfering*

For the following portions, chamfering is not generated:

- *Starting point*
- *Before and after jump (Refer to (12) "Jump" on page 115.)*
- *Chamfered portion of groove*

If no data has been designated for chamfering or rounding in the groove definition process, chamfering is automatically generated. (Refer to (9) "GROOVE" on page 105.)

For detailed explanation, refer to (19) "Automatic Chamfer Generation".

Note 2: *In the turning shape definition process, the shape definition is started from the shape definition starting point. Therefore, designate the shape definition starting point (SX, SZ) and definition direction. Otherwise the turning shape definition process does not proceed.*

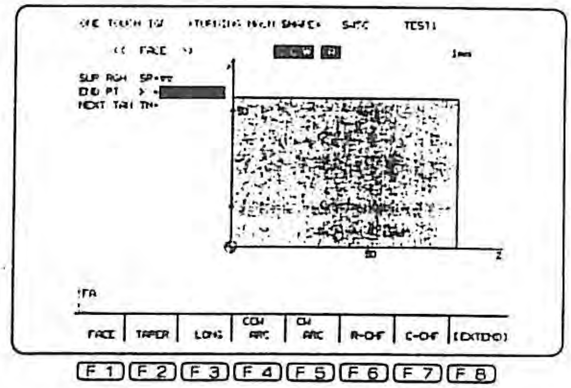
(2) FACE

a) Function

The shape of end face for TURNING MACH SHAPE is designated.

b) Data to be Designated

SUR RGH SR
END PT X
Z
NEXT TAN TN



c) How to Designate

Press the function key [F8] (EXTEND) until the function "FACE" is assigned to the function key [F1] .

Press the function key [F1] (FACE). The FACE screen will be displayed.

SUR RGH SR

Surface roughness is designated. As the prompt "**1 = ∇, 2 = ∇∇, 3 = ∇∇∇, 4 = ∇∇∇∇ 5 = ~**" will be displayed, select the desired surface roughness. For example, to select ∇∇ as surface roughness, enter "2". Enter "5" when no cutting is carried out. For the data column of SUR RGH, the following data has been set in advance.

1) When any shape has been designated prior to end face definition, the surface roughness designated for that shape is initially set.

2) When the shape of end face is designated as the first shape following the starting point, the surface roughness set at the one-tough IGF No. 15 FINISHING FEEDRATE is set initially.

When change of the surface roughness is not necessary, no data input is required. The surface roughness designated here is set for the surface roughness of the shape designated next.

END PT X, Z

The X- and Z-coordinates of end face end point are designated.

For END PT X, a diametrical value must be input.

NEXT TAN TN

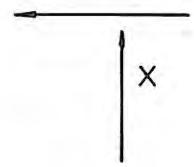
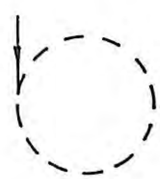
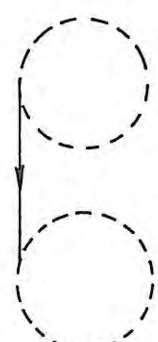
Whether or not the shape element of the end face is tangent to the shape element which is designated next is designated. As the prompt "**1 = next tangent, 2 = next not tangent**" will be displayed, select "1" or "2".

d) Conditions for Shape Determination

To determine the end face shape, enter necessary shape element data of end face.

There are however cases where all data is not necessarily to be designated. Enter only the data which is necessary to determine shape elements. If unnecessary data has been input, it will be ignored.

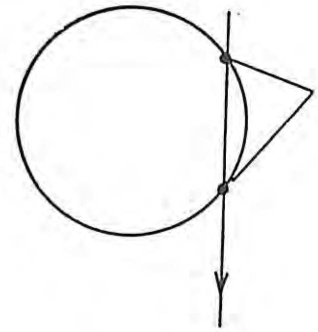
The combination of data sets necessary to define shape elements is as indicated in the table.

Shape Element = FACE	
Combination of Input Data	Conditions to be Met
① No entry	 The data for END POINT of the previous shape must be designated.
② END PT X	
③ No entry	 The data for NEXT TAN of the previous shape must be designated.
④ NEXT TAN TN	 Tangent to next shape
⑤ NEXT TAN TN	 The data for NEXT TAN of the previous shape must be designated. Tangent to next shape

Inquiry about starting point

When the end face shape tangent to the CW or CCW arc is designated and two cross points are present, the following prompt inquiring about the starting point will be displayed.

START PT : 1 = ←, 2 = →, 3 = ↓, 4 = ↑



Key in "3" when selecting the lower cross point as the starting point.

Key in "4" when selecting the upper cross point as the starting point.

Direction of end face

It is not necessary to consider the direction of end face, either upward or downward, when designating shape elements of end face. However, if the end face shape cannot be defined only by the entered data, the following prompt inquiring about the direction of end face will be displayed.

LINE DIR. : 3 = ↓, 4 = ↑

Key in "3" when the end face is in the downward direction.

Key in "4" when the end face is in the upward direction.

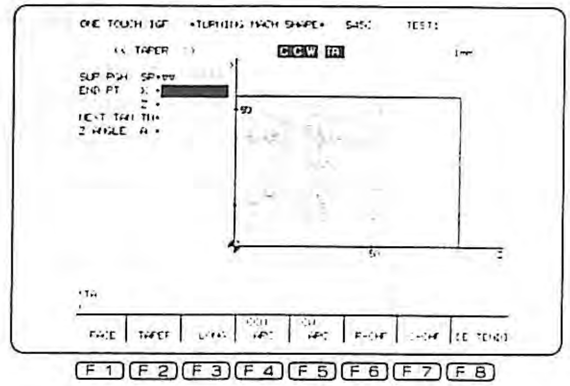
Note: *If the entered data is not sufficient or contradictive and no shape cannot be defined, an error occurs and designation of shape element is disabled. Or incorrect data has been entered, an abnormal shape will be displayed on the screen. In such cases, return the CRT back to the page where erroneous data has been entered using the cursor control key ← and correct the data accordingly.*

After the correction of data, advance the CRT to the previously displayed page using the cursor control key →.

(3) TAPER

a) Function

The shape of taper for TURNING MACH SHAPE is designated.



b) Data to be Designated

SUR RGH	SR
END PT	X
	Z
NEXT TAN	TN
Z ANGLE	A

c) How to Designate

Press the function key [F8] (EXTEND) until the function "TAPER" is assigned to the function key [F2].

Press the function key [F2] (TAPER). The TAPER screen will be displayed.

SUR RGH SR

Surface roughness is designated.

END PT X, Z

The X- and Z-coordinates of the end point of taper are designated.

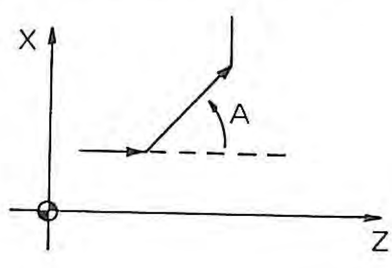
For END PT X, a diametrical value must be input.

NEXT TAN TN

Whether or not the taper is tangent to the shape which is designated next is designated. As the prompt "**1 = next tangent, 2 = next not tangent**" will be displayed, select "1" or "2".

Z ANGLE A

Taper angle in reference to the Z-axis is designated. (degrees)

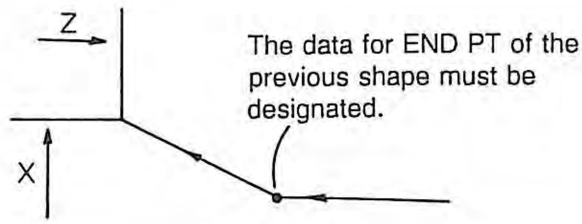
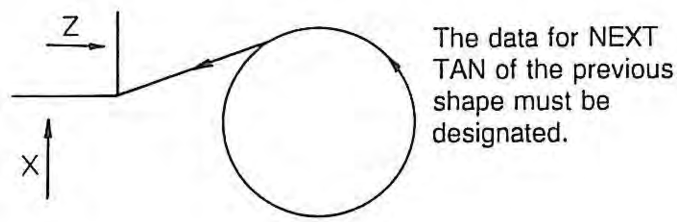
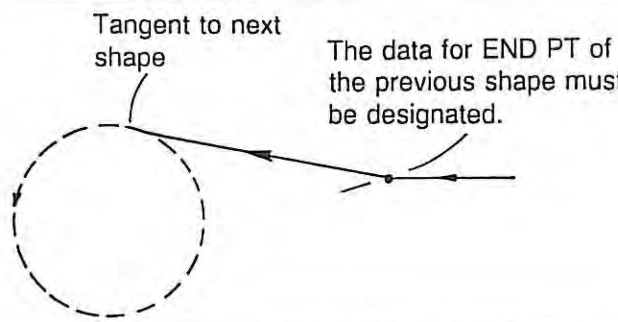
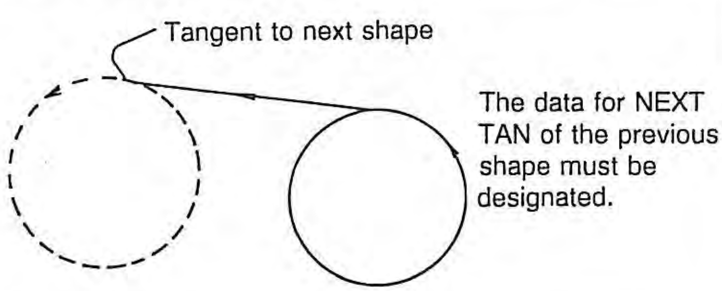


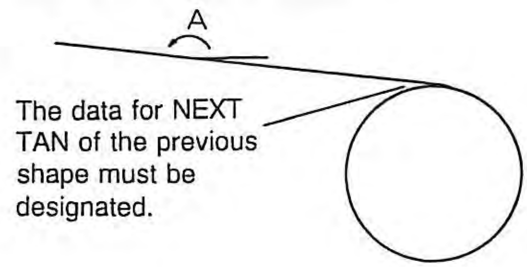
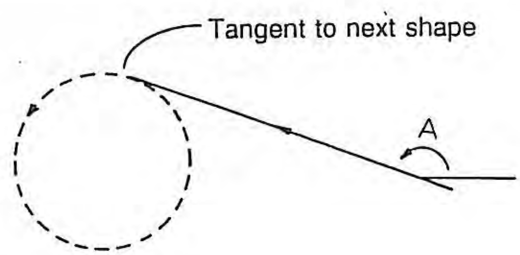
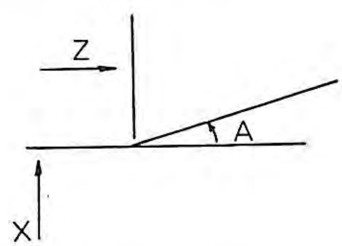
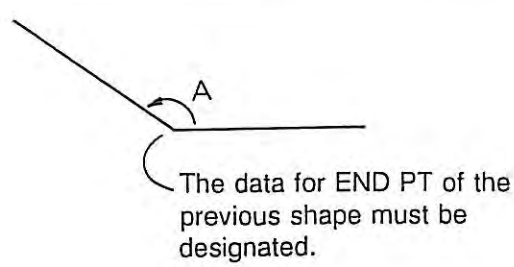
d) Conditions for Shape Determination

To determine the shape of taper, enter necessary shape element data of taper.

There are however cases where all data is not necessarily to be designated. Enter only the data which is necessary to determine shape elements. If unnecessary data has been input, it will be ignored.

The combination of data sets necessary to define shape elements is as indicated in the table.

Shape Element = TAPER	
Combination of Input Data	Conditions to be Met
① END PT X END PT Z	
② END PT X END PT Z	
③ NEXT TAN TN	
④ NEXT TAN TN	

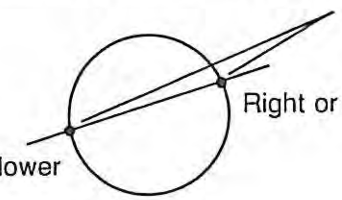
Shape Element = TAPER	
Combination of Input Data	Conditions to be Met
⑤ Z ANGLE A	
⑥ NEXT TAN TN Z ANGLE A	
⑦ END PT X END PT Z Z ANGLE A	
⑧ Z ANGLE A	

Inquiry about starting point

When the taper tangent to the CW or CCW arc is designated and two cross points are present, the following prompt inquiring about the starting point will be displayed.

START PT : 1 = ←, 2 = →, 3 = ↓, 4 = ↑

Designate the starting point by selecting a number from the prompt.



1 = ←	....	left
2 = →	....	right
3 = ↓	....	lower
4 = ↑	....	upper

Direction of taper

It is not necessary to consider the direction of taper when designating shape elements of taper. However, if the shape of taper cannot be defined only by the entered data, the following prompt inquiring about the direction of taper will be displayed.

LINE DIR. : 5 = ↖, 6 = ↙, 7 = ↘, 8 = ↗

Enter the data according to the direction of the taper to be designated.

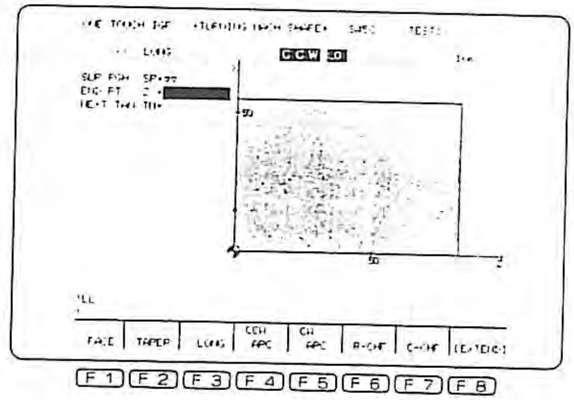
(4) LONG

a) Function

The shape in the longitudinal direction for TURNING MACH SHAPE is designated.

b) Data to be Designated

SUR RGH	SR
END PT	X
	Z
NEXT TAN	TN



c) How to Designate

Press the function key [F8] (EXTEND) until the function "LONG" is assigned to the function key [F3].

Press the function key [F3] (LONG). The LONG screen will be displayed.

SUR RGH SR

Surface roughness is designated.

END PT X, Z

The X- and Z-coordinates of the end point in the longitudinal direction are designated.

For END PT X, a diametrical value must be input.

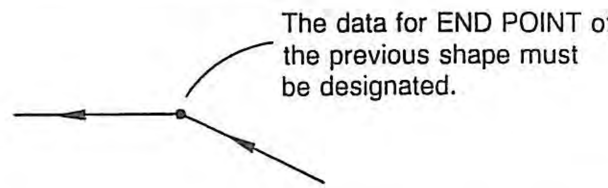
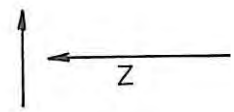
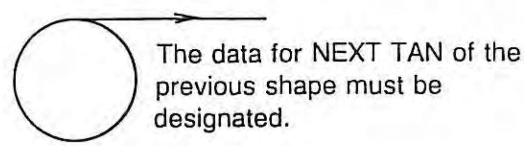
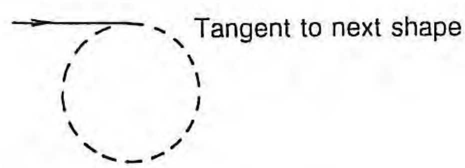
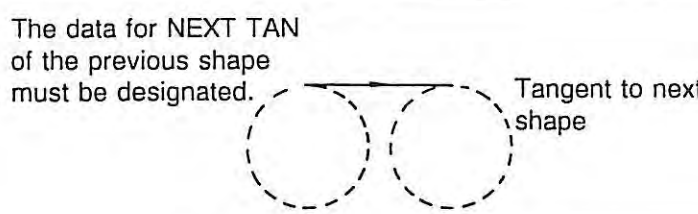
NEXT TAN TN

Whether or not the longitudinal portion is tangent to the shape which is designated next is designated. As the prompt "1 = **next tangent**, 2 = **next not tangent**" will be displayed, select "1" or "2".

d) Conditions for Shape Determination

To determine the shape in the longitudinal direction, enter necessary shape element data of longitudinal portion. There are however cases where all data is not necessarily to be designated. Enter only the data which is necessary to determine shape elements. If unnecessary data has been input, it will be ignored.

The combination of data sets necessary to define shape elements is as indicated in the table.

Shape Element = LONG	
Combination of Input Data	Conditions to be Met
① No entry	
② END PT Z	
③ No entry	
④ NEXT TAN TN	
⑤ NEXT TAN TN	

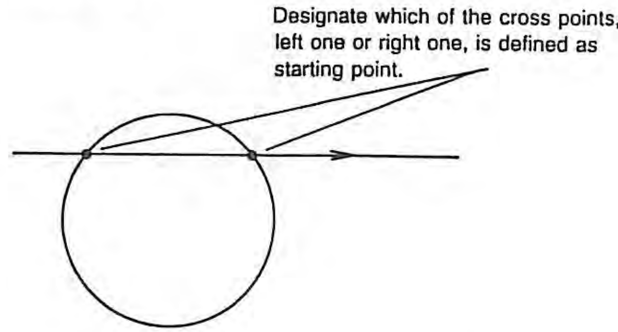
Inquiry about starting point

When the longitudinal portion tangent to the CW or CCW arc is designated and two cross points are present, the following prompt inquiring about the starting point will be displayed.

START PT : 1 = ←, 2 = →, 3 = ↓, 4 = ↑

Key in "1" when the cross point to the left is defined as starting point.

Key in "2" when the cross point to the right is defined as starting point.



Direction of longitudinal portion

It is not necessary to consider the direction of longitudinal portion when designating shape elements of longitudinal portion. However, if the shape of longitudinal portion cannot be defined only by the entered data, the following prompt inquiring about the direction of longitudinal portion will be displayed.

LINE DIR. : 1 = ←, 2 = →

Key in "1" when the longitudinal portion is in the leftward direction.

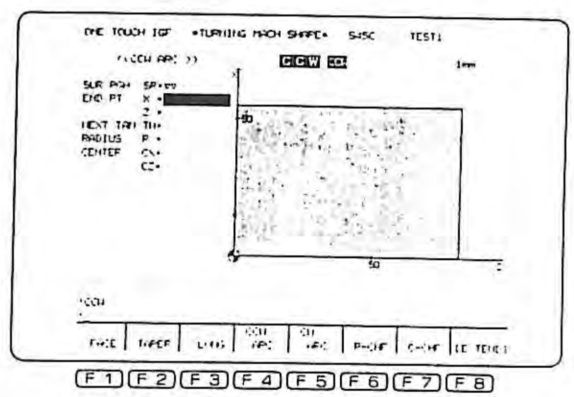
Key in "2" when the longitudinal portion is in the rightward direction.

(5) **CCW ARC**

(6) **CW ARC**

a) Function

The shape of CCW arc or CW arc for TURNING MACH SHAPE is designated.



b) Data to be Designated

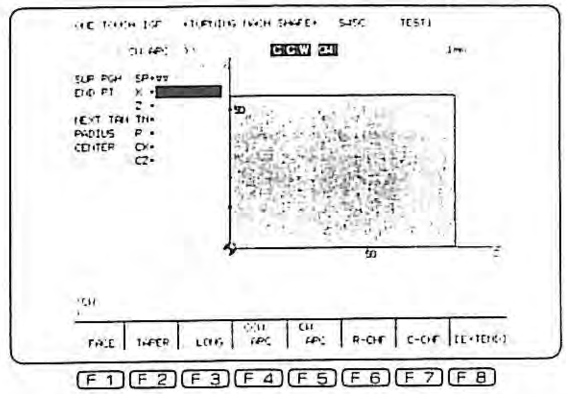
SUR RGH	SR
END PT	X
	Z
NEXT TAN	TN
RADIUS	R
CENTER	CX
	CZ

c) How to Designate

Press the function key [F8] (EXTEND) until the function "CCW ARC" is assigned to the function key [F4] and "CW ARC" to [F5].

Press the function key [F4] (CCW ARC). The CCW ARC screen will be displayed.

Press the function key [F5] (CW ARC). The CW ARC screen will be displayed.



SUR RGH SR

Surface roughness is designated.

END PT X, Z

The X- and Z-coordinates of the end point of CCW or CW arc are designated.

For END PT X, a diametrical value must be input.

NEXT TAN TN

Whether or not the arc is tangent to the shape which is designated next is designated. As the prompt "**1 = next tangent, 2 = next not tangent**" will be displayed, select "1" or "2".

RADIUS R

Radius R of the arc is designated.

CENTER CX, CZ

The X- and Z-coordinates of the center point of CCW or CW arc are designated.

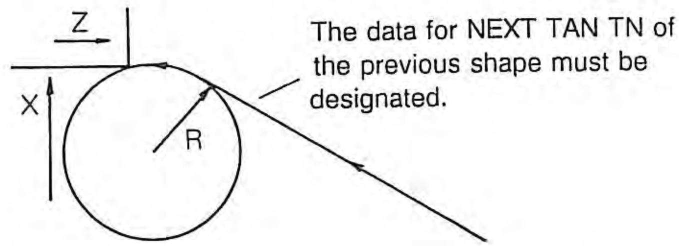
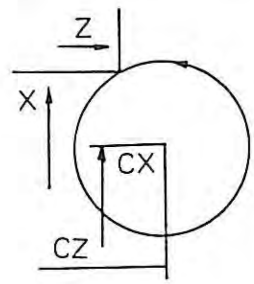
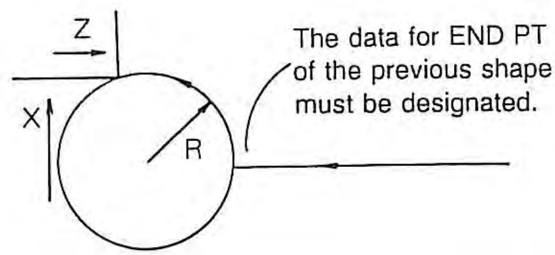
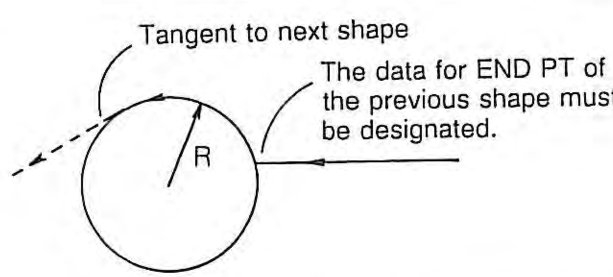
For CENTER CX, a diametrical value must be input.

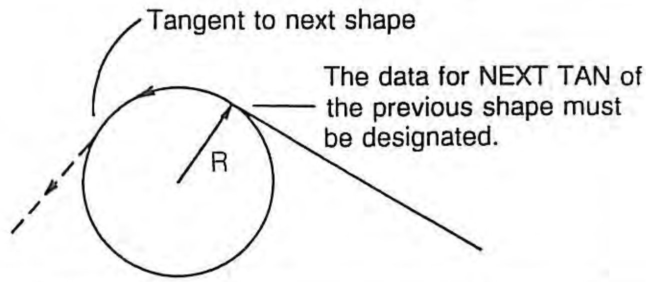
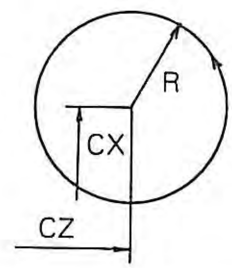
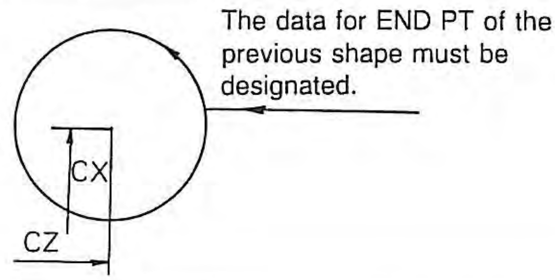
d) Conditions for Shape Determination

To determine the shape of arc, enter necessary shape element data of arc.

There are however cases where all data is not necessarily to be designated. Enter only the data which is necessary to determine shape elements. If unnecessary data has been input, it will be ignored.

The combination of data sets necessary to define shape elements is as indicated in the table.

<table>
<tr><th colspan="2">Shape Element = CCW ARC, CW ARC</th></tr>
<tr><th>Combination of Input Data</th><th>Conditions to be Met</th></tr>
<tr><td>① END PT X
END PT Z
RADIUS R</td><td>
</td></tr>
<tr><td>② END PT X
END PT Z
CENTER CX
CENTER CZ</td><td>
</td></tr>
<tr><td>③ END PT X
END PT Z
RADIUS R</td><td>
</td></tr>
<tr><td>④ NEXT TAN TN
RADIUS R</td><td>
</td></tr>
</table>

<table>
<tr><th colspan="2">Shape Element = CCW ARC, CW ARC</th></tr>
<tr><th>Combination of Input Data</th><th>Conditions to be Met</th></tr>
<tr><td>⑤ NEXT TAN TN
RADIUS R</td><td>
</td></tr>
<tr><td>⑥ RADIUS R
CENTER CX
CENTER CZ</td><td>
</td></tr>
<tr><td>⑦ CENTER CX
CENTER CZ
RADIUS R</td><td>
</td></tr>
</table>

Inquiry about starting point

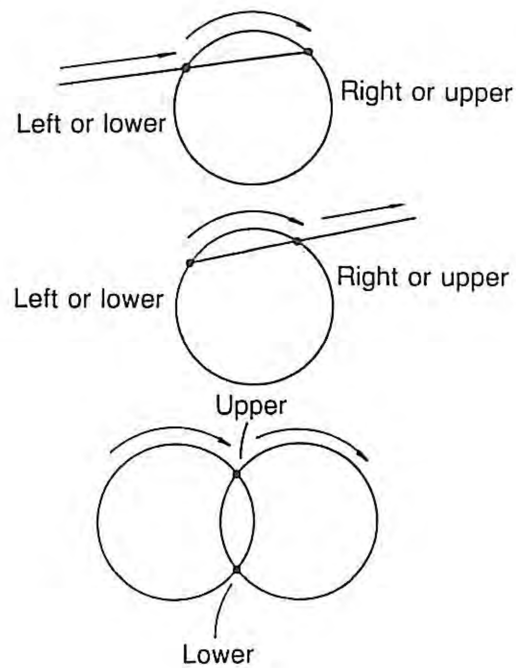
In the following cases, two cross points are present and the prompt inquiring about the starting point will be displayed.

START PT : 1 = ←, 2 = →, 3 = ↓, 4 = ↑

- An arc is designated following linear shape (FACE, TAPER, LONG).
- Linear shape is designated following an arc.
- An arc is designated following another arc.

Designate the starting point by selecting a number from the prompt.

1 = ← ...... left
2 = → ...... right
3 = ↓ ...... lower
4 = ↑ ...... upper



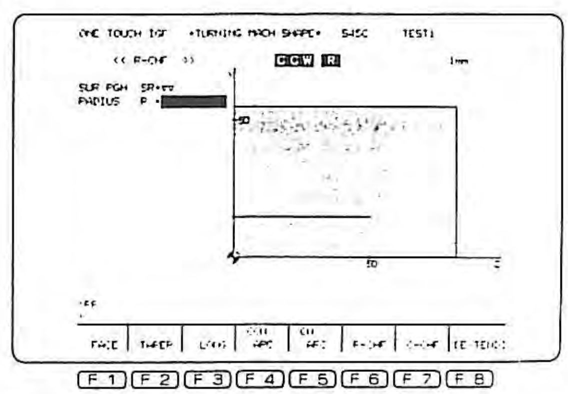
(7) R-CHF

a) Function

The shape of rounding R for TURNING MACH SHAPE is designated.

b) Data to be Designated

SUR RGH SR
RADIUS R



c) How to Designate

Press the function key [F8] (EXTEND) until the function "R-CHF" is assigned to the function key [F6].

Press the function key [F6] (R-CHF). The R-CHF screen will be displayed.

SUR RGH SR

Surface roughness is designated.

RADIUS R

Radius of rounding R is designated.

d) Conditions for Shape Determination

To determine the shape of rounding R, enter necessary shape element data of rounding R.

The shape of rounding R is displayed on the screen when the shape preceding and following rounding R has been defined.

If the preceding and following shape elements have two cross points, rounding is defined at the cross point selected in response to the prompt given.

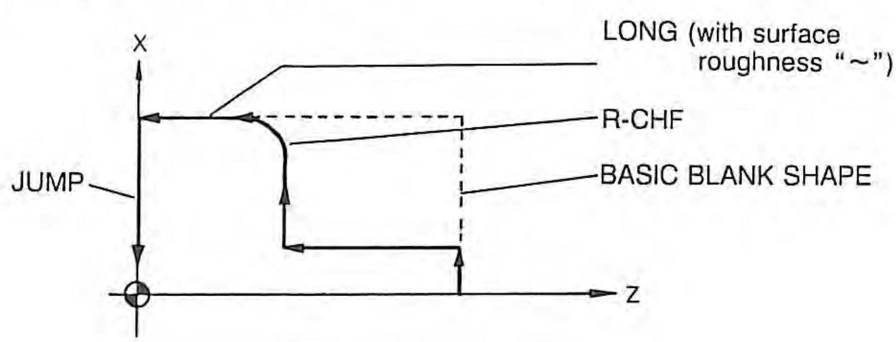
Sections indicated by the symbol "{" are not displayed unless the end point is given.

Note 1: *The following shape cannot be designated preceding or following the rounding R.*

Jump
Rounding R
Chamfering C
Recess
Groove (Rounding R can be designated in the groove shape definition process.)

Also note rounding R cannot be designated as the first shape following the shape definition starting point or the last shape preceding the shape definition end point.

Note 2: *When shape elements for rounding R should be designated preceding or following the jump, proceed as follows.*



(8) CHAMFER

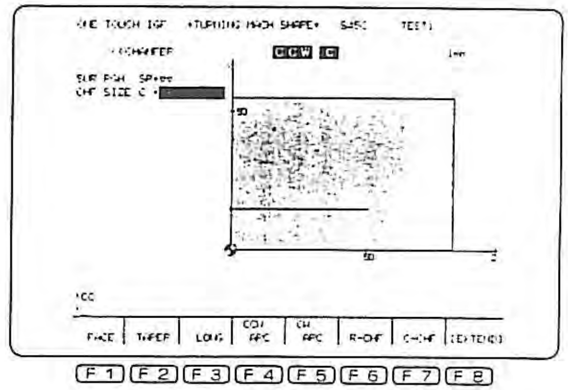
a) Function

The shape of chamfer C for TURNING MACH SHAPE is designated.

This function is also used to automatically generate chamfer.

b) Data to be Designated

SUR RGH SR
CHF SIZE C



c) How to Designate

Press the function key [F8] (EXTEND) until the function "C-CHF" is assigned to the function key [F7].

Press the function key [F7] (C-CHF). The C-CHF screen will be displayed.

SUR RGH SR

Surface roughness is designated.

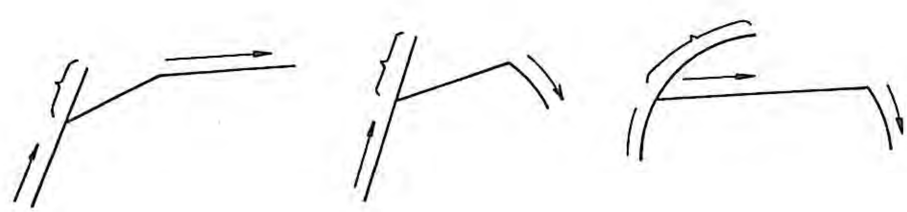
CHF SIZE C

Chamfer size is designated.

d) Conditions for Shape Determination

To determine the shape of chamfering C, enter necessary shape element data of chamfering C.

The shape of chamfering C is displayed on the screen when the shape preceding and following chamfering C has been defined. If the preceding and following shape elements have two cross points, chamfering is defined at the cross point selected in response to the prompt given.



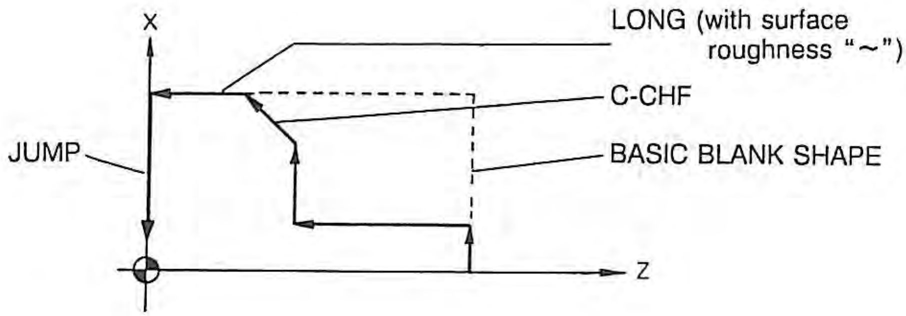
Sections indicated by the symbol "{" are not displayed unless the end point is given.

Note 1: *The following shape cannot be designated preceding or following the chamfering C.*

Jump
Rounding R
Chamfering C
Recess
Groove (Chamfering C can be designated in the groove shape definition process.)

Also note chamfering C cannot be designated as the first shape following the shape definition starting point or the last shape preceding the shape definition end point.

Note 2: *When shape elements for chamfering C should be designated preceding or following the jump, proceed as follows.*



Note 3: *To automatically generate chamfer, press the function key [F7] (C-CHF) after the completion of turning shape definition.*

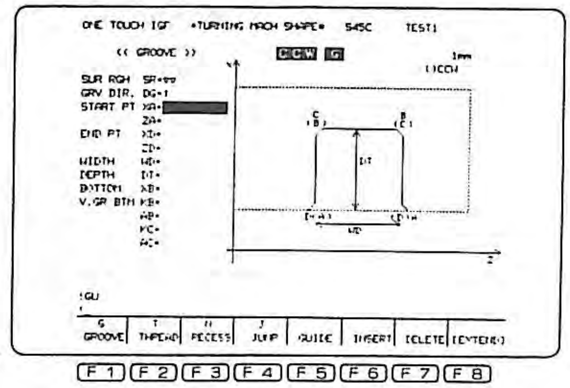
(9) GROOVE

a) Function

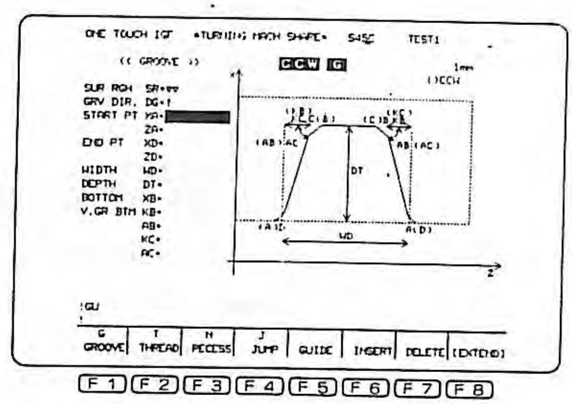
The shape of groove for TURNING MACH SHAPE is designated.

b) Data to be Designated

SUR RGH	SR
GRV DIR.	DG
START PT	XA
	ZA
END PT	XD
	ZD
WIDTH	WD
DEPTH	DT
BOTTOM	XB (ZB)
V.GR BTM	KB (IB)
	AB
	KC (IC)
	AC
Number of grooves	NG
Shift amount	K (I)
Chamfering at point A	AK
	AS
Chamfering at point B	BK
	BS
Chamfering at point C	CK
	CS
Chamfering at point D	DK
	DS



(Guide drawing is displayed.)



(Guide drawing is displayed.)

* Positions of points A, B, C, and D change according to the shape definition direction selected.

c) How to Designate

Press the function key [F8] (EXTEND) until the function "G GROOVE" is assigned to the function key [F1].

Press the function key [F1] (G GROOVE). The GROOVE screen will be displayed.

SUR RGH SR

Surface roughness is designated.

GRV DIR. DG

Infeed direction of groove cutting is designated. As the prompt "1 = ↓, 2 = ↑, 3 = ←" is displayed, select the number corresponding to the infeed direction. For example, select "1" for OD grooving and "2" for ID grooving.

START PT XA, ZA

The X- and Z-coordinates of the starting point of groove shape (point A on the guide drawing) are designated.

For START PT ZA, a diametrical value must be input.

END PT XD, ZD

The X- and Z-coordinates of the end point of groove shape (point D on the guide drawing) are designated.

When the data has been set for WIDTH WD, no data input is required. The shape preceding the groove has not been defined, designate both END PT and WIDTH WD.

For END PT XD, a diametrical value must be input.

WIDTH WD (no symbol)

The width of groove is designated. When the data has been set for END PT XD and END PT ZD, no data input is required. The shape preceding the groove has not been defined, designate both END PT and WIDTH WD.

When the groove direction is ←, enter the value in radius.

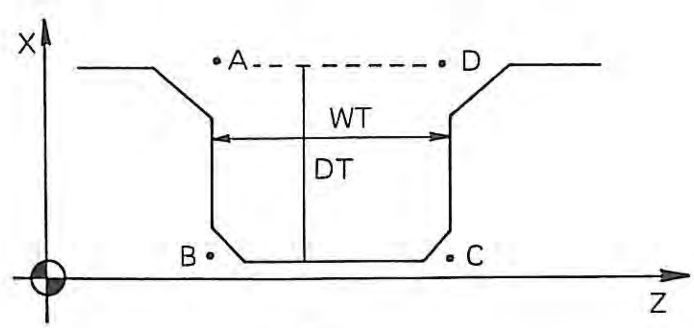
DEPTH DT

The depth of groove is designated. When the data has been set for BOTTOM XB (ZB), no data input is required. When the groove direction is either ↓ or ↑, enter the value in radius.

BOTTOM XB (ZB)

The coordinate value of the end face of groove is designated. Enter the data for BOTTOM XB when the groove direction is either ↓ or ↑, and enter the data for BOTTOM ZB when the groove direction is ←.

When the data has been set for DEPTH DT, no data input is required. A diametrical value must be input for BOTTOM XB.



V. GR BTM KB (IB)
AB
KC (IC)
AC

When designating the shape of V-groove, enter the data as follows.

KB .... Difference between the Z-coordinates of point A and point B
(No data input is required when AB has been designated.)

IB .... Difference between the X-coordinates of point A and point B
(No data input is required when AB has been designated.)
Enter the value in radius.

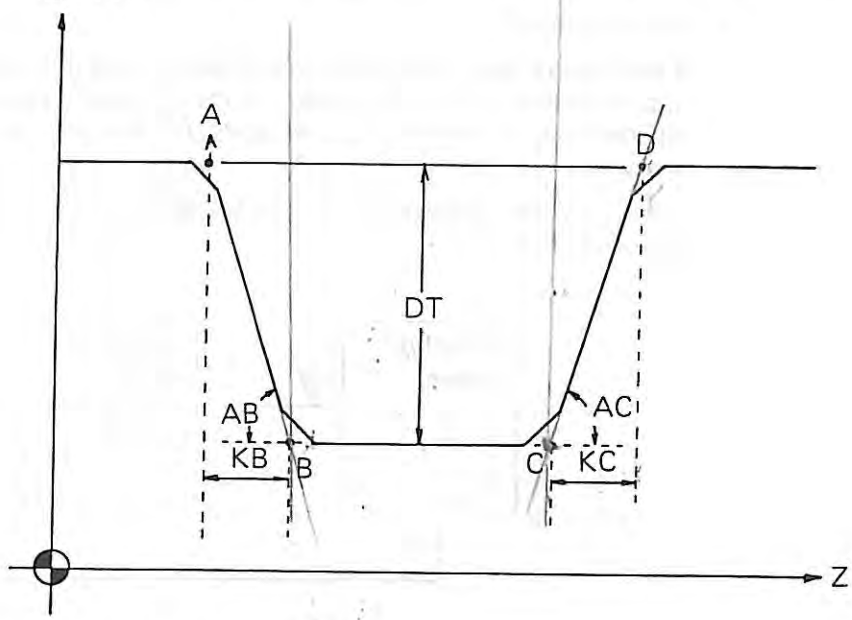
AB .... Slope of V-groove between point A and point B
(No data input is required when KB and IB have been designated.)

KC .... Difference between the Z-coordinates of point C and point D
(No data input is required when AC has been designated.)

IC .... Difference between the X-coordinates of point C and point D
(No data input is required when AC has been designated.)
Enter the value in radius.

AC .... Slope of V-groove between point C and point D
(No data input is required when KC and IC have been designated.)

* When KB and IB (KC, IC) have been designated, the value entered for AB (AC) is ignored.

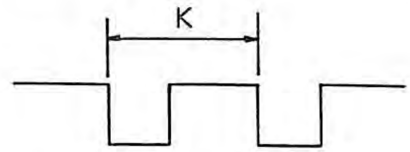


Number of grooves NG

The number of grooves is designated in the case of continuous groove cutting. The default can be set by the integer parameter No. 14 NO. OF GROOVE.

Shift amount K (I) (no symbol)

The interval between two grooves is designated in the case of continuous groove cutting. Designate the distance between the two starting points.



The sequence for continuous groove cutting can be set at the integer parameter No. 30 CONTINUOUS GROOVING DIR. (0 = → ↑ , 1 = ← ↓).

Chamfering at point A AK
AS

Chamfering at point B BK
BS

Chamfering at point C CK
CS

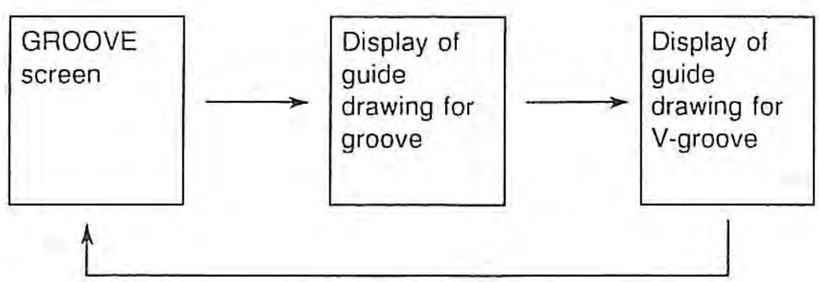
Chamfering at point D DK
DS

The type of chamfer and the chamfer size for each of points A, B, and C are designated.

d) Guide Drawing

If the function key [F5] (GUIDE) is pressed when designating the groove shape elements, a guide drawing will be displayed. Since the guide drawing displayed varies depending on the direction of groove, first designate the direction of groove and then call the guide drawing.

Each time the function key [F5] (GUIDE) is pressed, the CRT will be changed as diagramed below.



(10) **THREAD**

a) Function

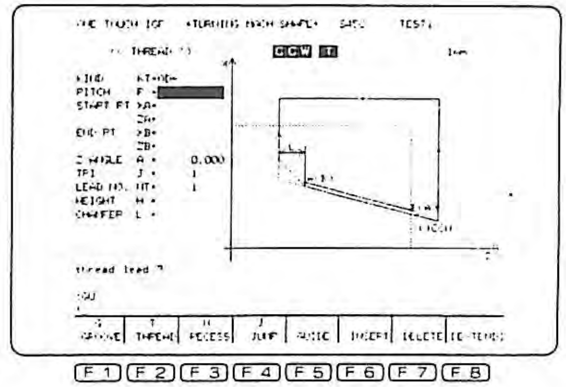
The shape of thread/tap for TURNING MACH SHAPE is designated.

Thread

b) Data to be Designated

KIND	KT
PITCH	P
START PT	XA
	ZA
END PT	XB
	ZB
Z ANGLE	A
TPI	J
LEAD NO.	NT
HEIGHT	H
CHAMFER	L

* The positions of starting point A and end point B are reverse according to the shape definition direction.



(Guide drawing is displayed.)

c) How to Designate

Press the function key [F8] (EXTEND) until the function "T THREAD" is assigned to the function key [F2].

Press the function key [F2] (T THREAD). The THREAD screen will be displayed.

In the one-touch IGF, designation of right or left thread is specified by the one-touch IGF parameter (SPINDLE ROT of SPINDLE CONTROL PARAMETER). As the initial setting is "right thread (CW)", right thread is selected whichever type of thread, OD← or OD→, is designated. To select left thread, change the setting of parameter or correct the spindle rotating direction of the thread cutting process in the process edit mode.

KIND KT

The type of thread is designated. As the prompt "**1 = OD←, 2 = ID←, 3 = FACE ↓, 4 = OD→, 5 = ID→, 6 = FACE ↑, 7 = TAP←**" is displayed, enter the number corresponding to the thread to be cut.

For 7 = TAP←, refer to the explanation on pages 112.

PITCH P

The thread pitch is designated.

The lead pitch is designated in the case of multi-thread cutting.

START PT XA, ZA

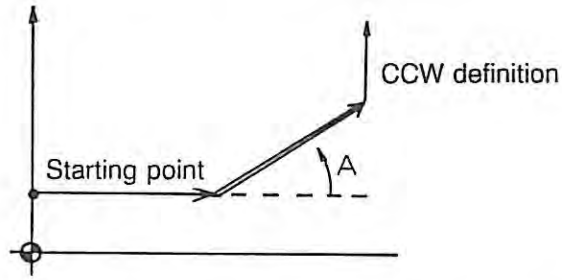
The X- and Z-coordinates of the starting point of thread shape (point A on the guide drawing) are designated.

END PT XB, ZB

The X- and Z-coordinates of the end point of thread shape (point B on the guide drawing) are designated.

Z ANGLE A

The angle A between the thread cutting direction and the Z-axis is designated.



TPI J

The number of threads per pitch is designated. The default can be set by integer parameter No. 12 PRESET VALUE OF THREAD LEAD PER INCH. J.

By changing the settings for PITCH P and TPI, selection of metric or inch thread is possible.

Example:

Metric thread

{ P = 2 (pitch)
J = 1 (number of threads per pitch) ⟶ Metric thread with pitch 2

Inch thread

{ P = 25.4 (pitch)
J = 8 (number of threads per pitch) ⟶ Inch thread with eight threads per inch

To insert the shape of the thread, input the value P/J for P, and "1" for J.

LEAD NO. NT

The number of leads is designated in the case of multi-lead thread cutting. The default can be set by integer parameter No. 13 PRESET VALUE OF THREAD LEAD NO. NT.

HEIGHT H

The height of thread is designated. The values calculated in the following formula are preset. Enter the diametrical value.

- OD threading H = P/(J*NT)*(factor for OD threading height)
- ID threading H = P/(J*NT)*(factor for ID threading height)
- FACE threading H = P/(J*NT)*(factor for FACE threading height)

Factor for each threading height can be set by the percent parameter.

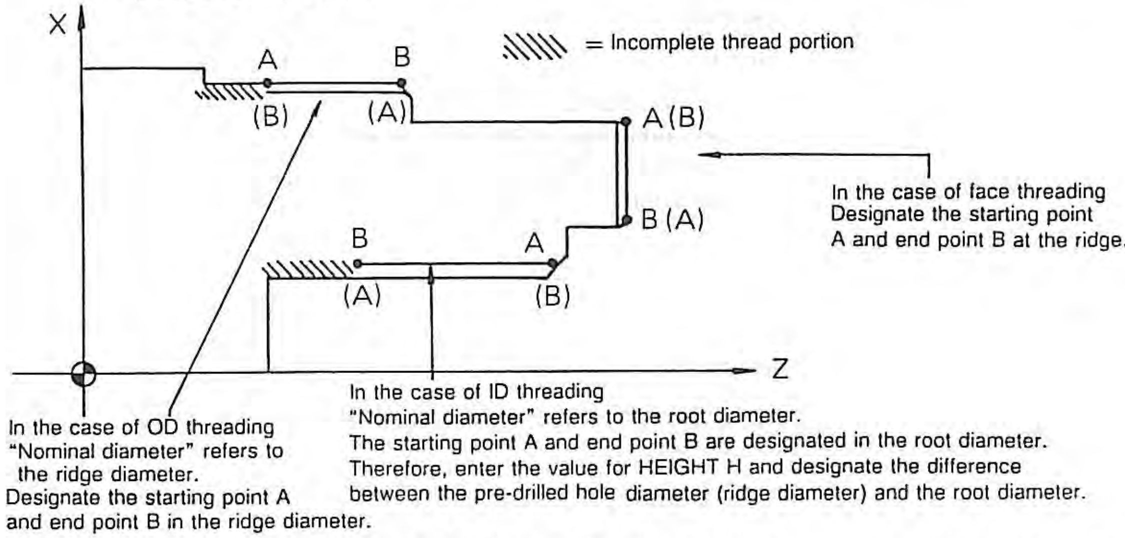
- Percent parameter No. 17 HEIGHT OF SCREW THREAD (EXTERNAL)
- Percent parameter No. 18 HEIGHT OF SCREW THREAD (INTERNAL)
- Percent parameter No. 19 HEIGHT OF SCREW THREAD (END FACE)

CHAMFER L

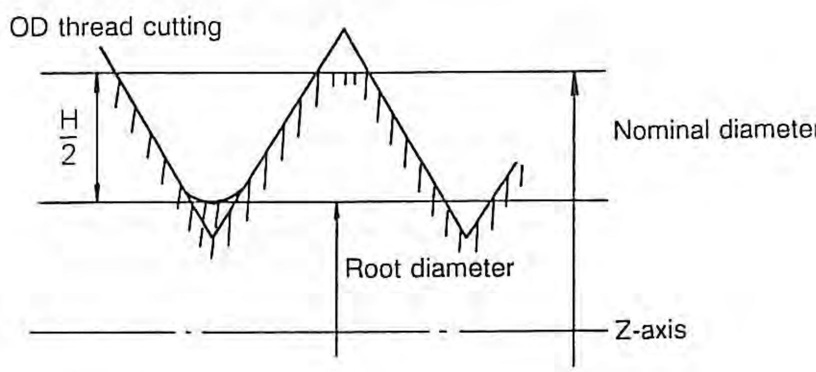
Distance L displayed on the guide drawing is designated. The default calculated in the following formula using the setting of percent parameter No. 16 THREAD CHAMFERING DISTANCE (*PITCH) is preset.

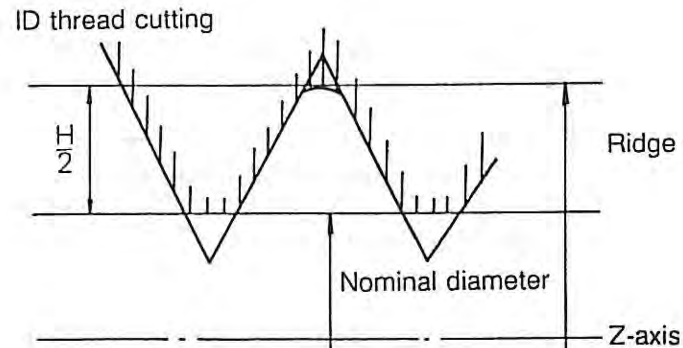
- CHAMFERING DISTANCE L = (P/J)*(setting of percent parameter No. 16)

Note 1: *When designating the START PT XA, ZA and END PT XB, ZB, use the nominal diameter. Both the starting point and the end point must be located at the complete thread portion.*



The shape preceding and following the thread shape must be understood as incomplete thread portion.





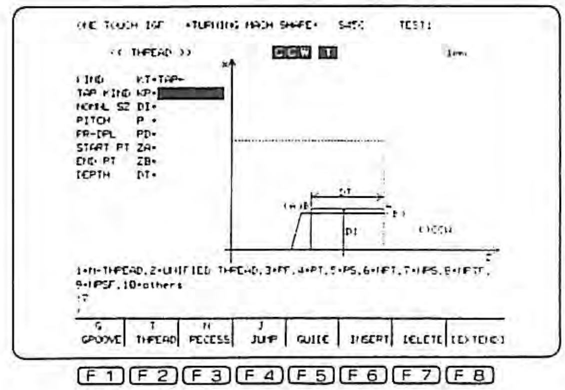
Note 2: *Thread cutting mode and thread cutting pattern can be designated by the one-touch IGF integer parameters. (integer parameters No. 17 and No. 18)*

Tapping

b) Data to be Designated

KIND	KT
TAP KIND	KP
NOMINAL SZ	DI
PITCH	P
PR-DRL	PD
START PT	ZA
END PT	ZB
DEPTH	DT

* The positions of starting point A and end point B are reverse according to the shape definition direction.



(Guide drawing is displayed.)

c) How to Designate

Press the function key [F8] (EXTEND) until the function "T THREAD" is assigned to the function key [F2].

Press the function key [F2] (T THREAD). The THREAD screen will be displayed.

Designate "7 = TAP←" for KIND KT.

The shape element designation screen for "tap" will be displayed. The word "tap" used here refers to the tapping operation in which no revolving tool is used and the workpiece is rotated.

KIND KT

The type of thread is designated. Enter "7" to select "7 = TAP←".

TAP KIND KP

The type of tap is designated. The prompt "**1 = M-THREAD, 2 = UNIFIED THREAD, 3 = PF, 4 = PT, 5 = PS, 6 = NPT, 7 = NPS, 8 = NPTF, 9 = NPSF, 10 = others**" is displayed. Enter the number corresponding to the thread to be cut. For example, enter "1" to select "1 = M-THREAD (metric thread)".

NOMNL SZ DI

The nominal diameter of tap is designated. Select the number from the prompt displayed and enter it. When "10 = others" has been designated for TAP KING KP, enter the diameter of the tapping tool.

PITCH P

The pitch is designated. When the standard thread data has been designated for TAP KIND KP and NOMNL SZ DI, the value for PITCH P is automatically set on the basis of the standard hole data.

PR-DRL PD

The pre-drilled diameter of tap is designated.

When the standard thread data has been designated for TAP KIND KP and NOMNL SZ DI, the value for PR-DRL PD is automatically set on the basis of the standard hole data.

START PT ZA

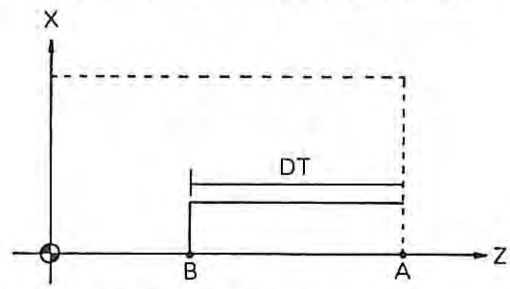
The Z-coordinate of the tapping starting point (point A on the guide drawing) is designated.

END PT ZB

The Z-coordinate of the tapping end point (point B on the guide drawing) is designated.

DEPTH DT

The distance from the tapping starting point (depth) is designated. No data input is required when END PT ZB has been designated.



Note 1: *The NC program format for tapping can be changed by changing the setting at the one-touch IGF parameters.*

i) When the setting at the integer parameter No. 29 THREAD, TAPPING CYCLE OUTPUT OR NOT (G77) is "1" (initial setting), the tapping program is output in the G77 fixed cycle format.

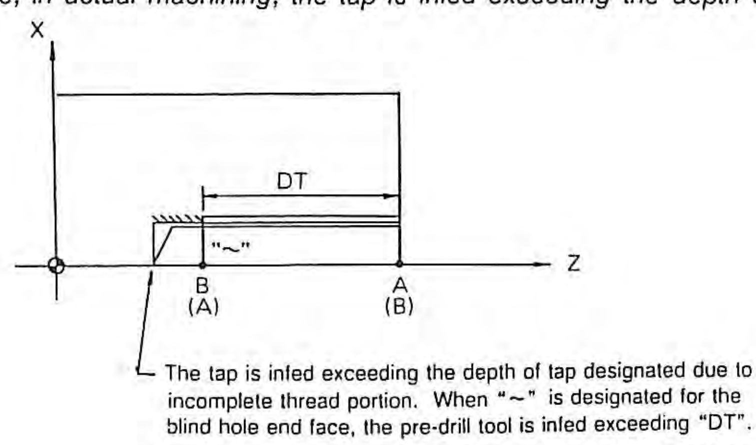
G77 X Z F

ii) When the setting at the integer parameter No. 29 THREAD, TAPPING CYCLE OUTPUT OR NOT (G77) is "0", the tapping program is output in the following format.

Spindle forward rotation	*M03*
Positioning	*G00*
Z-axis feed to the target point	*G01*
Spindle stop	*M05*
Spindle reverse rotation	*M04*
Z-axis feed to the cutting starting point	*G01*
Spindle stop	*M05*

The single block ignore M code (M164/M165) and the override ignore M code (M48/M49, M160/M161) are also output.

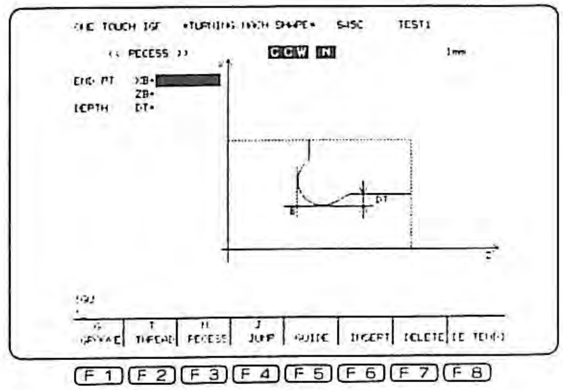
Note 2: *The starting point, end point, and depth of tap are designated in complete thread portion. Therefore, in actual machining, the tap is infed exceeding the depth of tap designated.*



(11) RECESS

a) Function

The shape of recessing for TURNING MACH SHAPE is designated.



b) Data to be Designated

END PT XB
ZB
DEPTH DT

c) How to Designate

Press the function key [F8] (EXTEND) until the function "N RECESS" is assigned to the function key [F3].

Press the function key [F3] (N RECESS). The RECESS screen will be displayed.

END PT XB, ZB

The X- and Z-coordinates of the end point of recessing are designated. For END PT XB, enter the diametrical value. Recessing in the arbitrary direction can also be designated.

DEPTH DT

The depth of recess is designated in radius. When designating the shape elements of recess by DEPTH DT, the recessing direction is determined by the shape preceding and following the recess, and recessing is made in 45° angle direction. Therefore actual depth is $\sqrt{2} \times$ DT.

d) Conditions for Shape Definition

The shape of recessing can be defined by designating either END PT XB, ZB or DEPTH DT. As the recessing direction is determined by the shape preceding and following the recess, the shape of defined recess will be displayed on the screen when the shape preceding and following the recess has been defined.

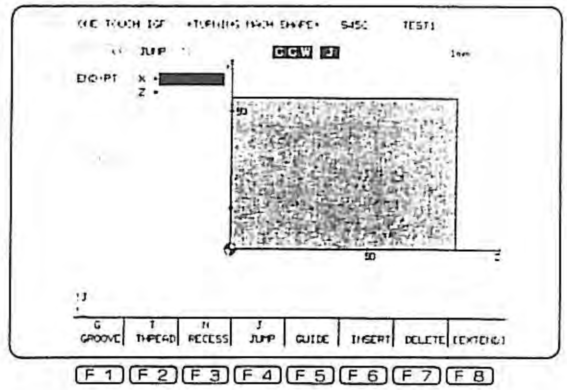
Note 1: *When END PT XB and END PT ZB have been designated, the value entered for DEPTH DT is ignored.*

Note 2: *The following shape cannot be designated preceding or following the recess.*

Jump
Round R
Chamfer C
Groove
Thread
Tap
Recess

Also note recess cannot be designated as the first shape following the shape definition starting point or the last shape preceding the shape definition end point.

(12) JUMP



a) Function

The shape of the portion where no cutting operation is made is designated.

b) Data to be Designated

END PT X
Z

c) How to Designate

Press the function key [F8] (EXTEND) until the function "J JUMP" is assigned to the function key [F4].

Press the function key [F4] (J JUMP). The JUMP screen will be displayed.

END PT X, Z

The X- and Z-coordinates of the end point of jump are designated. For END PT X, enter the diametrical value.

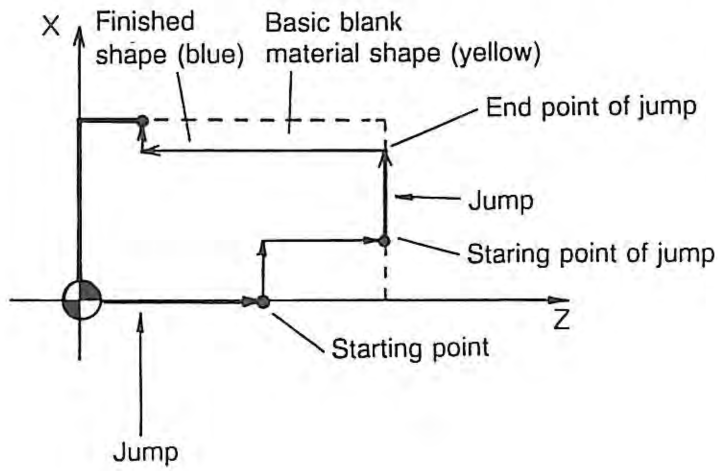
Through designating the end point, the contour of basic blank material shape is designated.

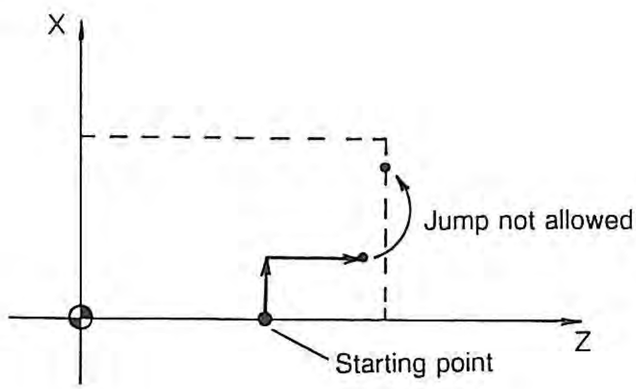
d) Conditions for Shape Definition

The shape of jump can be defined by designating both END PT X and END PT Z.

Note 1: *The shape of jump can only be designated on the contour of basic blank material shape. (on the contour of the shape displayed in yellow on the screen.) In other words, the starting and end points of jump must be located on the contour of basic blank material shape.*

Example:

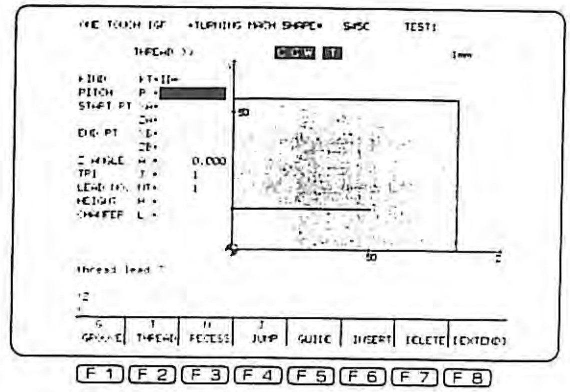




(13) Display of Guide Drawing

a) Function

The guide drawing for designating shape elements is displayed.



b) How to Designate

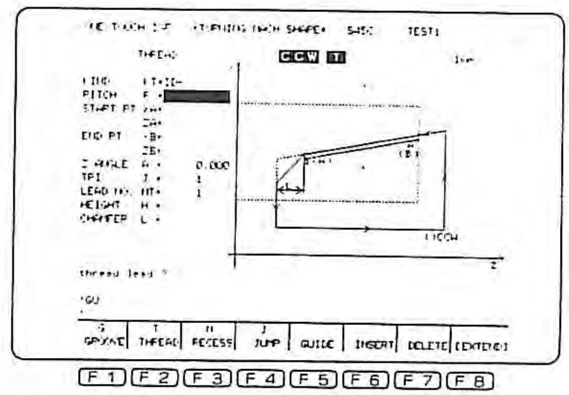
Press the function key [F8] (EXTEND) until the function "GUIDE" is assigned to the function key [F5].

Press the function key [F5] (GUIDE). The guide drawing will be displayed on the screen.

The display of guide drawing is turned on/off each time the function key [F5] (GUIDE) is pressed.

Note 1: *Guide drawing is displayed when designating the shape elements for groove, thread (tap), and recess.*

Note 2: *Since the guide drawing displayed varies depending on the direction of groove, first designate the direction of groove and then call the guide drawing. Another guide drawing is displayed when the direction of groove has been changed during shape element designation of groove.*



(Guide drawing is displayed.)

Note 3: *Since the guide drawing displayed varies depending on the type of thread selected, first designate the type of thread and then call the guide drawing. Another guide drawing is displayed when the type of thread has been changed during shape element designation of thread.*

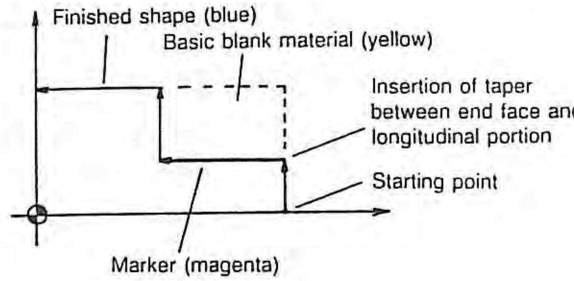
(14) **Insertion of Shape Element**

a) Function

Shape element is inserted.

b) How to Designate

For the example, shape element is inserted as follows.



Example 1:

1) The finished shape which have been defined is displayed (in blue color) on the contour of basic blank material shape (yellow) on the screen.

   Insert "taper" for example.

2) Using the cursor control keys, locate the marker (magenta) at any position of the shape following the taper.

   FACE —— $\boxed{\text{LONG}}$ —— FACE —— LONG

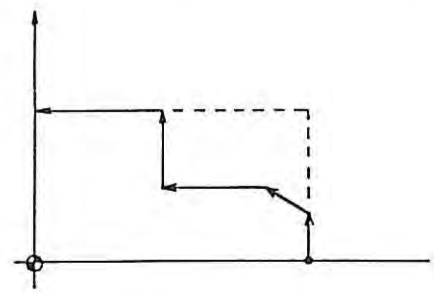
   Locate the marker at any position of LONG.

3) Press the function key [F8] (EXTEND) until the function "INSERT" is assigned to the function key [F6].

4) Press the function key [F6] (INSERT) to establish the insert mode. (The indication "INSERT" will be displayed on the screen.)

5) Enter the shape elements of taper.

   Pressing the cursor control key ← or → will cancel the insert mode and the shape inserted will be displayed.



FACE — TAPER — LONG — FACE — LONG

Insertion of groove, thread, or tap is also possible. Note that insertion of these shapes is possible only on the shape stated below.

Groove .... LONG, FACE,

Thread .... LONG, FACE, TAPER

Tap .... LONG

Example 2:

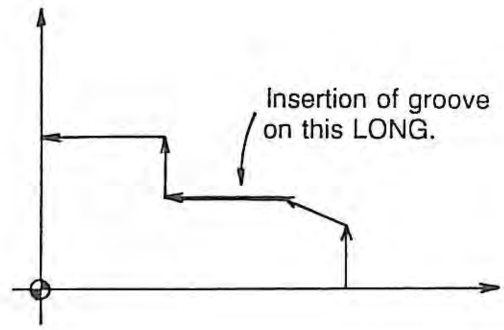
Insertion of groove

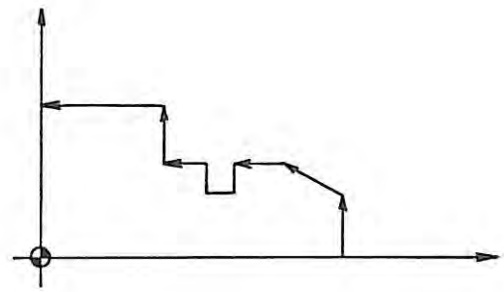
1) Using the cursor control keys, locate the marker (magenta) at any position of LONG where groove is to be inserted.

2) Press the function key [F6] (INSERT) to establish the insert mode. (The indication "INSERT" will be displayed on the screen.)

3) Enter the shape elements of groove.

4) Pressing the cursor control key ← or → will cancel the insert mode and the shape inserted will be displayed.





Note 1: *The insert mode can be canceled by pressing the function key [F6] (INSERT) again.*

Note 2: *There are cases where shape definition is disabled due to the insertion of shape. In such cases, call the necessary shape element designation screen using the cursor control keys ← and →, and correct the data as required.*

(15) **Deletion of Shape Element**

a) Function

The shape in the longitudinal direction for TURNING MACH SHAPE is designated.

b) How to Designate

For example, shape element is deleted as follows.

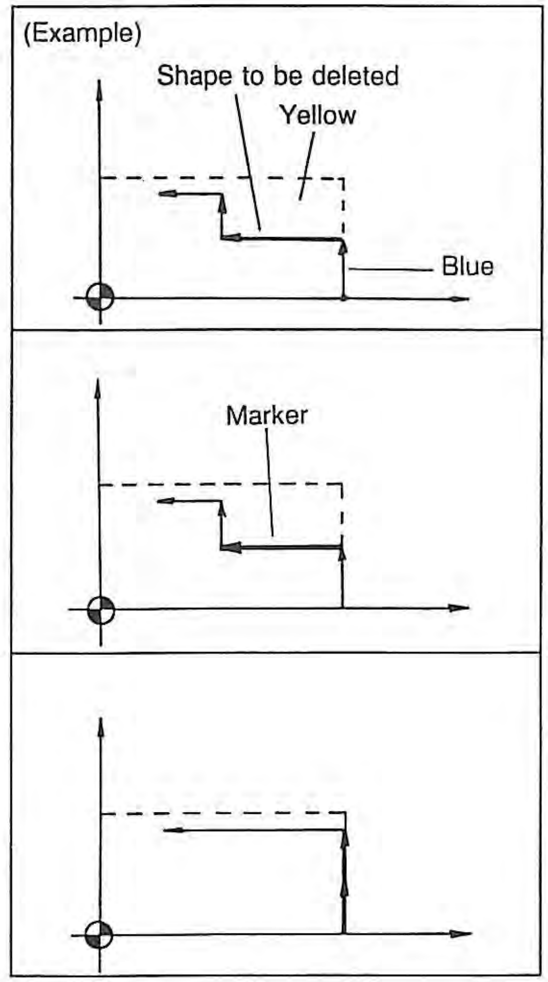
1) The finished shape which have been defined is displayed (in blue color) on the contour of basic blank material shape (yellow) on the screen.

2) Using the cursor control keys, locate the marker (magenta) at any position of the shape to be deleted.

The shape element designation screen of the shape to be deleted will be displayed.

3) Press the function key [F8] (EXTEND) until the function "DELETE" is assigned to the function key [F7].

4) Press the function key [F7] (DELETE). The shape designated in step 2) will be displayed.



Note: *After the deletion of shape element, there are cases where no shape is displayed on the screen since the shape cannot be defined due to the deleted shape. In such cases, call the necessary shape element designation screen using the cursor control keys ← and →, and correct the data as required.*

(16) Re-display of Shape

a) Function

The shape defined by entered data is re-displayed.

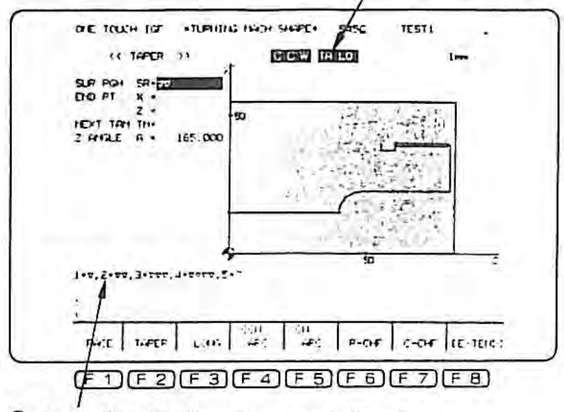
b) How to Designate

Press the function key [F8] (EXTEND) until the function "SHAPE RE-DIS." is assigned to the function key [F6].

Press the function key [F6] (SHAPE RE-DIS.). The shape defined by the entered data will be re-displayed.

The re-display function is used to re-display the finished shape after the shape element data has been changed.

Indication of shape element which has not been defined (magenta)



Screen for designating undefined shape elements

Note 1: *If the function key [F6] (SHAPE RE-DIS.) is pressed when there is still undefined shape element, the shape element designation screen for that shape element will be called and the undefined shape element is displayed with a symbol at the upper part of the screen. The undefined shape is not displayed on the screen.*

Note 2: *If the function key [F6] (SHAPE RE-DIS.) is pressed when all shape elements have been defined, the shape element designation screen of the latest defined shape element will be displayed and the entire shape will be displayed.*

(17) Termination of TURNING MACH SHAPE Definition

When TURNING MACH SHAPE definition beginning from the starting point in such a manner as single-stroke drawing has been completed, press the function key [F2] (ORDER ↓). The TURNING MACH SHAPE definition process is terminated.

When ROUND BAR is designated for BLANK SIZE, the CRT will return to the start-up page of the graphic edit screen.

Note: *The end point of the last shape element is identical to the shape definition starting point. Therefore, beside terminating the TURNING MACH SHAPE definition process by designating the end point coordinates, it is also possible to terminate it by entering "E". Proceed as follows.*

i) Display the shape element designation screen of the last shape element.

ii) Move the cursor to END PT X (or END PT Z).

*iii) Enter "E" and press the **WRITE** key.*

This completes the TURNING MACH SHAPE definition process.

(18) Shape Referencing Function

This is the function to display points of each individual shape elements which have been determined by entering the end point coordinate or calculating points of intersection.

This function can be used for the following three finished shape definition screens.

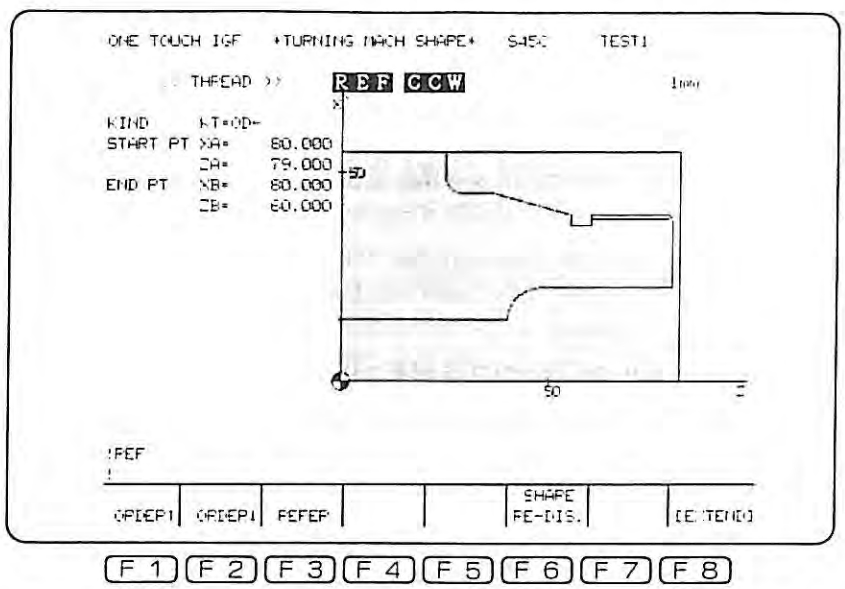
- TURNING MACH SHAPE definition screen
- GENERATING SHAPE definition screen
- BLANK SHAPE definition screen

a) Function

1) Press the function key [F8] (EXTEND) until the function "SHAPE REFER" is assigned to the function key [F3].

   Press the function key [F3] (SHAPE REFER). The shape referencing mode will be established.

2) Using the cursor control keys ← and →, locate the marker (magenta) to the finished shape or the blank material shape. Data such as starting point, end point, center coordinate (arc), type of thread, or direction of groove of the marker-indicated shape will be displayed on the screen. Shape data cannot be entered in the shape referencing mode.



* The values for START PT and END PT are displayed on the basis of ridge diameter.

Shift the marker using the cursor control keys ← and →. The data of the marker-indicated shape will be displayed on the screen.

3) To cancel the shape referencing mode, press any of the following function keys.

   SHAPE REFER, INSERT, DELETE, SHAPE RE-DIS., ORDER , ORDER , QUIT, and each shape element key

(19) Automatic Generation of Chamfering

The one-touch IGF system automatically generates chamfering. Follow the procedure indicated below:

1) Designate the chamfer size on the START PT designation screen in turning shape definition.
2) Press the function key [F7] (C-CHF) after the final shape element has been designated in turning shape definition.

For the following portions, chamfering is not generated:

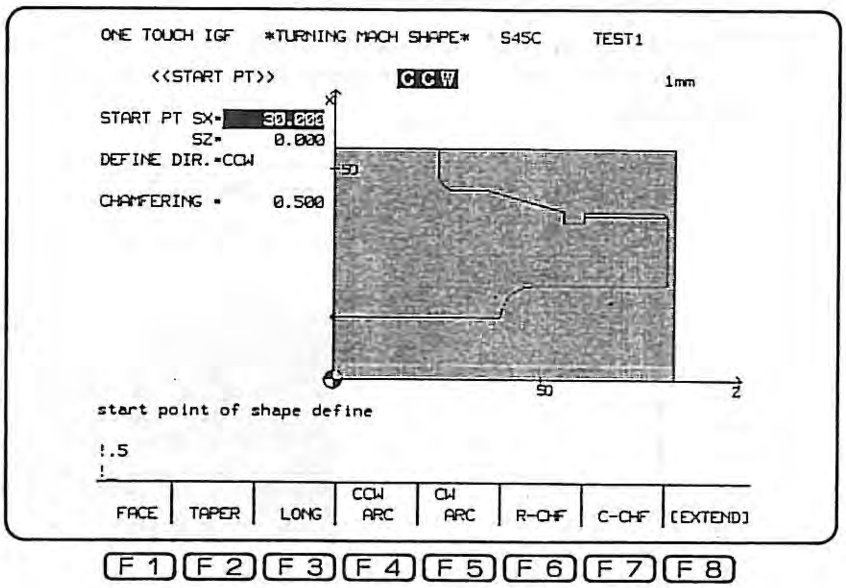
- Starting point
- Before and after jump
- Chamfered or rounded portion of groove
- When the angle between two shape elements is larger than the setting of the integer parameter No. 34 (FORMATION ANGLE OF CHAMFERING).

Example:

An explanation is given taking Section 4 in the Operation Manuel - Basic as an example.

Note operations other than automatic chamfering generation are not explained in detail here.

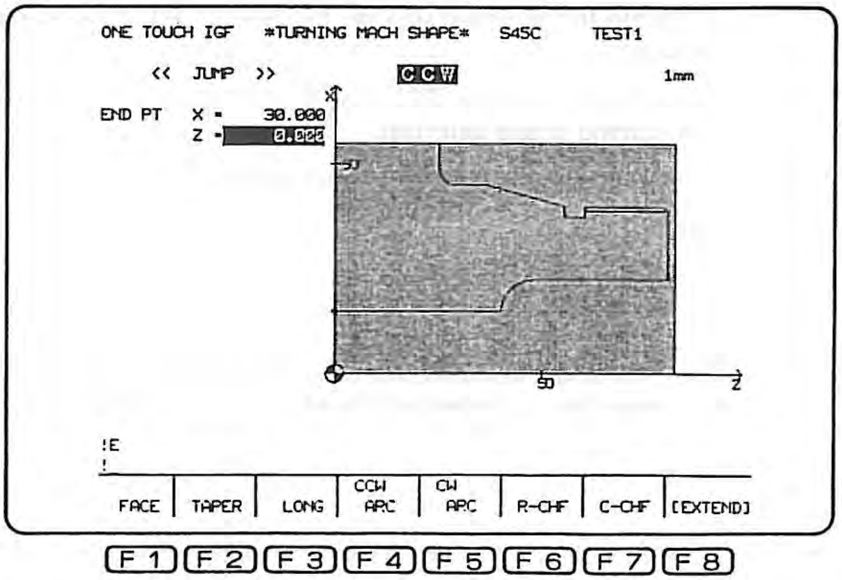
1) Set a required chamfer size to the data column of CHAMFERING C = on the START PT screen in turning shape definition.



* C = 0.5 in this example

2) Designate each shape element from the starting point to the end in the similar manner as "drawing with a single stroke". This means the end point of the final shape element is identical to the starting point.

3) The final shape element, or in other words, the shape element to return to the starting point, is "JUMP" in this example. Designate END PT X, Z of jump operation. This ends designation of all the shape elements.

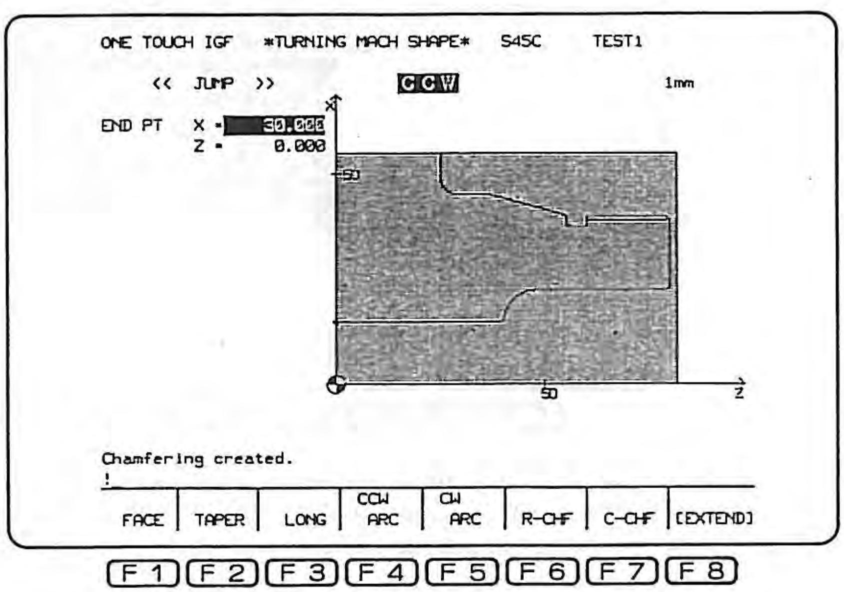


* To designate the END PT X, Z, either directly enter the end point coordinates or key in "E" and press the WRITE key.

4) Press the function key [F8] (EXTEND) until the function "C-CHF" is assigned to the function key [F7].

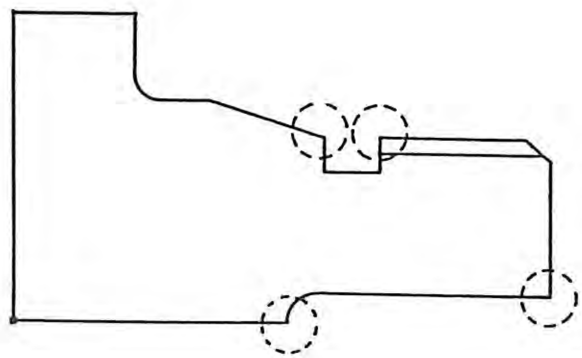
   Then, press the function key [F7] (C-CHF). This will automatically generate chamfering.

5) To check the generated chamfer(s), press the cursor control key ←. Each time the ← key is pressed, the shape element designation screen is displayed in reverse order of shape designation.



6) Press the function key [F8] (EXTEND) two times and press the function key [F2] (ORDER ↓). This completes turning shape definition.

7) Chamfer generated positions



Chamfer generated positions

When "130°" is set for the integer parameter No. 34 and shape elements are designated following the procedure in Section 4 of the Operation Manual - Basic (Publication No. 3365-E), chamfers are generated at the positions marked in the diagram above.

Note 1: *To amend the automatically generated chamfer size, call the necessary shape element designation screen using the cursor control keys ← and →.*

Note 2: *To automatically generate chamfering, press the function key [F7] (C-CHF) with turning shape definition complete (finished workpiece shape has been drawn by a single stroke).*

Note 3: *Chamfering is not generated automatically when the shape element insertion or shape referencing mode is effective.*

Note 4: *Chamfering is not generated automatically in the following cases:*

- The ID of the machined workpiece is smaller than the setting of the dimension parameter No. 28 (BLIND HOLE MIN. DIA. VALUE).
- Very small steps

4. Finished Shape Definition for Multiple Machining (optional)

The explanation that follows is the procedure for finished shape definition for multiple machining.

The finished shape definition process consists of the following shape definition steps.

The following functions are provided to define the multiple machining shape.

(1) DRILL

(2) BORING

(3) TAP

(4) REAMING

(5) KEY WAY

(6) Contour generating (Refer to 5. and 6. in this section.)

(7) Display of guide drawing

(8) Deletion of shape element

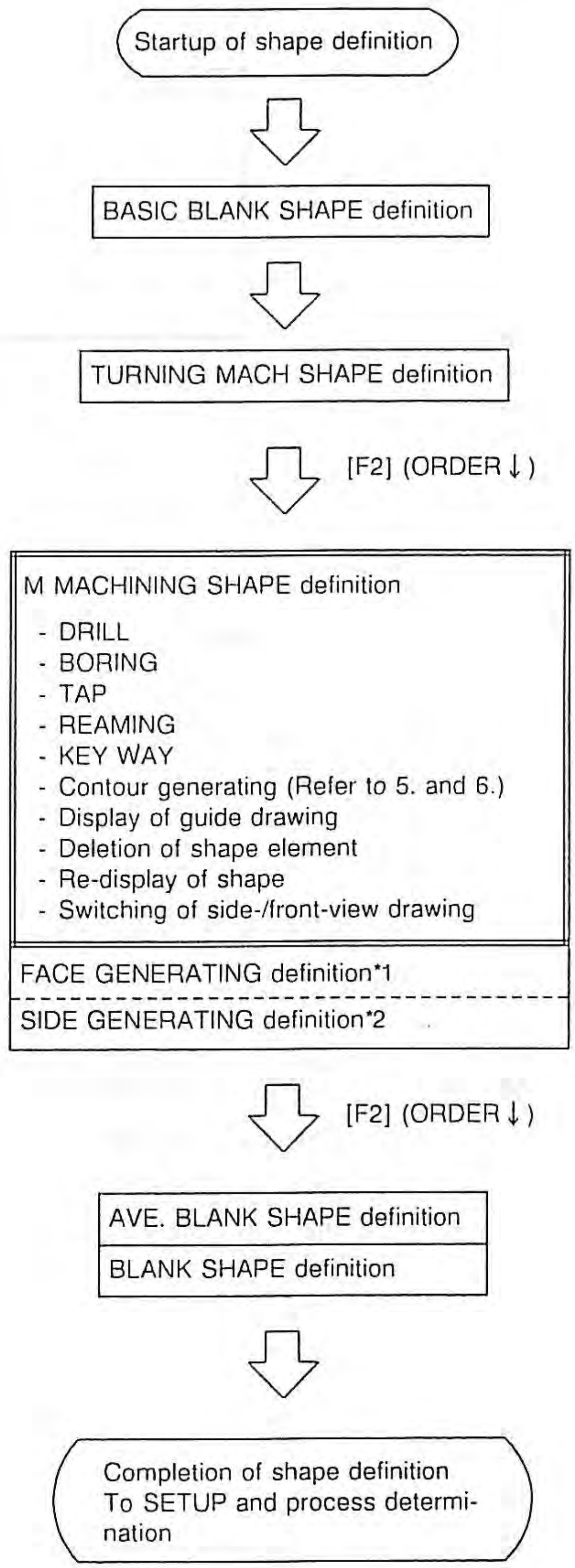
(9) Re-display of shape

(10) Switching of side-/front-view drawing

In the multi-machining shape definition process, enter the shape elements (1) to (6) sequentially according to the finished shape. The functions (1) to (6) are assigned to the function keys.

A total of 256 shape elements can be designated.

The relationship between the sequence of shape definition and the multi-machining shape definition is diagramed below.

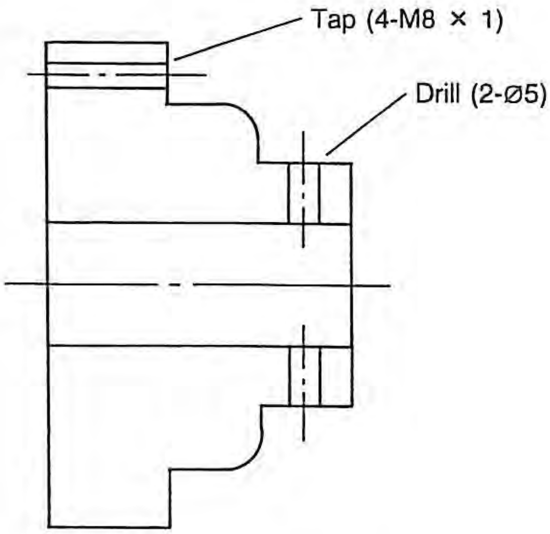


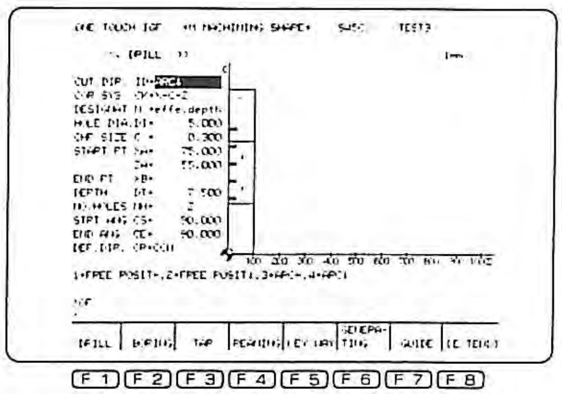
*1 Optional: Multi-machining specification
*2 Optional: Contour generating machining specification

When multi-machining shape definition is completed, press the function key [F2] (ORDER ↓). The CRT will advance to the BLANK SHAPE screen.

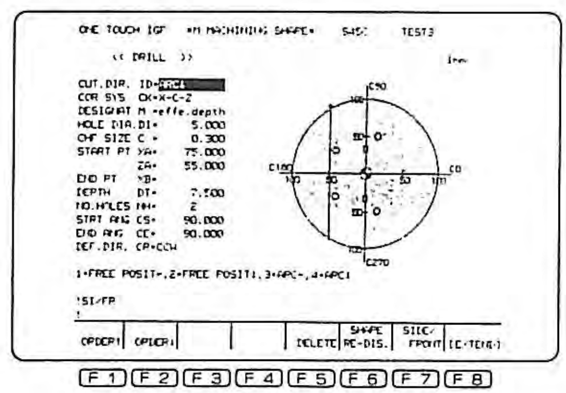
Multi-machining shape definition is carried out in the following sequence.

Example:





Side-view drawing - X-C-Z coordinate system



Front-view drawing - X-C-Z coordinate system

1) After the completion of turning shape definition, press the function key [F2] (ORDER ↓) to advance to the M MACHINING SHAPE screen.

2) Select the shape element and define the multi-machining shape.

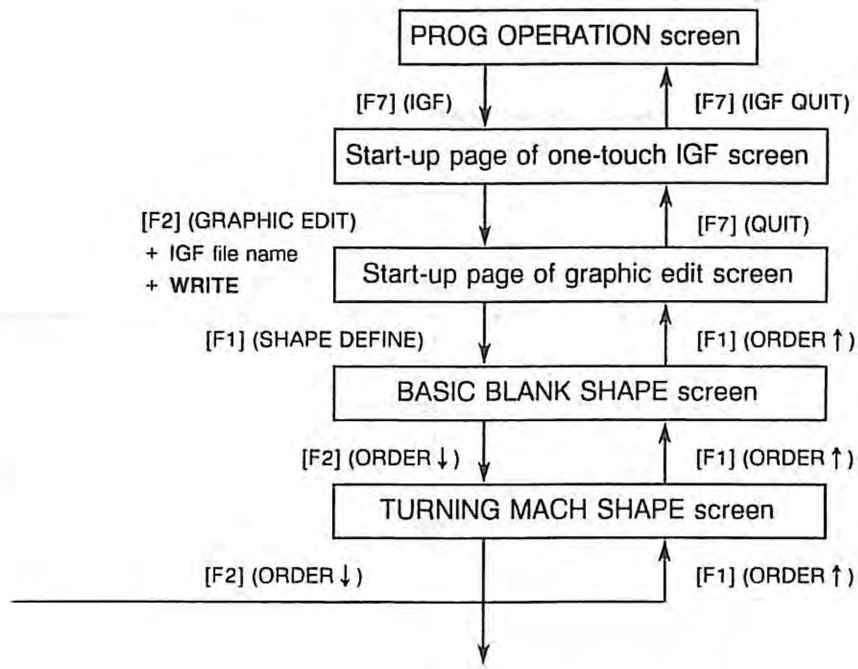
   Necessary shape elements: TAP, DRILL

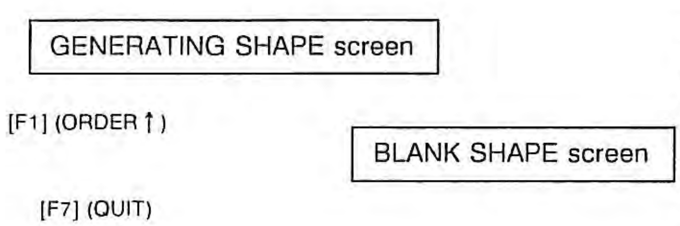
3) After the completion of multi-machining shape definition, press the function key [F2] (ORDER ↓).

   The CRT will advance to the BLANK SHAPE screen.

Screen Transfer:

The M MACHINING SHAPE screen is called from the PROG OPERATION screen as diagramed below.



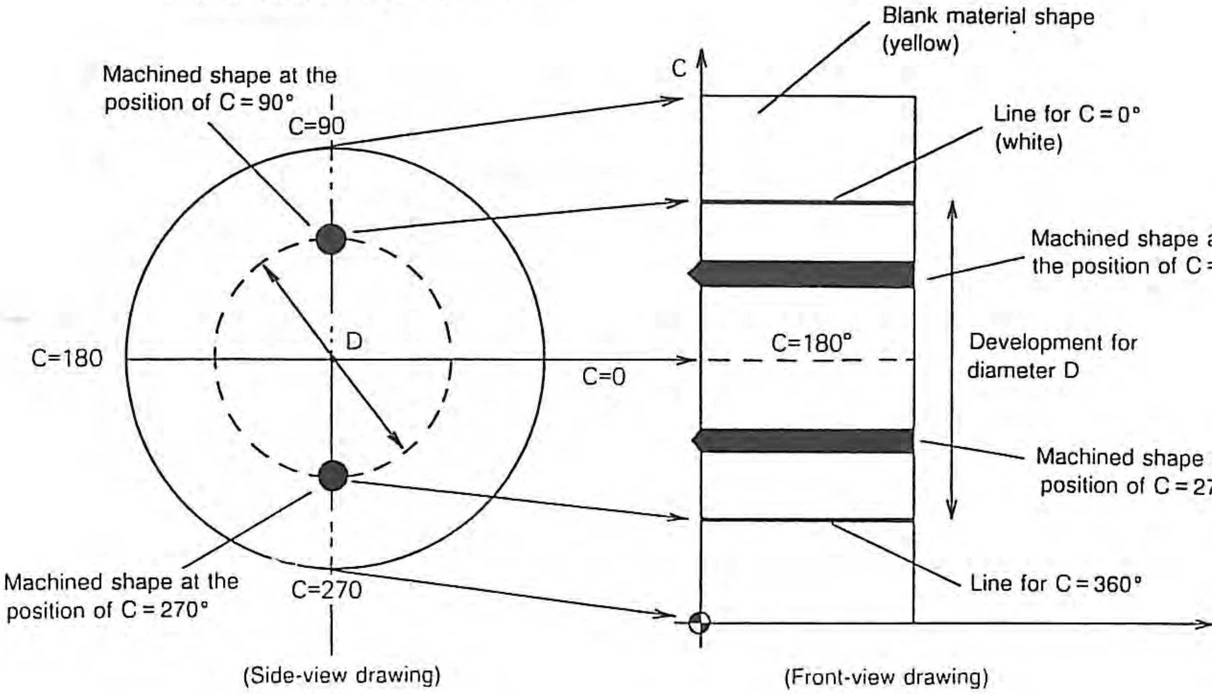


When the M MACHINING SHAPE definition process is completed, press the function key [F2] (ORDER ↓). The CRT will advance to the BLANK SHAPE screen. (When ROUND BAR is set for BLANK SIZE, the start-up page of the graphic edit screen will be displayed.)

Display the side-view drawing on the M MACHINING SHAPE screen:

The side-view drawing on the M MACHINING SHAPE screen is a development on the basis of the X-coordinate of the machining starting point.

The reference angle is C = 180°.



The developments as above are placed one upon another on the screen.

(1) **DRILL**

a) Function

The shape of drilling for M MACHINING SHAPE is designated.

When the shape of drilling is defined, the one-touch IGF will automatically generate the preparatory processes for drilling, and determine also the tools to be used and cutting conditions.

(a) DRILL

- M-drill centering process
- M-drill blind/thru-hole drilling process
- M-drill chamfering process

(b) DRILL

- M-drill blind/thru-hole drilling process (with centering motion)
- M-drill chamfering process

(c) DRILL

- M-drill chamfering process (for including centering)
- M-drill blind/thru-hole drilling process

Selection of preparatory process generation pattern, (a), (b), or (c) can be designated by the one-touch IGF integer parameter No. 55.

One-touch IGF integer parameter No. 55 M MACHINING HOLE PROCESS WAY

Setting		Pattern
0	......	(a)
1	......	(b)
2	......	(c)

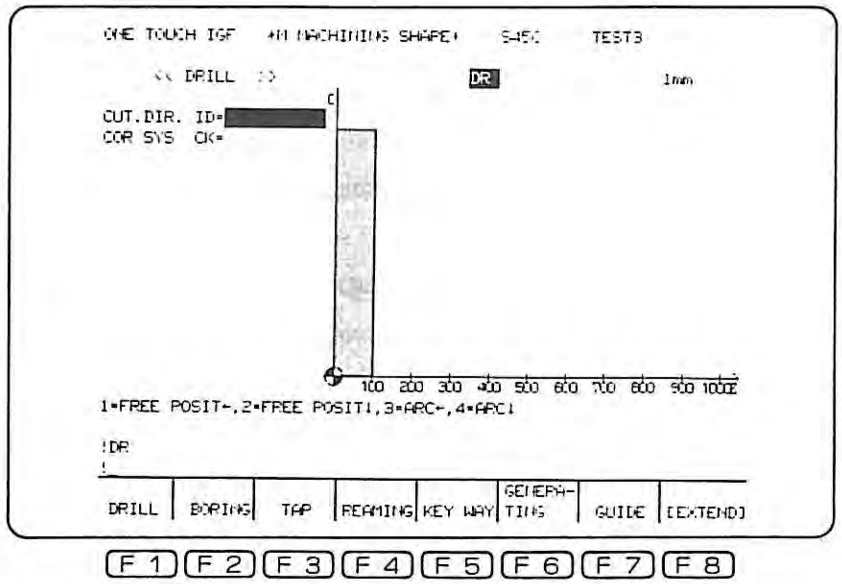
Drilling processes can be edited in the process editing mode.

For details of process editing, refer to Section 11.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "DRILL" is assigned to the function key [F1].

Press the function key [F1] (DRILL). The shape element designation screen for DRILL will be displayed. Make sure that input items CUT.DIR. ID and COR SYS CK are displayed on this screen.



CUT.DIR. ID

The infeed direction of drilling is designated. As the prompt "**1 = FREE POSIT←, 2 = FREE POSIT↓, 3 = ARC←, 4 = ARC↓**" is displayed, enter the corresponding number. For example, enter "1" to select "1 = FREE POSIT←" and "3" for "3 = ARC←".

FREE POSIT .... To drill one (1) hole at the arbitrary position

ARC .... To drill multiple holes on the circumference at regular intervals

← .... Drilling on the front surface of material

↓ .... Drilling on the side surface of material

COR SYS CK

The coordinate system in which the drill position is designated is selected. As the prompt "**1 = X-C-Z, 2 = X-Y-Z**" is displayed, enter the corresponding number. For example, enter "1" to select "1 = X-C-Z".

By designating CUT.DIR. ID and COR SYS CK, one of the following six shape definition combinations will be selected.

FREE POSIT← X-C-Z coordinate system

ARC← X-C-Z coordinate system

FREE POSIT← X-Y-Z coordinate system

ARC← X-Y-Z coordinate system

FREE POSIT↓ X-C-Z coordinate system

ARC↓ X-C-Z coordinate system

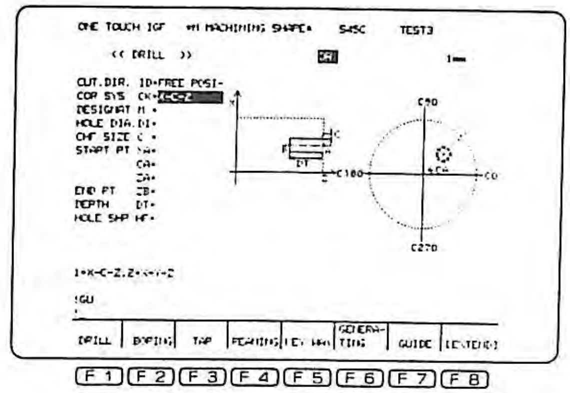
Shape element data to be designated in each of the combinations is explained below.

<u>For FREE POSIT←X-C-Z Coordinate System and ARC← X-C-Z Coordinate System</u>

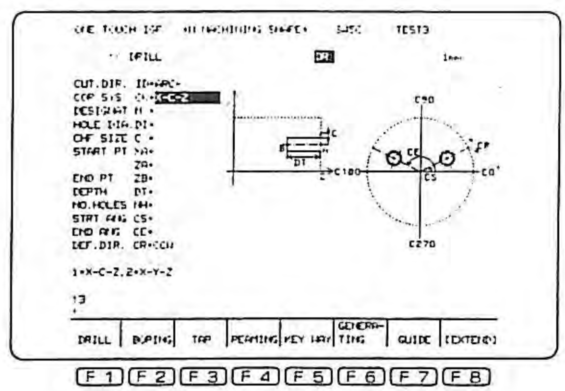
a) Data to be Designated

CUT.DIR.	ID
COR SYS	CK
DESIGNAT	M
HOLE DIA.	DI
CHF SIZE	C
START PT	XA
	CA (*1)
	ZA
END PT	ZB
DEPTH	DT
NO. HOLES	NH (*2)
STRT ANG	CS (*2)
END ANG	CE (*2)
DEF.DIR.	CR (*2)
HOLE SHP	HF

*1 For FREE POSIT only
*2 For ARC only



(Guide drawing is displayed.)

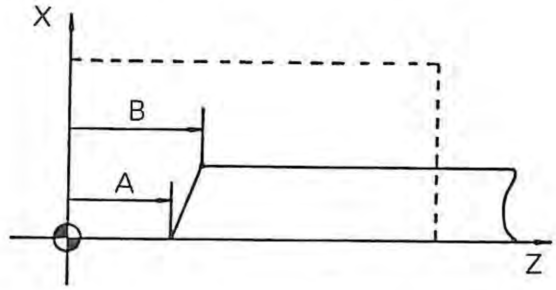


(Guide drawing is displayed.)

b) How to Designate

DESIGNAT M

How the drilling depth is designated, by tool tip or effective length, is selected.



On the drawing above, if point A is recognized as the end point, this is tool tip designation. If point B is recognized as the end point, this is effective length designation.

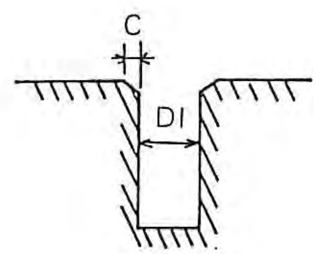
HOLE DIA.DI

The drill hole diameter is designated.

CHF SIZE C

The chamfer size is automatically set on the basis of the standard hole data when DI has been designated.

Correct the setting if required.



START PT XA
CA (for FREE POSIT only)
ZA

The X-, C-, and Z-coordinates of the starting point of drilling (point A on the guide drawing) are designated.

For START PT XA, a diametrical value must be input.

END PT ZB

The Z-coordinate of the end point of drilling shape (point B on the guide drawing) is designated.

No data input is required when DEPTH DT has been designated.

DEPTH DT

The drilling depth is designated. No data input is required when END PT ZB has been designated.

NO. HOLES NH (for ARC only)

The number of holes designated here is drilled at regular intervals within the angle designated by STRT ANG CS and END ANG CE. If no data is set, the system recognizes the number of hole to be "1".

STRT ANG CS (for ARC only)

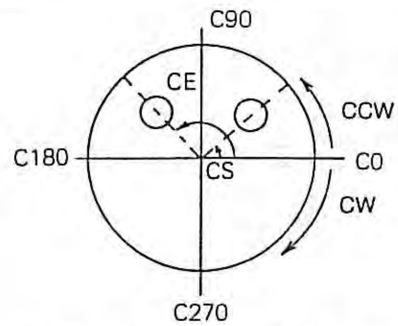
The angle for the first hole is designated.

END ANG CE (for ARC only)

The angle for the last hole is designated.

DEF.DIR. CR (for ARC only)

Direction of cut from CS to CE is designated. Enter "1" for CCW direction and "0" for CW direction.



Note: *Relationship between NH, CS, and CE*

i) NO. HOLES NH = 0 or NO. HOLES NH = 1
(If no data is set, the system recognizes the number of hole to be "1".)

A hole is drilled at the position designated by STRT ANG CS.

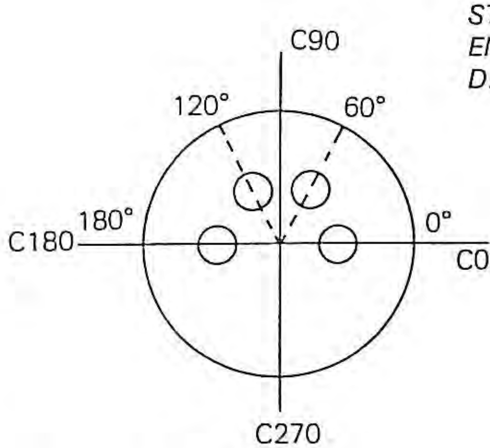
No data input is required for END ANG CE.

ii) NO. HOLES NH≧2

The designated number of holes is drilled at regular intervals within the angle designated by STRT ANG CS and END ANG CE.

Example: *Hole positions when the data is set as follows:*

NO. HOLES NH = 4
STRT ANG CS = 0
END ANG CE = 180°
DEF.DIR. CR = CCW



The number of holes designated by NO. HOLES NH is drilled on the circumference at regular intervals when AND ANG CE is not designated or the same value is set for both STRT ANG CS and END ANG CE.

HOLE SHP HF

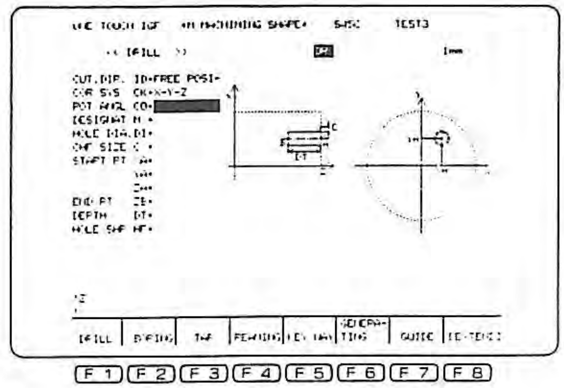
The type of hole, blind hole or through hole, is designated.

For FREE POSIT← X-Y-Z Coordinate System and ARC← X-Y-Z Coordinate System

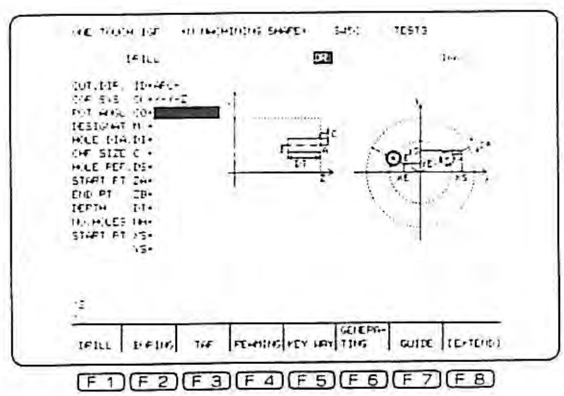
a) Data to be Designated

CUT. DIR.	ID
COR SYS	CK
ROT ANGL	CO
DESIGNAT	M
HOLE DIA	DI
CHF SIZE	C
HOLE REF	DS (*1)
START PT	XA (*2)
	YA (*2)
	ZA
END PT	ZB
DEPTH	DT
NO. HOLES	NH (*1)
STRT ANG	XS (*1)
	YS (*1)
END ANG	XE (*1)
	YE (*1)
DEF. DIR.	CR (*1)
HOLE SHP	HF

*1 For ARC only
*2 For FREE POSIT only



(Guide drawing is displayed.)



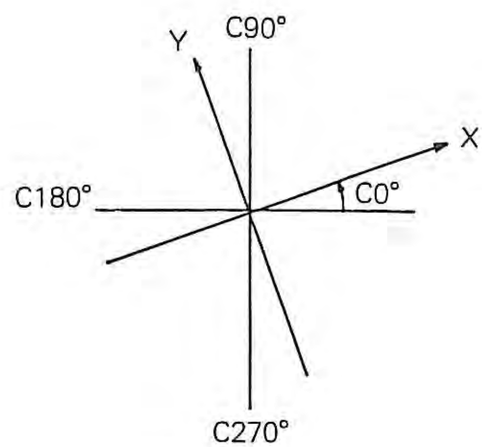
b) How to Designate

ROT ANGL CO

The reference angle in the X-Y-Z coordinate system is designated.

The reference angle refers to the rotation angle by which the X-Y coordinate is rotated from the X-C coordinate system.

The reference angle can be designated within the range from −360° to +360°.



DESIGNAT M

HOLE DIA. DI

CHF SIZE C

Refer to the explanation for FREE POSIT← X-C-Z coordinate system and ARC← X-C-Z coordinate system on page 120.

HOLE REF. DS (for ARC only)

The reference diameter of the drill hole is designated. Enter the value in the diametrical value.

START PT XA (for FREE POSIT only)
YA (for FREE POSIT only)
ZA

The X-, Y-, and Z-coordinates of the starting point of drilling (point A on the guide drawing) are designated.

For START PT XA, enter the value in radius.

END PT ZB

The Z-coordinate of the end point of drilled shape (point B on the guide drawing) is designated.

No data input is required when DEPTH DT has been designated.

DEPTH DT

The drilling depth is designated. No data input is required when END PT ZB has been designated.

NO. HOLES NH (for ARC only)

The number of holes designated are drilled at regular intervals within the angle designated by STRT ANG CS and END ANG CE. If no data is set, the system recognizes the number of holes to be "1".

STRT PT XS (for ARC only)
YS (for ARC only)

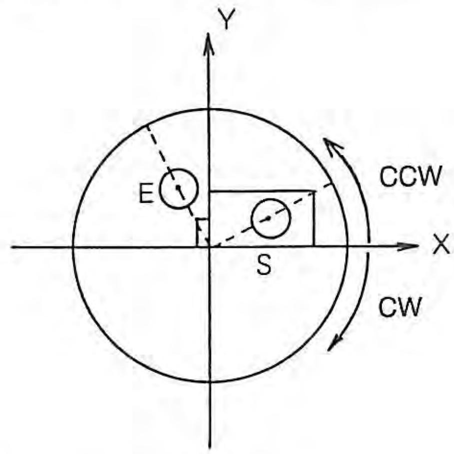
The X- and Y-coordinates of the first hole position is designated. For STRT PT XS, enter the value in radius.

END PT XE (for ARC only)
YE

The X- and Y-coordinates of the last hole position is designated. For END PT XE, enter the value in radius.

DEF. DIR. CR (for ARC only)

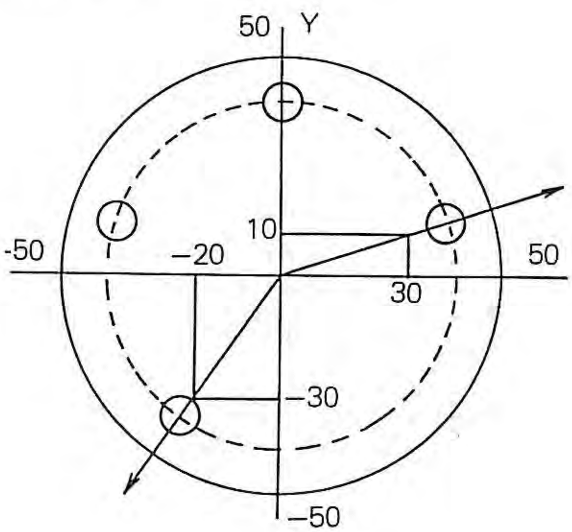
Direction of cut from (XS, YS) to (XE, YE) is designated. Enter "1" for CCW direction and "0" for CW direction.



Example:

Hole positions when the data is set as follows:

STRT PT XS =30
STRT PT YS =10
END PT XE =−20
END PT YE =−30
DEF. DIR. CR =CCW
HOLE REF. DS =80 ??(diameterical value)
NO. HOLES NH =4



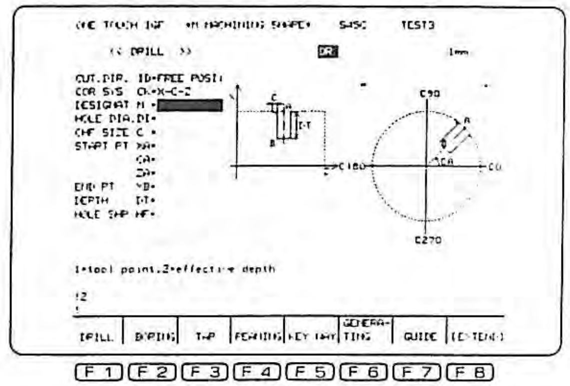
HOLE SHP HF

The type of hole, blind hole or through hole, is designated.

For FREE POSIT/ X-C-Z Coordinate System and ARC/ X-C-Z Coordinate System

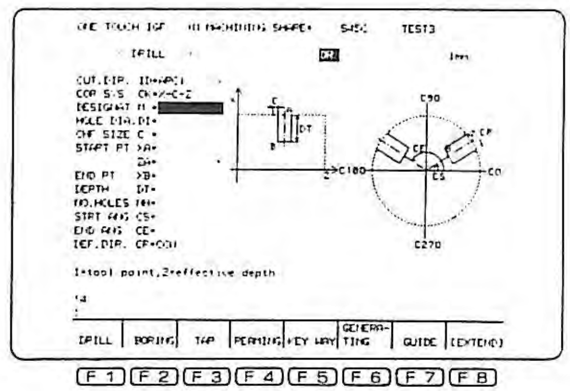
a) Data to be Designated

CUT.DIR.	ID
COR SYS	CK
DESIGNAT	M
HOLE DIA.	DI
CHF SIZE	C
START PT	XA
	CA (*1)
	ZA
END PT	XB
DEPTH	DT
NO.HOLES	NH (*2)
STRT ANG	CS (*2)
END ANG	CE (*2)
DEF.DIR.	CR (*2)
HOLE SHP	HF



(Guide drawing is displayed.)

*1 For FREE POSIT only
*2 For ARC only



(Guide drawing is displayed.)

b) How to Designate

DESIGNAT M

HOLE DIA. DI

CHF SIZE C

Refer to the explanation for FREE POSIT←X-C-Z coordinate system and ARC← X-C-Z coordinate system on page 133.

START PT XA
CA (for FREE POSIT only)
ZA

The X-, C-, and Z-coordinates of the starting point of drilling (point A on the guide drawing) are designated.

For START PT XA, a diametrical value must be input.

END PT XB

The X-coordinate of the end point of drilling shape (point B on the guide drawing) is designated. Enter the diametrical value. No data input is required when DEPTH DT has been designated.

DEPTH DT

The drilling depth is designated. Enter the value in radius. No data input is required when END PT XB has been designated.

NO. HOLES NH (for ARC only)

STRT ANG CS (for ARC only)

ND ANG CE (for ARC only)

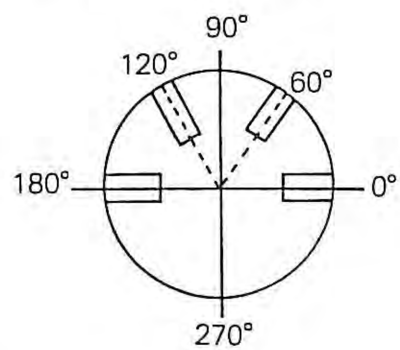
DEF. DIR. CR (for ARC only)

Refer to the explanation for FREE POSIT← X-C-Z coordinate system and ARC← X-C-Z coordinate system on page 133.

(Example)

Hole positions when the data is set as follows:

NO. HOLES NH = 4
STRT ANG CS = 0°
END ANG CE = 180°
DEF. DIR. CR = CCW



The number of holes designated by NO. HOLES NH is drilled on the circumference at regular intervals when AND ANG CE is not designated or the same value is set for both STRT ANG CS and END ANG CE.

HOLE SHP HF

The type of hole, blind hole or through hole, is designated.

c) Guide Drawing

If the function key [F7] (GUIDE) is pressed when designating the drill shape elements, a guide drawing will be displayed. Since the guide drawing displayed varies depending on the direction of cut and coordinate system selected, first designate CUT. DIR. ID and COR SYS CK and then call the guide drawing.

The display of guide drawing is turned on/off each time the function key [F7] (GUIDE) is pressed.

(Refer to (7) "Display of Guide Drawing" on page 150.)

(2) **BORING**

a) Function

The shape of boring for M MACHINING SHAPE is designated.

When the shape of boring is defined, the one-touch IGF will automatically generate the preparatory processes for boring, and determine also the tools to be used and cutting conditions.

(a) BORING

- M-drill centering process
- M-drill blind hole drilling process
- M-drill chamfering process
- M-boring (rough) process
- M-boring (finish) process

(b) BORING

- M-drill blind hole drilling process (with centering motion)
- M-drill chamfering process
- M-boring (rough) process
- M-boring (finish) process

(c) BORING

- M-drill chamfering process (for including centering)
- M-drill blind hole drilling process
- M-boring (rough) process
- M-boring (finish) process

Selection of preparatory process generation pattern, (a), (b), or (c) can be designated by the one-touch IGF integer parameter No. 55.

One-touch IGF integer parameter No. 55 M MACHINING HOLE PROCESS WAY

Setting		Pattern
0	......	(a)
1	......	(b)
2	......	(c)

Boring processes can be edited in the process editing mode.

For details of process editing, refer to Section 11.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "BORING" is assigned to the function key [F2].

Press the function key [F2] (BORING). The shape element designation screen for BORING will be displayed.

c) Data to be Designated

The data to be designated for boring is the same as that for drill, with the exception of SUR RGH SR (surface roughness). Refer to the explanation in (1) "DRILL" on page 118.

SUR RGH SR

Surface roughness of boring hole is designated.

As the prompt "1 = ∇, 2 = ∇∇, 3 = ∇∇∇, 4 = ∇∇∇∇, 5 = ~" is displayed, enter the corresponding number. For example, enter "2" to select "2 = ∇∇". When "1 = ∇" is designated for SUR RGH SR, the boring (finish) process will not be generated in process determination.

Note: *The M-boring (rough) process and the M-boring (finish) process are not distinguished each other on the PROCESS EDITING screen.*

Two processes with the same name are displayed on the screen as follows.

:
:
12 M boring
13 M boring
:
:

(3) **TAP**

a) Function

The shape of tap for M MACHINING SHAPE is designated.

When the shape of tap is defined, the one-touch IGF will automatically generate the preparatory processes for tapping, and determine also the tools to be used and cutting conditions.

(a) TAP

- M-drill centering process
- M-drill blind hole drilling process
- M-drill chamfering process
- M-tapping process

(b) TAP

- M-drill blind hole drilling process
- M-drill chamfering process
- M-tapping process

(c) TAP

- M-drill chamfering process (for including centering)
- M-drill blind hole drilling process
- M-tapping process

Selection of preparatory process generation pattern, (a), (b), or (c) can be designated by the one-touch IGF integer parameter No. 55.

One-touch IGF integer parameter No. 55 M MACHINING HOLE PROCESS WAY

Setting		Pattern
0	......	(a)
1	......	(b)
2	......	(c)

Tapping processes can be edited in the process editing mode.

For details of process editing, refer to Section 11.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "TAP" is assigned to the function key [F3].

Press the function key [F3] (TAP). The shape element designation screen for TAP will be displayed.

c) Data to be Designated

The data to be designated for tap is the same as that for drill, with the exception of the data indicated below. Refer to the explanation in (1) "DRILL" on page 118.

TAP KIND KP

The type of tap is designated. The prompt "**1 = M-THREAD, 2 = UNIFIED THREAD, 3 = PF, 4 = PT, 5 = PS, 6 = NPT, 7 = NPS, 8 = NPTF, 9 = NPSF, 10 = others**" is displayed. Enter the number corresponding to the thread to be cut. For example, enter "1" to select "1 = M-THREAD (metric thread)".

NOMNL SZ DI

The nominal diameter of tap is designated. Select the number from the prompt displayed and enter it. When "10 = others" has been designated for TAP KING KP, enter the diameter of the tapping tool.

PITCH F

The pitch is designated. When the standard thread data has been designated for TAP KIND KP and NOMNL SZ DI, the value for PITCH P is automatically set on the basis of the standard hole data.

PR-DRL PD

The pre-drilled diameter of tap is designated.

When the standard thread data has been designated for TAP KIND KP and NOMNL SZ DI, the value for PR-DRL PD is automatically set on the basis of the standard hole data.

CHF SIZE C

The chamfer size is designated.

When the standard thread data has been designated for TAP KIND KP and NOMNL SZ DI, the value for CHF SIZE C is automatically set on the basis of the standard hole data.

(4) REAMING

a) Function

The shape of reaming for M MACHINING SHAPE is designated.

When the shape of reaming is defined, the one-touch IGF will automatically generate the preparatory processes for reaming, and determine also the tools to be used and cutting conditions.

(a) REAMING

- M-drill centering process
- M-drill blind hole drilling process
- M-drill chamfering process
- M-drill endmilling process
- M-reaming process

(b) REAMING

- M-drill blind hole drilling process
- M-drill chamfering process
- M-endmilling process
- M-reaming process

(c) REAMING

- M-drill chamfering process (for including centering)
- M-drill blind hole drilling process
- M-endmilling process
- M-reaming process

Selection of preparatory process generation pattern, (a), (b), or (c) can be designated by the one-touch IGF integer parameter No. 55.

One-touch IGF integer parameter No. 55 M MACHINING HOLE PROCESS PATTERN

Setting		Pattern
0	......	(a)
1	......	(b)
2	......	(c)

Reaming processes can be edited in the process editing mode.

For details of process editing, refer to Section 11.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "REAMING" is assigned to the function key [F4].

Press the function key [F4] (REAMING). The shape element designation screen for REAMING will be displayed.

c) Data to be Designated

The data to be designated for reaming is the same as that for drill. Refer to the explanation in (1) "DRILL" on page 131.

(5) **KEY WAY**

a) Function

The shape of keyway for M MACHINING SHAPE is designated. Both the keyway on the end surface of material and the keyway on the side surface of material can be designated.

When the shape of keyway is defined, the one-touch IGF will automatically generate the keyway rough cutting process and keyway finish cutting process, and determine also cutting conditions.

(a) KEY WAY

- M-keyway end surface (rough)
- M-keyway end surface (finish)

(b) KEY WAY

- M-keyway side surface (rough)
- M-keyway side surface (finish)

Whether both the roughing and finishing processes are created or not can be designated by the one-touch IGF integer parameter No. 56.

One-touch IGF integer parameter No. 56 KEY WAY CUTTING (0 = ROUGH, 1 = ROUGH/ FINISH)

Setting	Machining
0	Only roughing
1	Both roughing and finishing

Keyway cutting processes can be edited in the process editing mode.

For details of process editing, refer to Section 11.

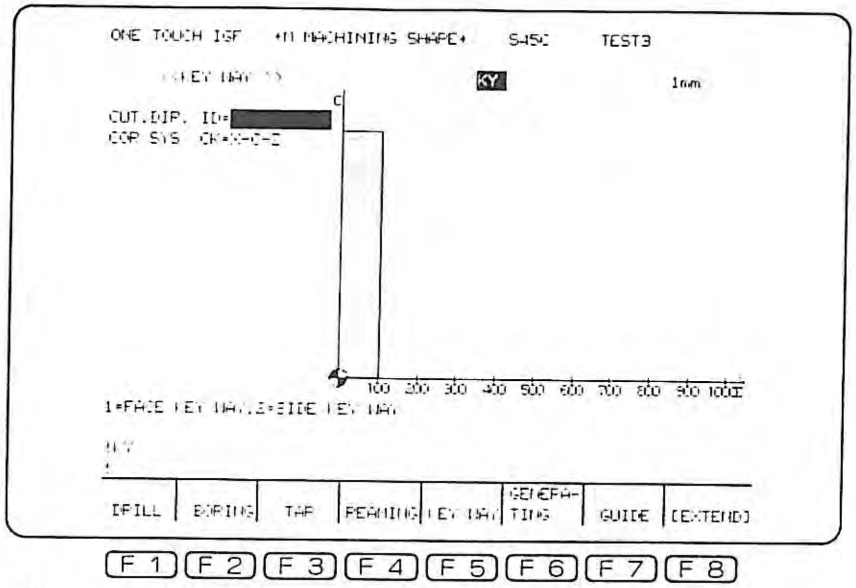
Note: *The keyway cutting processes are displayed as "M-keyway front" or "M-keyway side" on the PROCESS EDITING screen, and the M-keyway (rough) process and the M-keyway (finish) process are not distinguished each other.*

Two processes with the same name are displayed on the screen.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "KEY WAY" is assigned to the function key [F5].

Press the function key [F5] (KEY WAY). The shape element designation screen for KEY WAY will be displayed.



CUT. DIR. ID

Infeed direction of keyway cutting is designated. As the prompt "**1 = FACE KEY WAY, 2 = SIDE KEY WAY**" is displayed, select the number corresponding to the infeed direction. For example, enter "1" for selecting "1 = FACE KEY WAY".

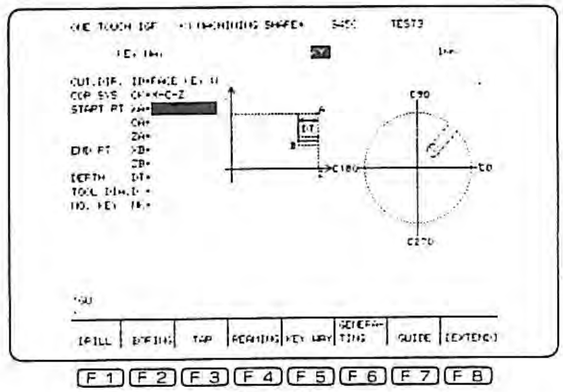
COR SYS CK

As the coordinate system for keyway cutting, only the X-C-Z coordinate system is available.

Data to be Designated for FACE KEY WAY

a) Data to be Designated

CUT. DIR.	ID
COR SYS	CK
START PT	XA
	CA
	ZA
END PT	XB
	ZB
DEPTH	DT
TOOL DIA.	D
NO. KEY	NK



(Guide drawing is displayed.)

b) How to Designate

START PT XA, CA, ZA

The X-, C-, and Z-coordinates of the starting point of keyway cutting (point A on the guide drawing) are designated.

For START PT XA, a diametrical value must be input.

END PT XB, ZB

The X- and Z-coordinates of the end point of keyway cutting (point B on the guide drawing) are designated.

For END PT XB, a diametrical value must be input.

No data input is required when DEPTH DT has been designated.

DEPTH DT

The depth of keyway is designated. When the data has been set for BOTTOM XB (ZB), no data input is required.

TOOL DIA. D

The diameter of the tool used for keyway cutting is designated.

NO. KEY NK

The number of keyways to be cut is designated.

Keyways are cut on the circumference at regular intervals starting from the starting point C-coordinate.

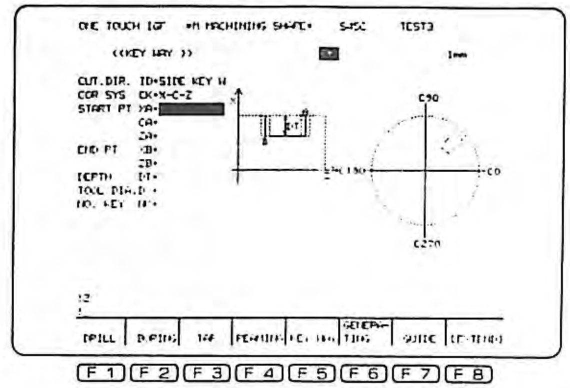
The coordinate values of the tool center position must be entered for START PT and END PT.

Data to be Designated for SIDE KEY WAY

a) Data to be Designated

CUT. DIR.	ID
COR SYS	CK
START PT	XA
	CA
END PT	XB
	ZB
DEPTH	DT
TOOL DIA.	D
NO. KEY	NK

The data to be designated for SIDE KEY WAY is the same as that for FACE KEY WAY. Refer to the explanation on page 148.



(6) **Contour Generating**

a) Function

The shape of contour generating machining for M MACHINING SHAPE is designated.

b) How to Designate

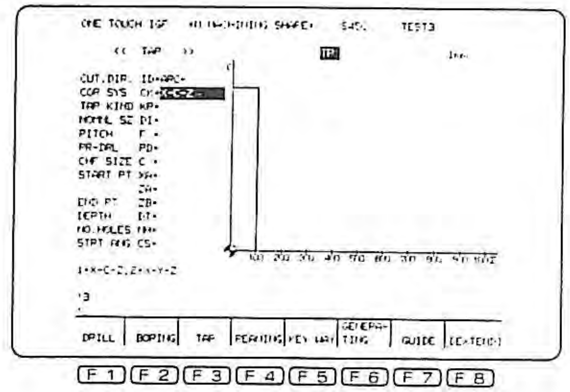
Press the function key [F8] (EXTEND) until the function "GENERATING" is assigned to the function key [F6].

Press the function key [F6] (GENERATING). The shape element designation screen for GENERATING SHAPE will be displayed. Refer to 5. "SIDE GENERATING SHAPE Definition" and 6. "FACE GENERATING SHAPE Definition" for further explanation.

(7) **Display of Guide Drawing**

a) Function

The guide drawing for designating shape elements for multi-machining is displayed.



b) How to Designate

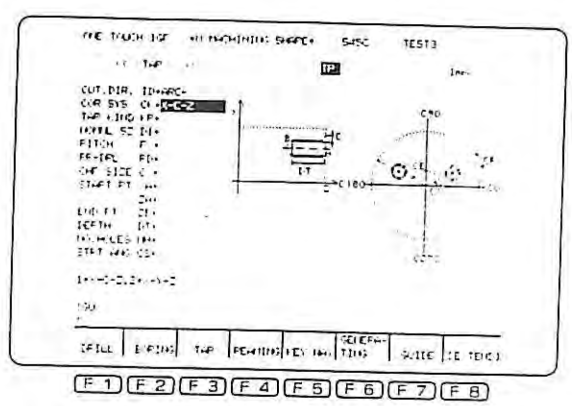
Press the function key [F8] (EXTEND) until the function "GUIDE" is assigned to the function key [F7].

Press the function key [F7] (GUIDE). The guide drawing will be displayed on the screen.

The display of guide drawing is turned on/off each time the function key [F7] (GUIDE) is pressed.

Note 1: *Guide drawing is displayed when designating the shape elements for drilling, boring, tapping, and key-way cutting.*

Note 2: *The guide drawing displayed varies depending on the infeed direction and coordinate system selected. Therefore, first designate the infeed direction and coordinate system and then call the guide drawing. Another guide drawing is displayed when the infeed direction has been changed during shape element designation.*



(Guide drawing is displayed.)

(8) **Deletion of Shape Element**

a) Function

The designated shape element is deleted.

b) How to Designate

The shape element is deleted in the same manner as explained in (15) "Deletion of Shape Element" of TURNING MACH SHAPE definition on page 120.

(9) **Re-display of Shape**

a) Function

The shape defined by entered data is re-displayed.

b) How to Designate

The shape is re-displayed in the same manner as explained in (16) "Re-display of Shape" of TURNING MACH SHAPE definition on page 121.

(10) **Switching of Side-/Front-view Drawing**

a) Function

The drawing on the shape element designation screen for multi-machining is switched between side view and front view.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "SIDE/FRONT" is assigned to the function key [F7].

Each time the function key [F7] (SIDE/FRONT) is pressed, the view angle of the drawing is switched between side and front.

The combination of the coordinate system and the view angle is as indicated below:

When X-C-Z is set for COR SYS CK:

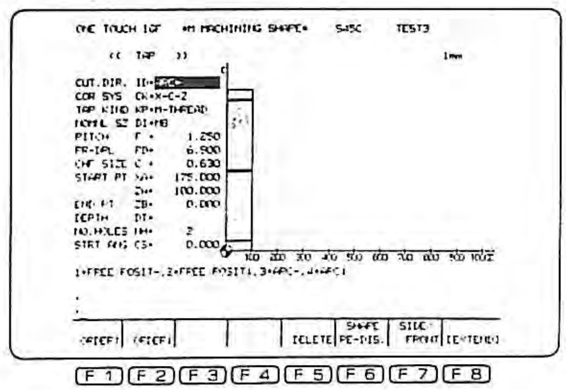
Side-view drawing
→ X-C-Z coordinate system

Front-view drawing
→ X-C-Z coordinate system

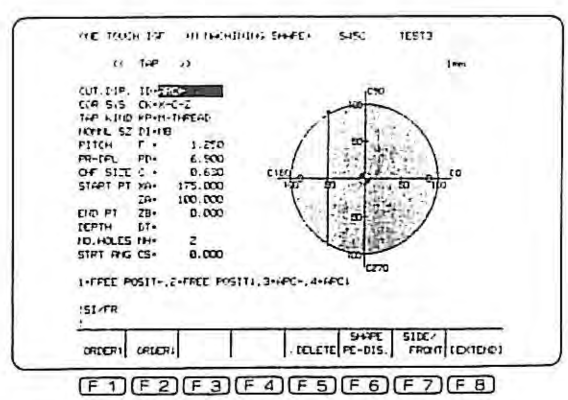
When X-Y-Z is set for COR SYS CK:

Side-view drawing
→ X-C-Z coordinate system

Front-view drawing
→ X-Y-Z coordinate system



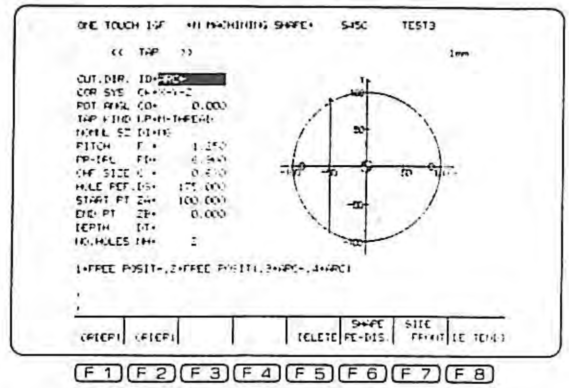
Side-view drawing - X-C-Z coordinate system



Front-view drawing - X-C-Z coordinate system

Note: *On the side-view drawing, hole shape, keyway shape, and side contour generating shape are displayed.*

On the front-view drawing, hole shape, keyway shape, and front contour generating shape are displayed.



Front-view drawing - X-Y-Z coordinate system

(11) Termination of M MACHINING SHAPE Definition

When M MACHINING SHAPE definition has been completed, press the function key [F2] (ORDER ↓). M MACHINING SHAPE definition process is terminated. The CRT will return to the start-up page of the graphic edit screen or advance to the BLANK SHAPE definition screen.

When AVERAGE STICK or FREE SHAPE is designated for BLANK SIZE, the CRT will advance to BLANK SHAPE screen.

5. SIDE GENERATING SHAPE Definition

The explanation that follows is the procedure for finished shape definition for side contour generating machining.

The following functions are provided to define the turning shape.

(1) START PT (START PT, DEFINE DIR., CHAMFERING)

(2) FACE

(3) TAPER

(4) LONG

(5) CCW ARC (optional: contour generating machining specification)

(6) CW ARC (optional: contour generating machining specification)

(7) R-CHF (optional: contour generating machining specification)

(8) CHAMFER

(9) Insertion of shape element

(10) Deletion of shape element

(11) Re-display of shape

(12) Shape referencing function

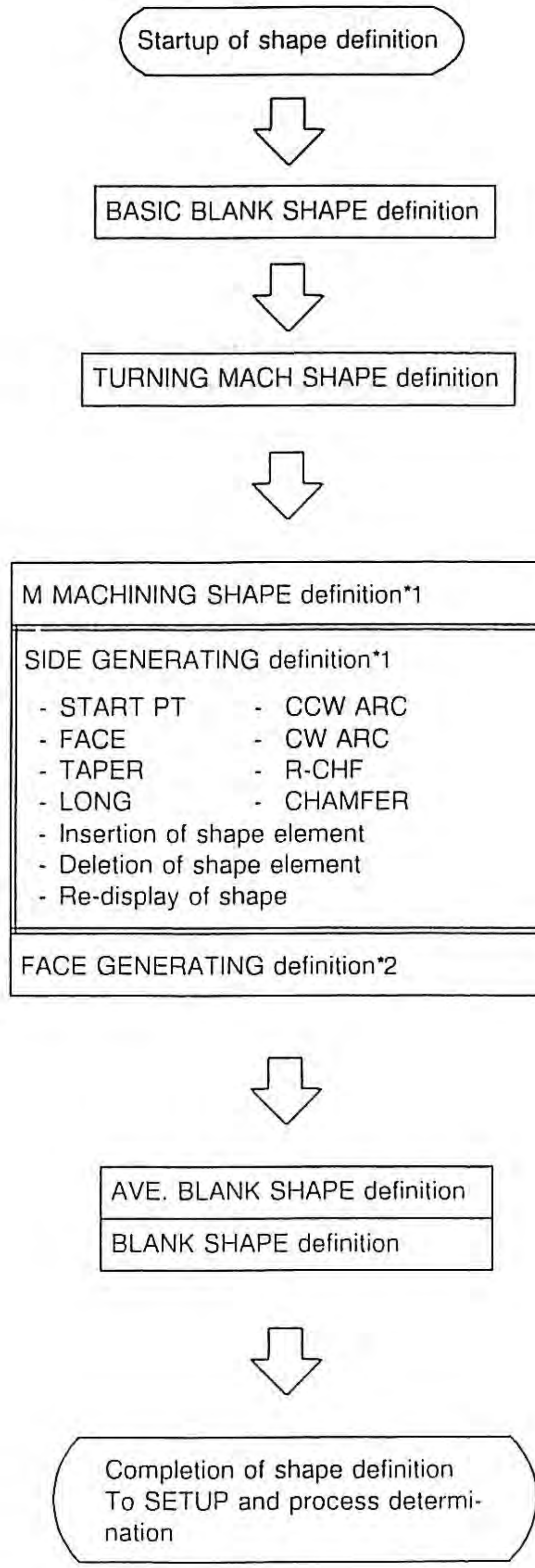
In the contour generating machining shape definition process, first designate the shape definition starting point and then enter the shape elements (2) to (8) sequentially according to the finished shape. The functions (2) to (8) are assigned to the function keys.

Note 1: *To define the shape elements for ARC or R-CHF, the optional contour generating machining specification must be selected.*

Note 2: *The shape of ARC or R-CHF cannot be checked in the process test mode.*

This is because the system recognizes the circular interpolation commanding G code (G132, G133) for side contour generating machining as the G01 command and carries out process test.

The relationship between the sequence of shape definition and the side contour generating shape definition is diagramed below.

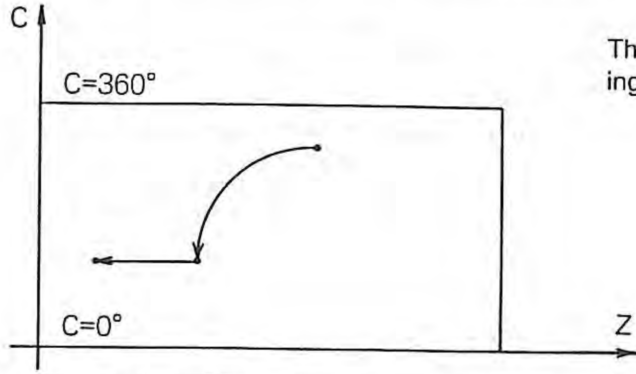


*1 Optional: Multi-machining specification
*2 Optional: Contour generating machining specification

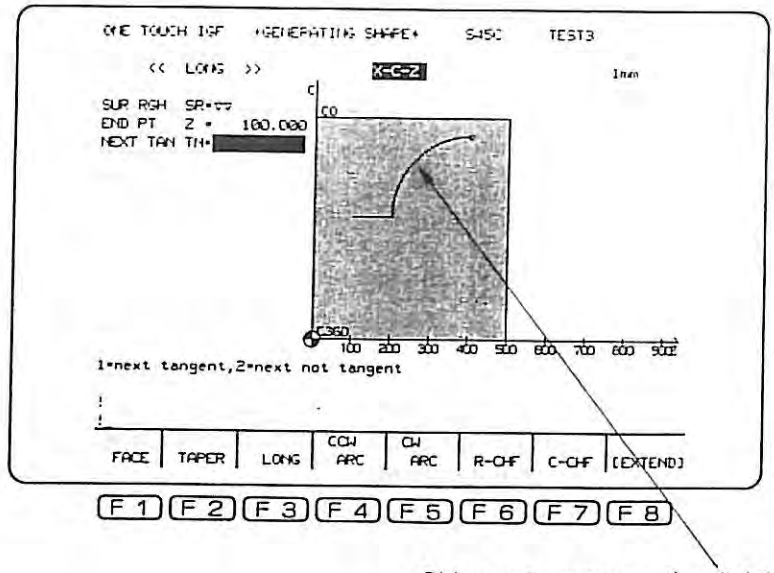
The side contour generating shape definition screen is displayed by pressing the function key [F6] (GENERATING) is pressed on the M MACHINING SHAPE screen. When the function key [F7] (QUIT) is pressed after the completion of side contour generating shape definition, the M MACHINING SHAPE screen will be displayed.

Side contour generating shape definition is carried out in the following sequence.

Example:



The shape is defined on the rotating side surface of workpiece.



Side contour generating finished shape (blue)

1) After the completion of multi-machining shape definition, press the function key [F6] (GENERATING) to advance to the GENERATING SHAPE screen.

2) Enter "2 = SIDE GENERATING" for CUT. DIR. ID and designate COR SYS CK.

3) Enter the data for START PT, CUT. END EX, and TOOL DIA. D.

4) Select the shape element and define the contour generating finished shape.

   Necessary shape elements: CCW ARC, LONG

5) After the completion of side contour generating finished shape, press the function key [F7] (QUIT).

   The CRT will return to the M MACHINING SHAPE screen.

To define multiple contour generating finished shapes, repeat the steps 1) to 5).

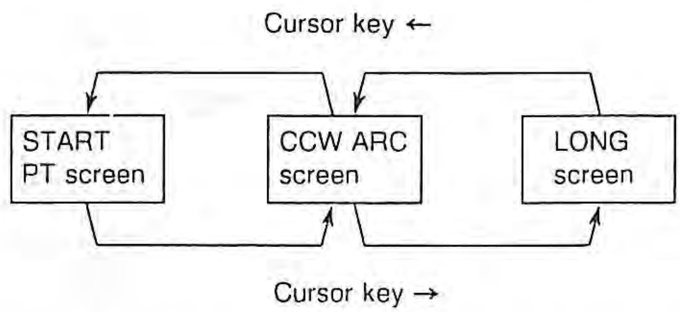
The one-touch IGF system will check the entered data when the function key [F7] (QUIT) is pressed.

Note: *Using the cursor control keys ← and →, it is possible to display the shape element designation screen for the side contour generating finished shape which has been already defined. Change of the set data is also possible. As the contour generating shape is a part of the multi-machining shape, the screen is displayed in the following sequence.*

A. When the GENERATING SHAPE screen is displayed (before the completion of contour generating shape designation)

The shape element designation screens for the side contour generating finished shape which has been already defined are displayed sequentially when the cursor control keys ← and → are pressed.

For the example on the previous page, the following screens can be displayed sequentially.



The (magenta) marker indicates where the finished shape (blue) is defined on the displayed shape element designation screen. Use the cursor control keys to move the marker.

B. When the M MACHINING SHAPE screen is displayed (after the completion of contour generating shape designation)

1) The shape designation screens for the multi-machining finished shape which has been already defined are displayed sequentially when the cursor control keys ← and → are pressed.

However, the shape element designation screens for the side contour generating finished shape are not displayed. The only screen which is displayed is the "GENERATING SHAPE" screen.

2) Display the "GENERATING SHAPE" screen of the multi-machining finished shape definition screens.

3) Press the **WRITE** *key. This allows the shape element designation screens for the side contour generating finished shape to be displayed.*

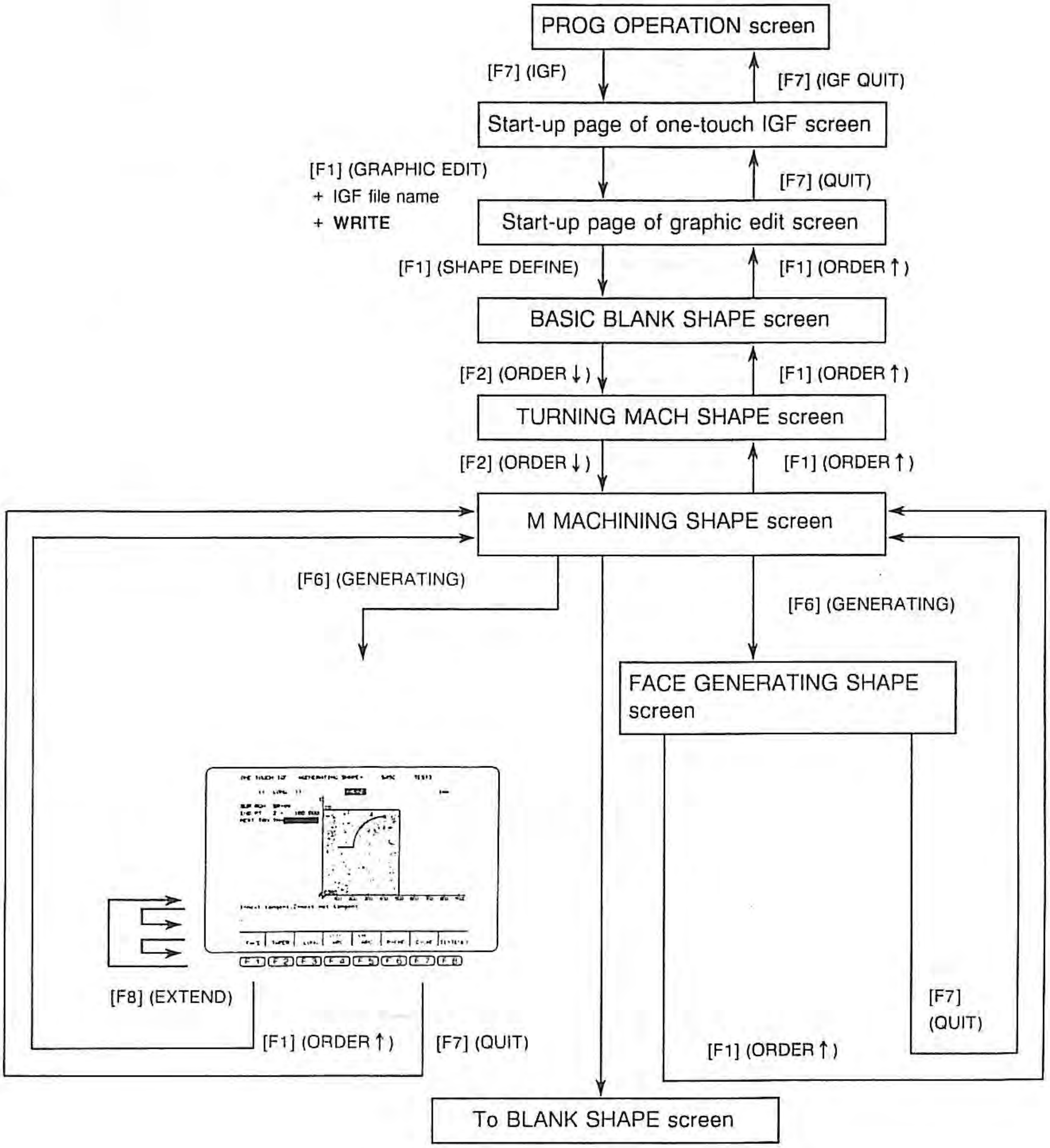
4) Press the function key F7 [QUIT]. The M MACHINING SHAPE screen will be displayed.

Screen Transfer:

Optional specifications to be selected:

- Multi-machining specification
- Contour generating machining specification

The SIDE GENERATING SHAPE screen is called from the PROG OPERATION screen as diagramed below.

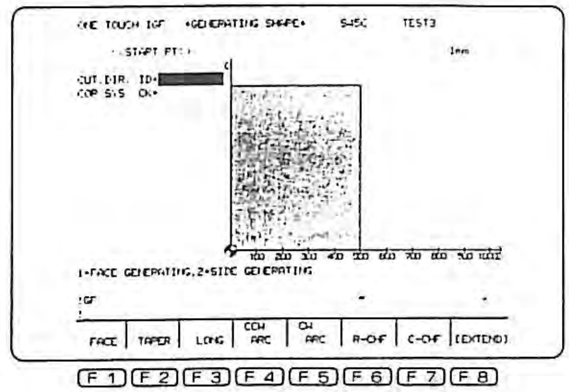


After the completion of finished shape definition, press the function key [F7] (QUIT). The CRT will return to the M MACHINING SHAPE screen.

(1) **START PT**

a) Function

Infeed direction, coordinate system, and the starting point for contour generating machining are designated.

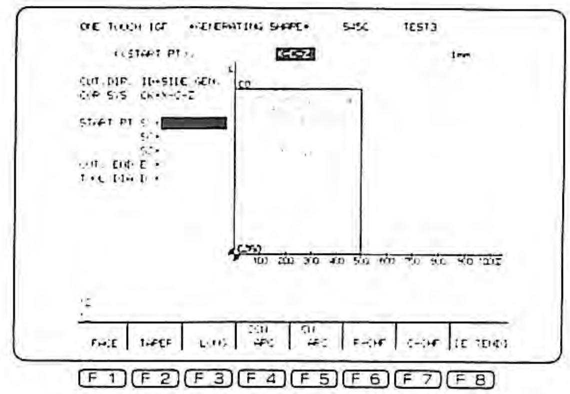


b) Data to be Designated

CUT. DIR. ID
COR SYS CK
START PT SX
SC
SZ
CUT. END EX
TOOL DIA. D

c) How to Designate

Press the function key [F6] (GENERATING) on the M MACHINING SHAPE screen. The START PT screen on which only CUT. DIR. ID and COR SYS CK are displayed will appear. Enter "2 = SIDE GENERATING" for CUT. DIR. ID. The data to be designated for START PT will be displayed.



CUT.DIR.ID

The infeed direction of contour generating machining is designated. As the prompt "1 = **FACE GENERATING, 2 = SIDE GENERATING**" is displayed, enter "2".

COR SYS CK

For side contour generating shape definition, only the X-C-Z coordinate system is available.

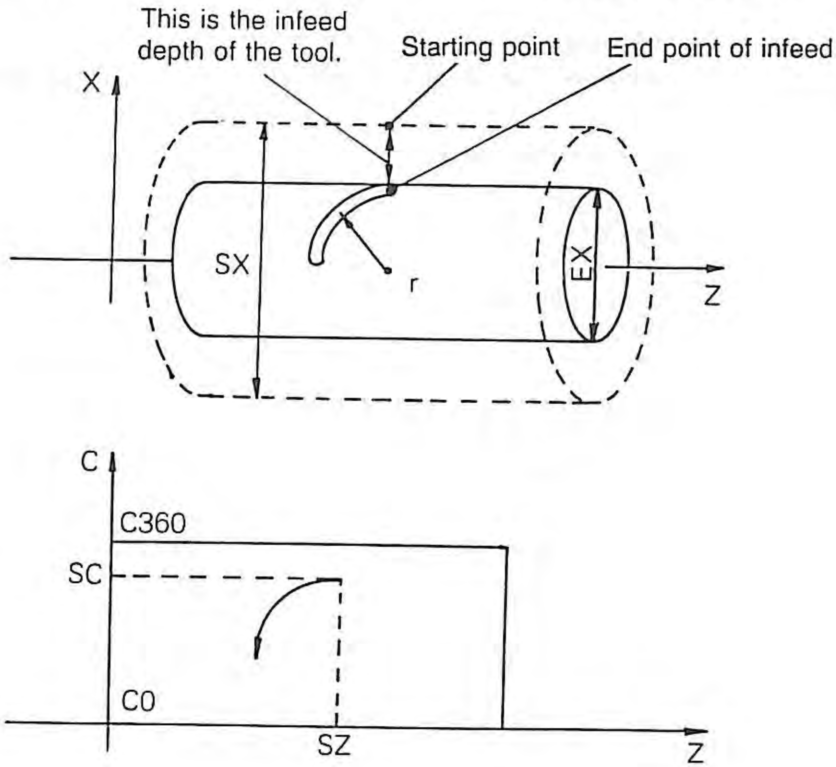
START PT SX, SC, SZ

The X-, C- and Z-coordinates of the shape definition starting point for SIDE GENERATING SHAPE are designated.

For START PT SX, enter the diameterial value.

CUT. END EX

The end point of infeed is designated. Enter the diametrical value.



TOOL DIA. D

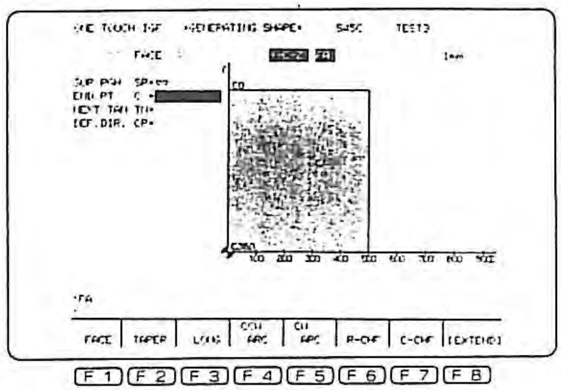
The diameter of the tool to be used is designated.

Note: *Contour generating shape definition cannot be executed unless the starting point is defined. Enter the data for all input items on the START PT screen on page 158.*

(2) FACE

a) Function

The shape of end face for SIDE GENERATING SHAPE is designated.



b) Data to be Designated

SUR RGH	SR
END PT	Z
	C
NEXT TAN	TN
DEF. DIR.	CR

c) How to Designate

Press the function key [F8] (EXTEND) until the function "FACE" is assigned to the function key [F1].

Press the function key [F1] (FACE). The FACE screen will be displayed.

SUR RGH SR

Surface roughness is designated. As the prompt "1 = ∇, 2 = ∇∇, 3 = ∇∇∇, 4 = ∇∇∇∇, 5 = ~" will be displayed, select the desired surface roughness. For example, to select ∇∇ as surface roughness, enter "2". For the data column of SUR RGH, the following data has been set in advance.

1) When any shape has been designated prior to end face definition, the surface roughness designated for that shape is initially set.

2) When the shape of end face is designated as the first shape following the starting point, the surface roughness set at the one-touch IGF No. 44 GENERATING FINISH FEEDRATE is set initially.

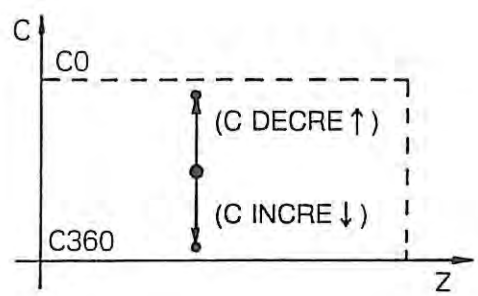
END PT Z, C

The Z- and C-coordinates of end face end point are designated.

Designate the C-coordinate within the range from −360° to +360° in reference to the C-coodinate of the starting point of taper.

DEF. DIR. CR

Since the C-axis is a rotary axis, the end point C can be approached from two directions. Enter the direction of approach. As the prompt is displayed, enter the number while referring to the drawing to the right.



1 = C DECRE ↑
2 = C INCRE ↓

The setting procedure for other data is the same as that for end face in TURNING MACH SHAPE definition. Refer to the explanation in 3. (2) "FACE" on page 88.

d) Conditions for Shape Determination

Refer to the explanation in 3. (2) "FACE" on page 88.

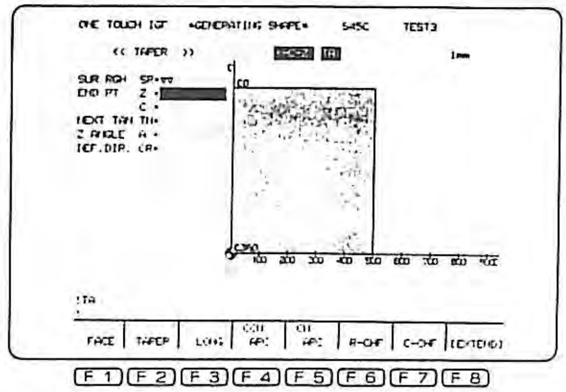
(3) TAPER

a) Function

The shape of taper for SIDE GENERATING SHAPE is designated.

b) Data to be Designated

SUR RGH	SR
END PT	Z
	C
NEXT TAN	TN
Z ANGLE	A
DEF. DIR.	CR



c) How to Designate

Press the function key [F8] (EXTEND) until the function "TAPER" is assigned to the function key [F2] .

Press the function key [F2] (TAPER). The TAPER screen will be displayed.

SUR RGH SR

Surface roughness is designated.

END PT Z, C

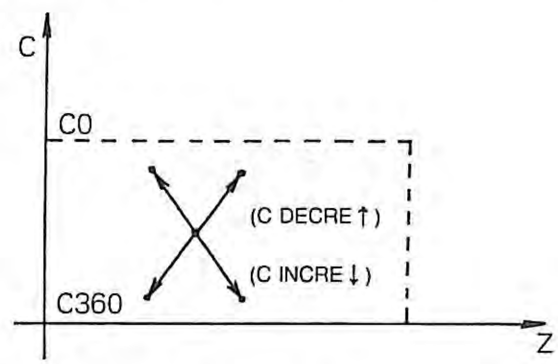
The Z- and C-coordinates of the end point of taper are designated.

Designate the C-coordinate within the range from −360° to +360° in reference to the C-coodinate of the starting point of taper.

DEF. DIR. CR

Since the C-axis is a rotary axis, the end point C can be approached from two directions. Enter the direction of approach. As the prompt is displayed, enter the number while referring to the drawing to the right.

1 = C DECRE ↑
2 = C INCRE ↓



The setting procedure for other data is the same as that for taper in TURNING MACH SHAPE definition. Refer to the explanation in 3. (3) "TAPER" on page 91.

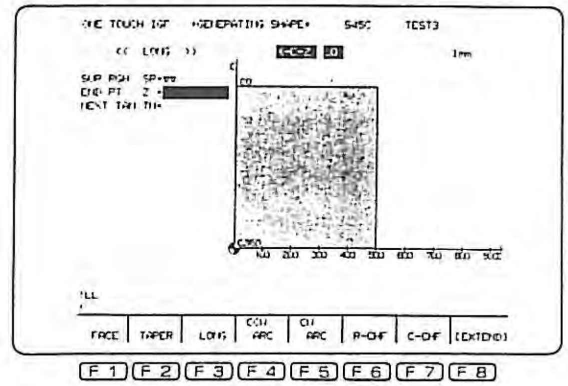
d) Conditions for Shape Determination

Refer to 3. (3) "TAPER" on page 91.

(4) **LONG**

a) Function

The shape in the longitudinal direction for SIDE GENERATING SHAPE is designated.



b) Data to be Designated

SUR RGH	SR
END PT	Z
	C
NEXT TAN	TN

c) How to Designate

Press the function key [F8] (EXTEND) until the function "LONG" is assigned to the function key [F3].

Press the function key [F3] (LONG). The LONG screen will be displayed.

SUR RGH SR

Surface roughness is designated.

END PT Z, C

The Z- and C-coordinates of the end point in the longitudinal direction are designated.

The setting procedure for other data is the same as that for longitudinal portion in TURNING MACH SHAPE definition. Refer to the explanation in 3. (4) "LONG" on page 95.

d) Conditions for Shape Determination

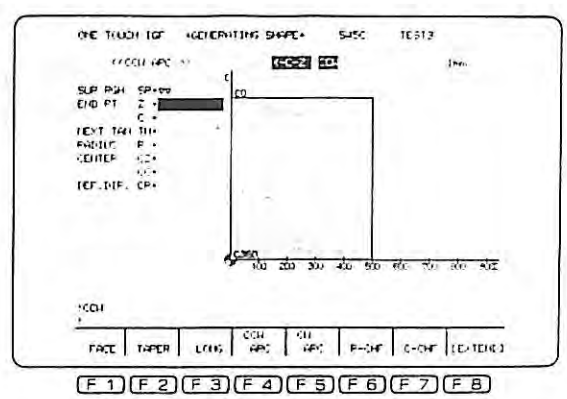
Refer to the explanation in 3. (4) "LONG" on page 95.

(5) **CCW ARC**

(6) **CW ARC**

a) Function

The shape of CCW arc or CW arc for SIDE GENERATING SHAPE is designated.



b) Data to be Designated

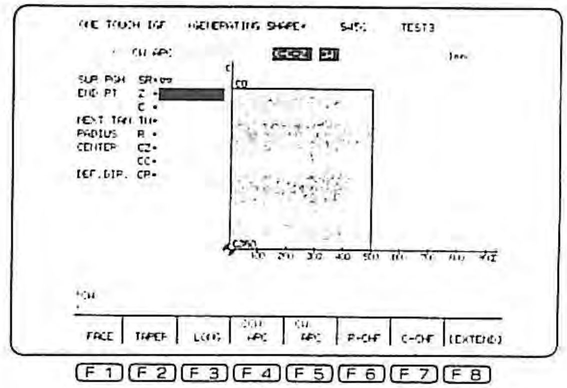
SUR RGH	SR
END PT	Z
	C
NEXT TAN	TN
RADIUS	R
CENTER	CZ
	CC
DEF. DIR.	CR

c) How to Designate

Press the function key [F8] (EXTEND) until the function "CCW ARC" is assigned to the function key [F4] and "CW ARC" to [F5].

Press the function key [F4] (CCW ARC). The CCW ARC screen will be displayed.

Press the function key [F5] (CW ARC). The CW ARC screen will be displayed.



SUR RGH SR

Surface roughness is designated.

END PT Z, C

The Z- and C-coordinates of the end point of CCW or CW arc are designated.

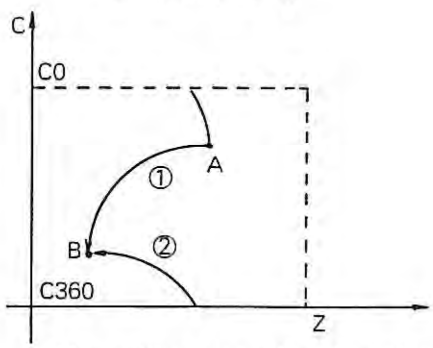
CENTER CZ, CC

The Z- and C-coordinates of the center point of CCW or CW arc are designated.

DEF. DIR. CR

Since the C-axis is a rotary axis, the end point C can be approached from two directions. Enter the direction of approach. As the prompt is displayed, enter the number while referring to the drawing to the right.

1 = C DECRE ↑
2 = C INCRE ↓



To connect point A and point B by CCW arc

① = C INCRE ↓
② = C DECRE ↑

To designate an arc which is longer than a semi-cicle, designate by dividing the arc into multiple arcs.

The setting procedure for other data is the same as that for arc in TURNING MACH SHAPE definition. Refer to the explanation in 3. (5) "CCW ARC" or (6) "CW ARC" on page 97.

d) Conditions for Shape Determination

Refer to 3. (5) "CCW ARC" or (6) "CW ARC" on page 97.

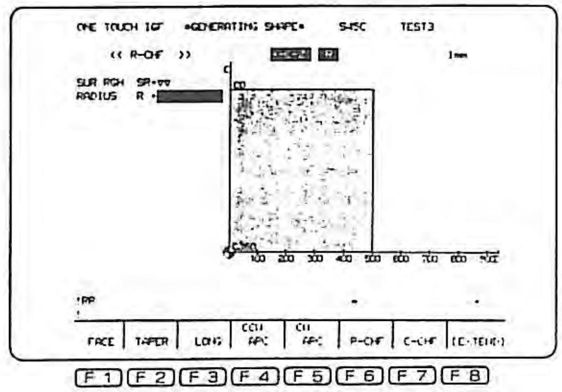
(7) **R-CHF**

a) Function

The shape of rounding R for SIDE GENERATING SHAPE is designated.

b) Data to be Designated

SUR RGH SR
RADIUS R



c) How to Designate

Press the function key [F8] (EXTEND) until the function "R-CHF" is assigned to the function key [F6].

Press the function key [F6] (R-CHF). The R-CHF screen will be displayed.

The data setting procedure is the same as that for rounding R in TURNING MACH SHAPE definition. Refer to the explanation in 3. (7) "R-CHF" on page 101.

d) Conditions for Shape Determination

Refer to 3. (7) "R-CHF" on page 101.

Note: *Rounding R cannot be designated as the first shape following the shape definition starting point or the last shape preceding the shape definition end point.*

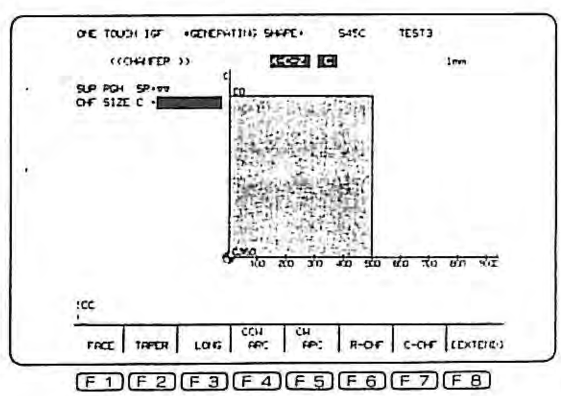
(8) **CHAMFER**

a) Function

The shape of chamfer for SIDE GENERATING SHAPE is designated.

b) Data to be Designated

SUR RGH SR
CHF SIZE C



c) How to Designate

Press the function key [F8] (EXTEND) until the function "C-CHF" is assigned to the function key [F7].

Press the function key [F7] (C-CHF). The C-CHF screen will be displayed.

The data setting procedure is the same as that for chamfer in TURNING MACH SHAPE definition. Refer to the explanation in 3. (8) "CHAMFER" on page 103.

d) Conditions for Shape Determination

Refer to 3. (8) "CHAMFER" on page 103.

Note: *Chamfer cannot be designated as the first shape following the shape definition starting point or the last shape preceding the shape definition end point.*

(9) Insertion of Shape Element

a) Function

Shape element is inserted between designated two shape elements.

b) How to Designate

Shape element is inserted in the same manner as in TURNING MACH SHAPE definition. Refer to the explanation in 3. (14) "Insertion of Shape Element" on page 118.

(10) Deletion of Shape Element

a) Function

The designated shape element is deleted.

b) How to Designate

Shape element is deleted in the same manner as in TURNING MACH SHAPE definition. Refer to the explanation in 3. (15) "Deletion of Shape Element" on page 120.

Note: *The function key "DELETE" on the SIDE GENERATING SHAPE screen is used to delete the shape element of side contour generating machining. To delete the shape element of side contour generating machining with the multi-machining shape definition mode selected after the completion of side contour generating finished shape definition, use the function key "DELETE" in the multi-machining shape definition mode.*

(11) Re-display of Shape

a) Function

The shape defined by entered data is re-displayed.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "SHAPE RE-DIS." is assigned to the function key [F6].

Press the function key [F6] (SHAPE RE-DIS.). The shape defined by the entered data will be re-displayed.

The re-display function is used to re-display the finished shape after the shape element data has been changed.

(12) Shape Referencing Function

Refer to Section 6, 3. (18) "Shape Referencing Function".

6. FACE GENERATING SHAPE Definition (optional)

The explanation that follows is the procedure for finished shape definition for front contour generating machining.

The following functions are provided to define the turning shape.

(1) START PT

(2) FACE

(3) TAPER

(4) LONG

(5) CCW ARC

(6) CW ARC

(7) R-CHF

(8) CHAMFER

(9) Insertion of shape element

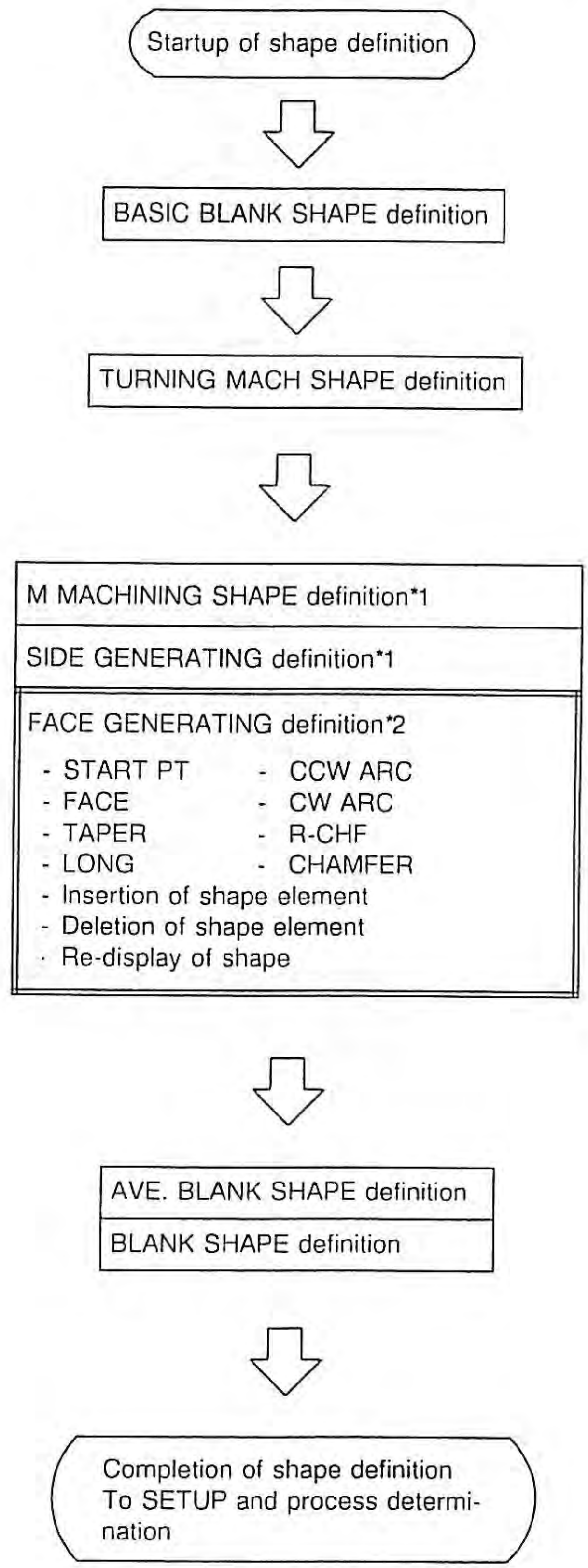
(10) Deletion of shape element

(11) Re-display of shape

(12) Shape referencing function

In the contour generating machining shape definition process, first designate the shape definition starting point and then enter the shape elements (2) to (8) sequentially according to the finished shape. The functions (2) to (8) are assigned to the function keys.

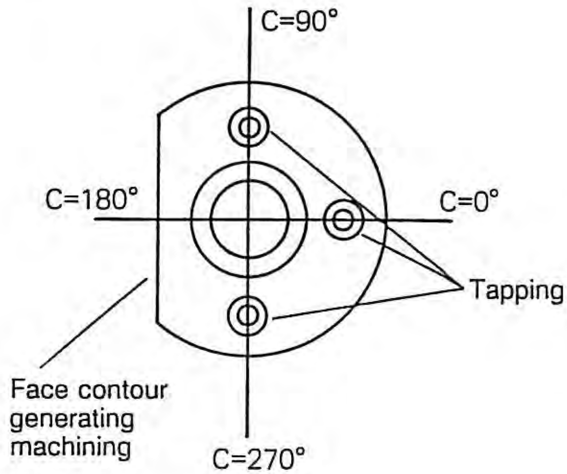
The relationship between the sequence of shape definition and the front contour generating shape definition is diagramed below.

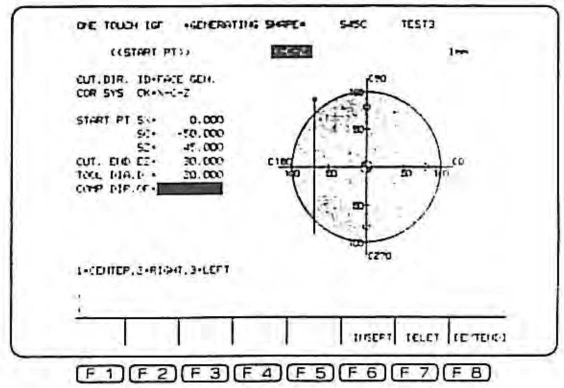


*1 Optional: Multi-machining specification
*2 Optional: Contour generating machining specification

The face contour generating shape definition screen is displayed by pressing the function key [F6] (GENERATING) is pressed on the M MACHINING SHAPE screen. When the function key [F7] (QUIT) is pressed after the completion of side contour generating shape definition, the M MACHINING SHAPE screen will be displayed.

Face contour generating shape definition is carried out in the following sequence.





Face contour generating finished shape (blue)

1) After the completion of multi-machining shape definition, press the function key [F6] (GENERATING) to advance to the GENERATING SHAPE screen.
2) Enter "1 = FACE GENERATING" for CUT. DIR. ID and designate COR SYS CK.
3) Enter the data for START PT, CUT. END EX, and TOOL DIA. D.
4) Select the shape element and define the contour generating finished shape.

   Necessary shape elements: FACE
5) After the completion of side contour generating finished shape, press the function key [F7] (QUIT).

   The CRT will return to the M MACHINING SHAPE screen.

To define multiple contour generating finished shapes, repeat the steps 1) to 5).

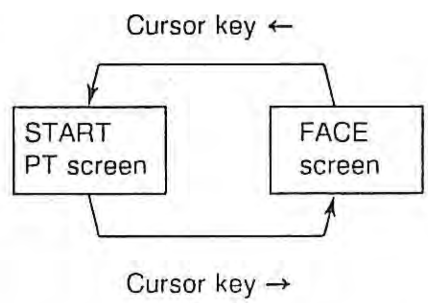
The one-touch IGF system will check the entered data when the function key [F7] (QUIT) is pressed.

Note: *Using the cursor control keys ← and →, it is possible to display the shape element designation screen for the face contour generating finished shape which has been already defined. Change of the set data is also possible. As the contour generating shape is a part of the multi-machining shape, the screen is displayed in the following sequence.*

A. When the GENERATING SHAPE screen is displayed (before the completion of contour generating shape designation)

The shape element designation screen for the face contour generating finished shape which has been already defined is displayed sequentially when the cursor control keys ← and → are pressed.

For the example on the previous page, the following screens can be displayed sequentially.



The (magenta) marker indicates where the finished shape (blue) is defined on the displayed shape element designation screen. Use the cursor control keys to move the marker.

B. When the M MACHINING SHAPE screen is displayed (after the completion of contour generating shape designation)

1) The shape designation screens for the multi-machining finished shape which has been already defined are displayed sequentially when the cursor control keys ← and → are pressed.

However, the shape element designation screens for the side contour generating finished shape are not displayed. The only screen which is displayed is the "GENERATING SHAPE" screen.

2) Display the "GENERATING SHAPE" screen of the multi-machining finished shape definition screens.

*3) Press the **WRITE** key. This allows the shape element designation screens for the side contour generating finished shape to be displayed.*

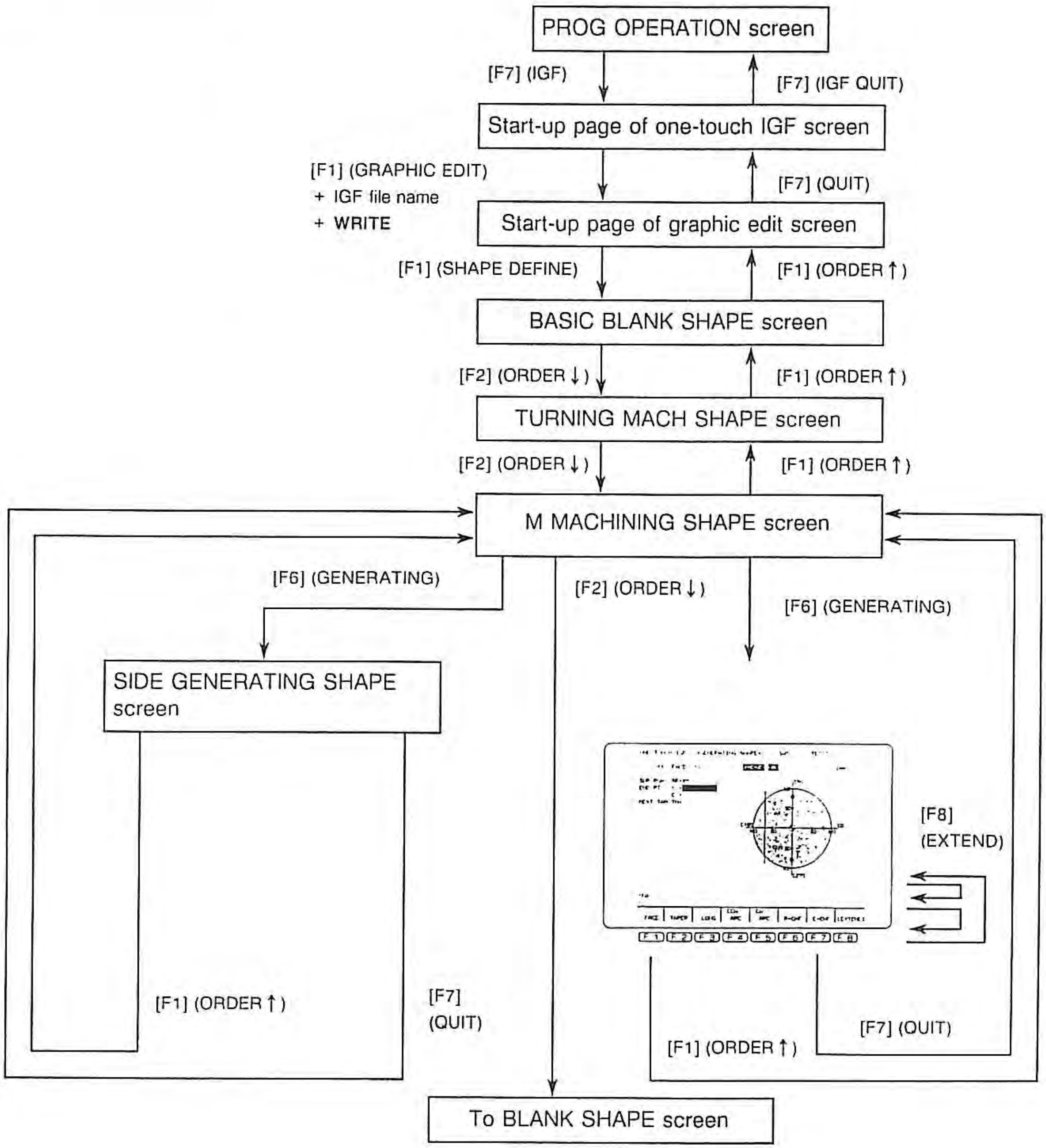
4) Press the function key F7 [QUIT]. The M MACHINING SHAPE screen will be displayed.

Screen Transfer:

Optional specifications to be selected:

- Multi-machining specification
- Contour generating machining specification

The FACE GENERATING SHAPE screen is called from the PROG OPERATION screen as diagramed below.

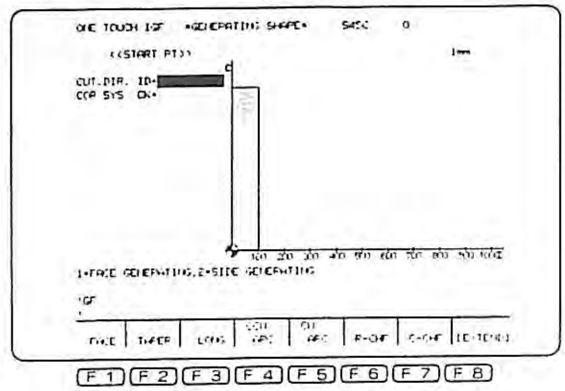


After the completion of finished shape definition, press the function key [F7] (QUIT). The CRT will return to the M MACHINING SHAPE screen.

(1) START PT

a) Function

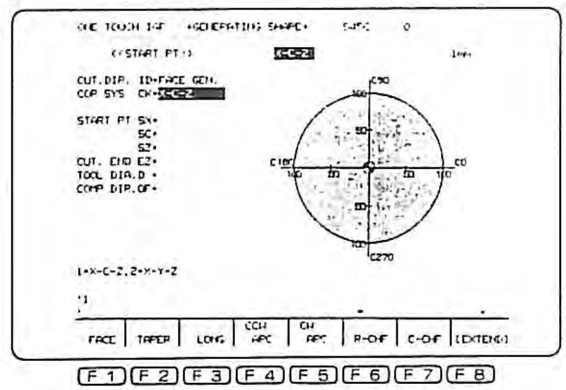
Infeed direction, coordinate system, and the starting point for contour generating machining are designated.



b) Data to be Designated

(X-C-Z coordinate system)

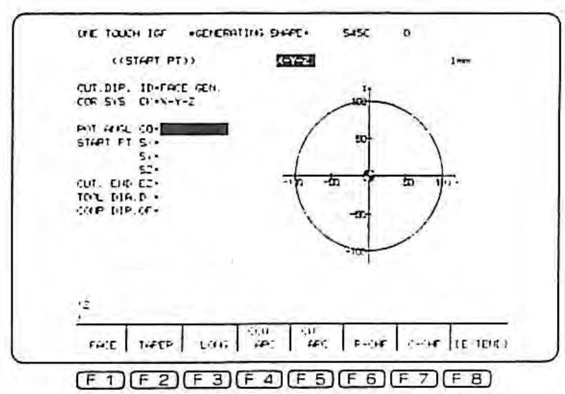
CUT. DIR. ID
COR SYS CK
START PT SX
SC
SZ
CUT. END EZ
TOOL DIA. D
COMP DIR. OF



(X-Y-Z coordinate system)

CUT. DIR. ID
COR SYS CK
ROT ANGL CO
START PT SX
SC
SZ
CUT. END EZ
TOOL DIA. D
COMP DIR. OF

c) How to Designate



Press the function key [F6] (GENERATING) on the M MACHINING SHAPE screen. The START PT screen on which only CUT. DIR. ID and COR SYS CK are displayed will appear. Enter "1 = SIDE GENERATING" for CUT. DIR. ID and "1 = X-C-Z" for COR SYS CK. The data to be designated for START PT in the X-C-Z coordinate system will be displayed.

When "1 = SIDE GENERATING" is designated for CUT. DIR. ID and "2 = X-Y-Z" for COR SYS CK, the data to be designated for START PT in the X-Y-Z coordinate system will be displayed.

CUT. DIR. ID

The infeed direction of contour generating machining is designated. As the prompt "1 = **FACE GENERATING, 2 = SIDE GENERATING**" is displayed, enter "1".

COR SYS CK

The coordinate system for face contour generating shape definition is designated. As the prompt "1 = **X-C-Z, 2 = X-Y-Z**" is displayed, enter "1" or "2".

START PT SX, SC, SZ (X-C-Z coordinate system)

The X-, C- and Z-coordinates of the shape definition starting point for FACE GENERATING SHAPE are designated.

For START PT SX, enter the diameterial value.

START PT SX, SY, SZ (X-Y-Z coordinate system)

The X-, Y- and Z-coordinates of the shape definition starting point for FACE GENERATING SHAPE are designated.

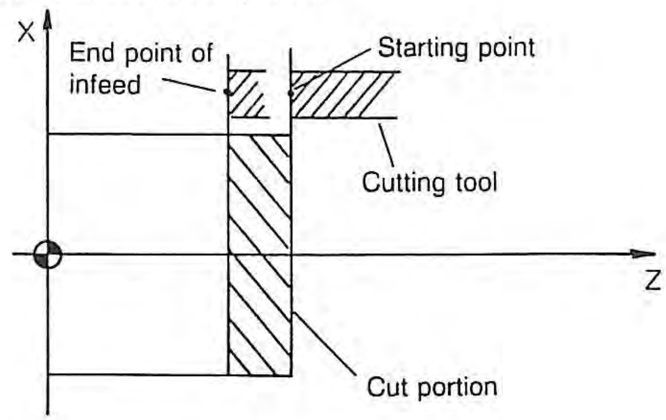
For START PT SX, enter the value in radius.

ROT ANGL CO

The reference angle in the X-Y-Z coordinate system is designated.

CUT. END EZ

The end point of infeed is designated.



TOOL DIA. D

The diameter of the tool to be used is designated.

COMP DIR. OF

COMP DIR. OF refers to the offset direction of tool diameter with respect to the finished shape.

Tool is to the right of finished shape ...... RIGHT

Tool is to the left of finished shape .... LEFT

Tool center is on the finished shape ...... CENTER

As the prompt

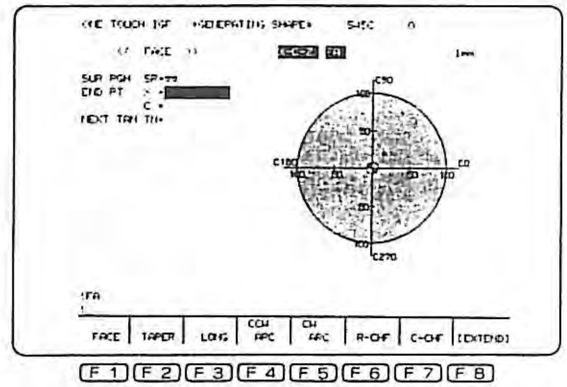
"Select cutting side of workpiece, 1 = CENTER, 2 = RIGHT, 3 = LEFT," is displayed, enter "1", "2", or "3" to select the direction.

Note: *Contour generating shape definition cannot be executed unless the starting point is defined. Enter the data for all input items on the START PT screen on page 171.*

(2) **FACE**

a) Function

The shape of end face for FACE GENERATING SHAPE is designated.

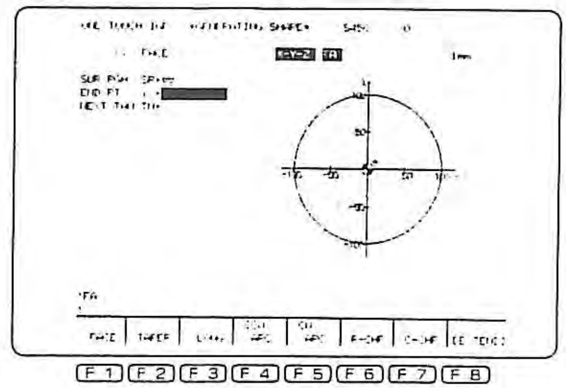


b) Data to be Designated

(X-C-Z coordinate system)

SUR RGH	SR
END PT	X
	C
NEXT TAN	TN

(X-Y-Z coordinate system)



SUR RGH	SR
END PT	X
	Y
NEXT TAN	TN

c) How to Designate

Press the function key [F8] (EXTEND) until the function "FACE" is assigned to the function key [F1] .

Press the function key [F1] (FACE). The FACE screen will be displayed.

SUR RGH SR

Surface roughness is designated. As the prompt "1 = ∇, 2 = ∇∇, 3 = ∇∇∇, 4 = ∇∇∇∇, 5 = ~" will be displayed, select the desired surface roughness. For example, to select ∇∇ as surface roughness, enter "2". For the data column of SUR RGH, the following data has been set in advance.

1) When any shape has been designated prior to end face definition, the surface roughness designated for that shape is initially set.

2) When the shape of end face is designated as the first shape following the starting point, the surface roughness set at the one-tough IGF No. 44 GENERATING FINISH FEEDRATE is set initially.

END PT X, C (X-C-Z coordinate system)

The X- and C-coordinates of end face end point are designated. For END PT X, a diametrical value must be input.

END PT X, Y (X-Y-Z coordinate system)

The X- and Y-coordinates of end face end point are designated. For END PT X, enter the value in radius.

The setting procedure for other data is the same as that for end face in TURNING MACH SHAPE definition. Refer to the explanation in 3. (2) "FACE" on page 88.

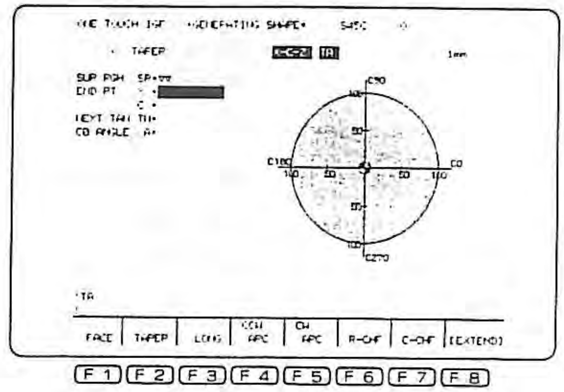
d) Conditions for Shape Determination

Refer to the explanation in 3. (2) "FACE" on page 88.

(3) TAPER

a) Function

The shape of taper for SIDE GENERATING SHAPE is designated.



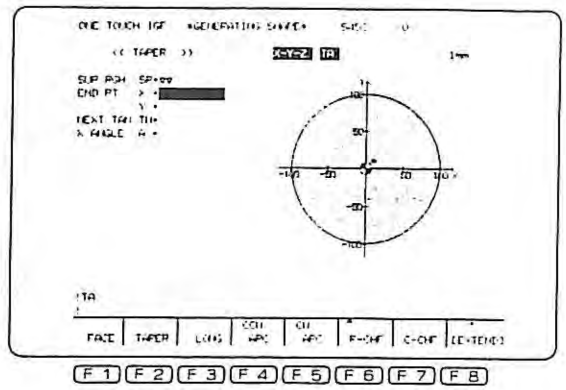
b) Data to be Designated

(X-C-Z coordinate)

SUR RGH	SR
END PT	X
	C
NEXT TAN	TN
CO ANGLE	A

(X-Y-Z coordinate)

SUR RGH	SR
END PT	X
	Y
NEXT TAN	TN
X ANGLE	A



c) How to Designate

Press the function key [F8] (EXTEND) until the function "TAPER" is assigned to the function key [F2].

Press the function key [F2] (TAPER). The TAPER screen will be displayed.

SUR RGH SR

Surface roughness is designated.

END PT X, C (X-C-Z coordinate system)

The X- and C-coordinates of the end point of taper are designated. For END PT X, a diametrical value must be input.

END PT X, Y (X-Y-Z coordinate system)

The X- and Y-coordinates of the end point of taper are designated. For END PT X, enter the value in radius.

X ANGLE A

Angle made between taper and X-axis is designated. (degrees)

The setting procedure for other data is the same as that for taper in TURNING MACH SHAPE definition. Refer to the explanation in 3. (3) "TAPER" on page 91.

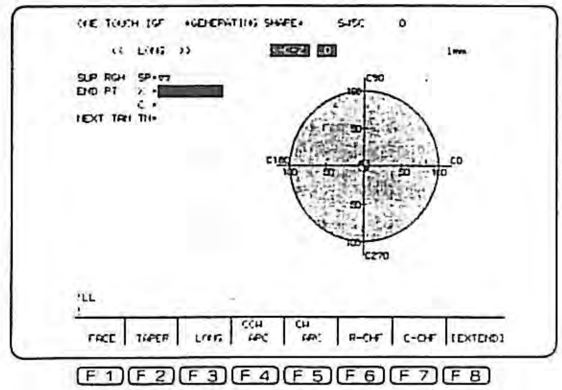
d) Conditions for Shape Determination

Refer to 3. (3) "TAPER" on page 91.

(4) LONG

a) Function

The shape in the longitudinal direction for FACE GENERATING SHAPE is designated.



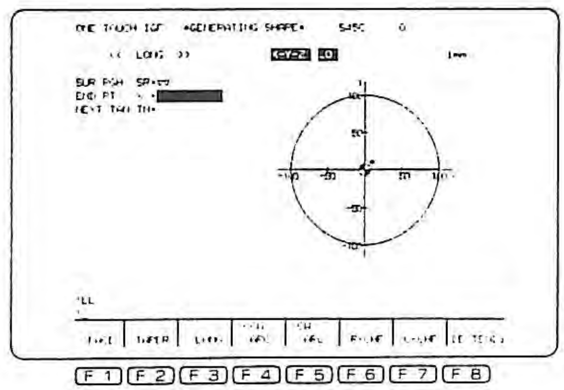
b) Data to be Designated

(X-C-Z coordinate system)

SUR RGH SR
END PT X
C
NEXT TAN TN

(X-Y-Z coordinate system)

SUR RGH SR
END PT X
Y
NEXT TAN TN



c) How to Designate

Press the function key [F8] (EXTEND) until the function "LONG" is assigned to the function key [F3].

Press the function key [F3] (LONG). The LONG screen will be displayed.

SUR RGH SR

Surface roughness is designated.

END PT X, C (X-C-Z coordinate system)

The X- and C-coordinates of the end point of taper are designated. For END PT X, a diametrical value must be input.

END PT X, Y (X-Y-Z coordinate system)

The X- and Y-coordinates of the end point of taper are designated. For END PT X, enter the value in radius.

The setting procedure for other data is the same as that for longitudinal portion in TURNING MACH SHAPE definition. Refer to the explanation in 3. (4) "LONG" on page 95.

d) Conditions for Shape Determination

Refer to the explanation in 3. (4) "LONG" on page 95.

(5) CCW ARC

(6) CW ARC

a) Function

The shape of CCW arc or CW arc for FACE GENERATING SHAPE is designated.

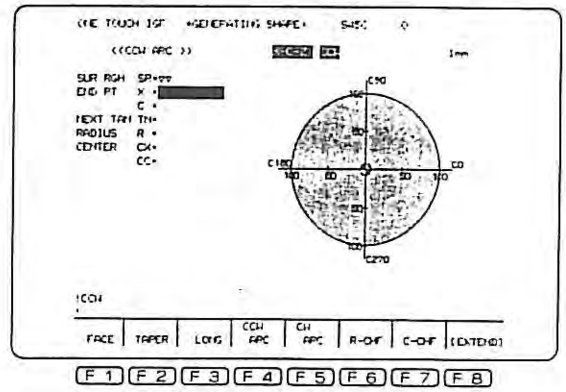
b) Data to be Designated

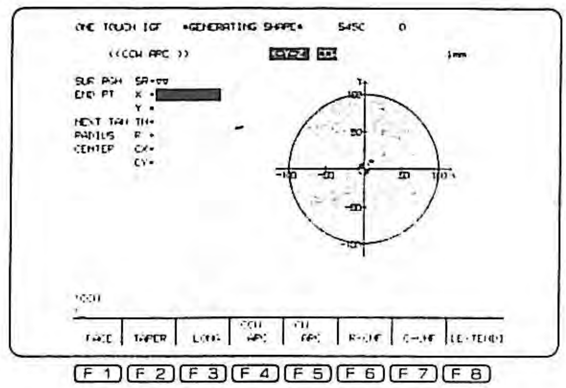
(X-C-Z coordinate)

SUR RGH	SR
END PT	X
	C
NEXT TAN	TN
RADIUS	R
CENTER	CX
	CC

(X-Y-Z coordinate)

SUR RGH	SR
END PT	X
	Y
NEXT TAN	TN
RADIUS	R
CENTER	CX
	CY





c) How to Designate

Press the function key [F8] (EXTEND) until the function "CCW ARC" is assigned to the function key [F4] and "CW ARC" to [F5].

Press the function key [F4] (CCW ARC). The CCW ARC screen will be displayed.

Press the function key [F5] (CW ARC). The CW ARC screen will be displayed.

SUR RGH SR

Surface roughness is designated.

END PT X, C (X-C-Z coordinate system)

The X- and C-coordinates of the end point of CW or CCW arc are designated. For END PT X, a diametrical value must be input.

END PT X, Y (X-Y-Z coordinate system)

The X- and Y-coordinates of the end point of CW or CCW arc designated. For END PT X, enter the value in radius.

CENTER CX, CC (X-C-Z coordinate system)

The X- and C-coordinates of the center position of CW or CCW arc are designated. For CENTER CX, a diametrical value must be input.

CENTER CX, CY (X-Y-Z coordinate system)

The X- and Y-coordinates of the center position of CW or CCW arc are designated. For END PT X, enter the value in radius.

To designate an arc which is longer than a semi-cicle, designate by dividing the arc into multiple arcs.

The setting procedure for other data is the same as that for arc in TURNING MACH SHAPE definition. Refer to the explanation in 3. (5) "CCW ARC" or (6) "CW ARC" on page 97.

d) Conditions for Shape Determination

Refer to 3. (5) "CCW ARC" or (6) "CW ARC" on page 97.

(7) **R-CHF**

a) Function

The shape of rounding R for FACE GENERATING SHAPE is designated.

b) Data to be Designated

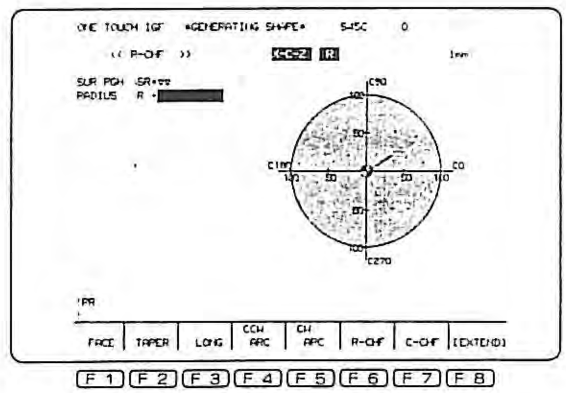
(X-C-Z coordinate)

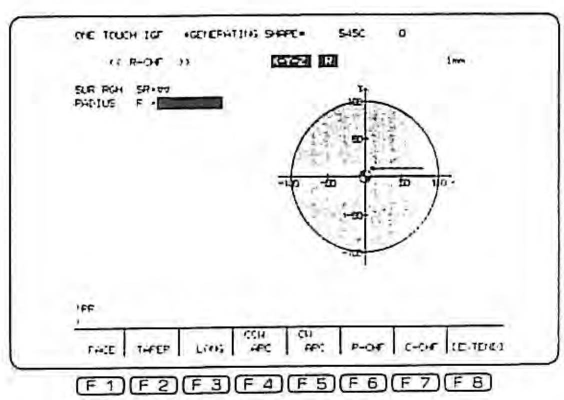
SUR RGH SR
RADIUS R

(X-Y-Z coordinate)

SUR RGH SR
RADIUS R

c) How to Designate





Press the function key [F8] (EXTEND) until the function "R-CHF" is assigned to the function key [F6].

Press the function key [F6] (R-CHF). The R-CHF screen will be displayed.

The data setting procedure is the same as that for rounding R in TURNING MACH SHAPE definition. Refer to the explanation in 3. (7) "R-CHF" on page 101.

d) Conditions for Shape Determination

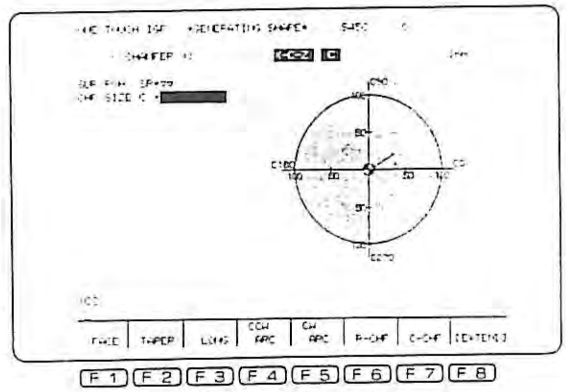
Refer to 3. (7) "R-CHF" on page 101.

Note: *Rounding R cannot be designated as the first shape following the shape definition starting point or the last shape preceding the shape definition end point.*

(8) CHAMFER

a) Function

The shape of chamfer for FACE GENERATING SHAPE is designated.



b) Data to be Designated

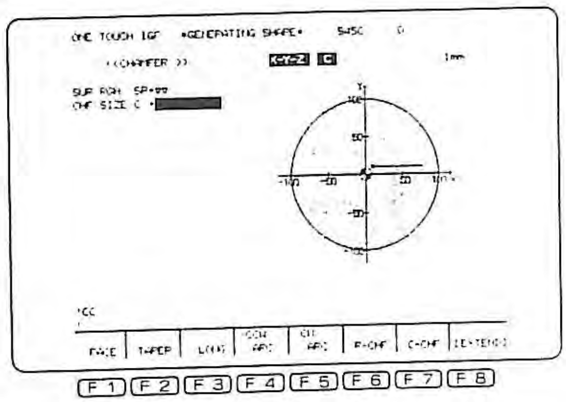
(X-C-Z coordinate)

SUR RGH SR
CHF SIZE C

(X-Y-Z coordinate)

SUR RGH SR
CHF SIZE C

c) How to Designate



Press the function key [F8] (EXTEND) until the function "C-CHF" is assigned to the function key [F7].

Press the function key [F7] (C-CHF). The C-CHF screen will be displayed.

The data setting procedure is the same as that for chamfer in TURNING MACH SHAPE definition. Refer to the explanation in 3. (8) "CHAMFER" on page 103.

d) Conditions for Shape Determination

Refer to 3. (8) "CHAMFER" on page 103.

Note: *Chamfer cannot be designated as the first shape following the shape definition starting point or the last shape preceding the shape definition end point.*

(9) Insertion of Shape Element

a) Function

Shape element is inserted between designated two shape elements.

b) How to Designate

Shape element is inserted in the same manner as in TURNING MACH SHAPE definition. Refer to the explanation in 3. (14) "Insertion of Shape Element" on page 118.

(10) **Deletion of Shape Element**

a) Function

The designated shape element is deleted.

b) How to Designate

Shape element is deleted in the same manner as in TURNING MACH SHAPE definition. Refer to the explanation in 3. (15) "Deletion of Shape Element" on page 120.

Note: The function key "DELETE" on the SIDE GENERATING SHAPE screen is used to delete the shape element of side contour generating machining. To delete the shape element of side contour generating machining with the multi-machining shape definition mode selected after the completion of side contour generating finished shape definition, use the function key "DELETE" in the multi-machining shape definition mode.

(11) **Re-display of Shape**

a) Function

The shape defined by entered data is re-displayed.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "SHAPE RE-DIS." is assigned to the function key [F6].

Press the function key [F6] (SHAPE RE-DIS.). The shape defined by the entered data will be re-displayed.

The re-display function is used to re-display the finished shape after the shape element data has been changed.

(12) **Shape Referencing Function**

Refer to Section 6, 3. (18) "Shape Referencing Function".

SECTION 7 BLANK MATERIAL SHAPE DEFINITION (SHAPE DEFINITION)

1. What is the Function of "Blank Material Shape Definition"?

In this section, blank material shape definition of the one-touch IGF is explained.

In the BASIC BLANK SHAPE definition process, rough blank material shape and type of blank material shape has been designated.

When ROUND BAR is set for BLANK SIZE, the blank material shape is defined by the data entered for OUT DIA. OD, OUT LENG. OL, IN DIA. ID, and IN LENG. IL, and nothing is necessary to be defined in this process. However, when AVERAGE STICK or FREE SHAPE is set for BLANK SIZE, it is necessary to define the detailed blank material shape. In this section, the procedure to define such blank material shape is explained.

The finished shape definition process consists of the following shape definition steps.

Startup of shape definition

⇩

BASIC BLANK SHAPE definition

⇩

TURNING MACH SHAPE definition

⇩

* Optional:
Multi-machining specification,
Contour generating machining
specification

M MACHINING SHAPE definition

SIDE GENERATING definition

FACE GENERATING definition

⇩

The detailed blank material shape must be defined depending on the type of blank material shape. ------

AVE. BLANK SHAPE definition

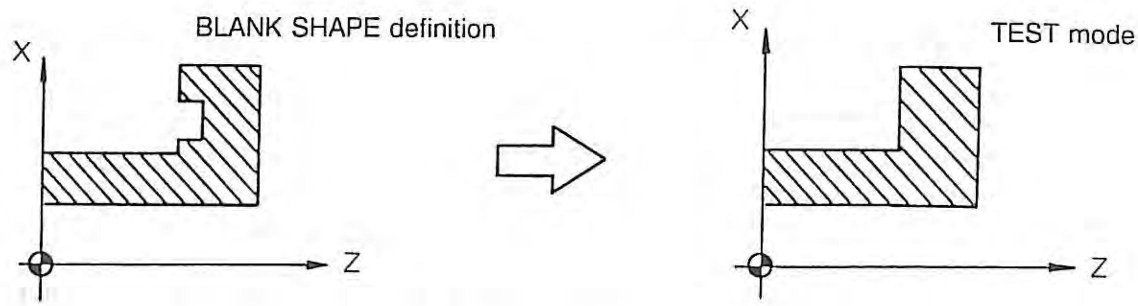
BLANK SHAPE definition

⇩

Completion of shape definition
To SETUP and process determination

Note: *The shape defined in this process is simulated on the CRT as the blank material shape in the process TEST mode. However, the blank material shape as the drawing below (with concave portion on the rear end face) cannot be simulated correctly.*

Example:

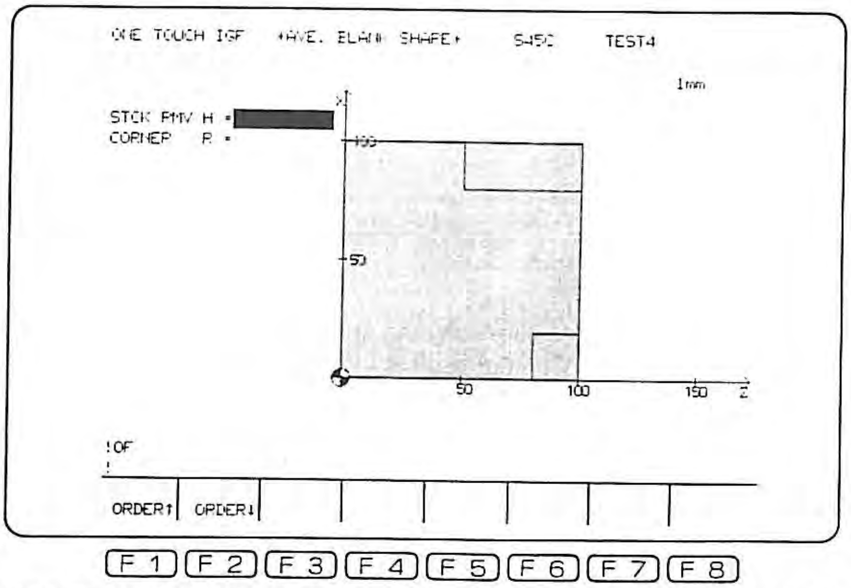


Similarly, in the AVE. BLANK SHAPE definition process, there are cases where blank material shape cannot be simulated correctly when attempting to display the impossible-to-turn shape with stock removal.

2. AVE. BLANK SHAPE Definition

The explanation that follows is the procedure for shape definition for blank material with uniform stock removal.

AVE. BLANK SHAPE refers to the shape of blank material such as casting which has uniform stock removal in respect to the finished shape. With the one-touch IGF, AVE. BLANK SHAPE can be defined only by setting the data for STCK RMV H (stock removal) and CORNER R.



To enable AVE. BLANK SHAPE definition, designate AVERAGE STICK for BLANK SIZE on the BASIC BLANK SHAPE screen.

The relationship between the sequence of shape definition and the AVE. BLANK SHAPE shape definition is diagramed below.

Startup of shape definition

⇩

Designate AVERAGE STICK for BLANK SIZE. ------ BASIC BLANK SHAPE definition

⇩

TURNING MACH SHAPE definition

* Optional:
Multi-machining specification,
Contour generating machining
specification

⇩

M MACHINING SHAPE definition

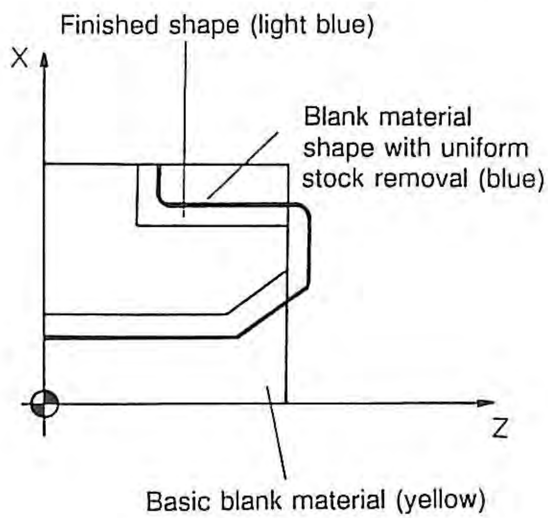
SIDE GENERATING definition

FACE GENERATING definition

⇩ [F2] (ORDER ↓)

AVE. BLANK SHAPE definition

Designation of STCK RMV H and CORNER R



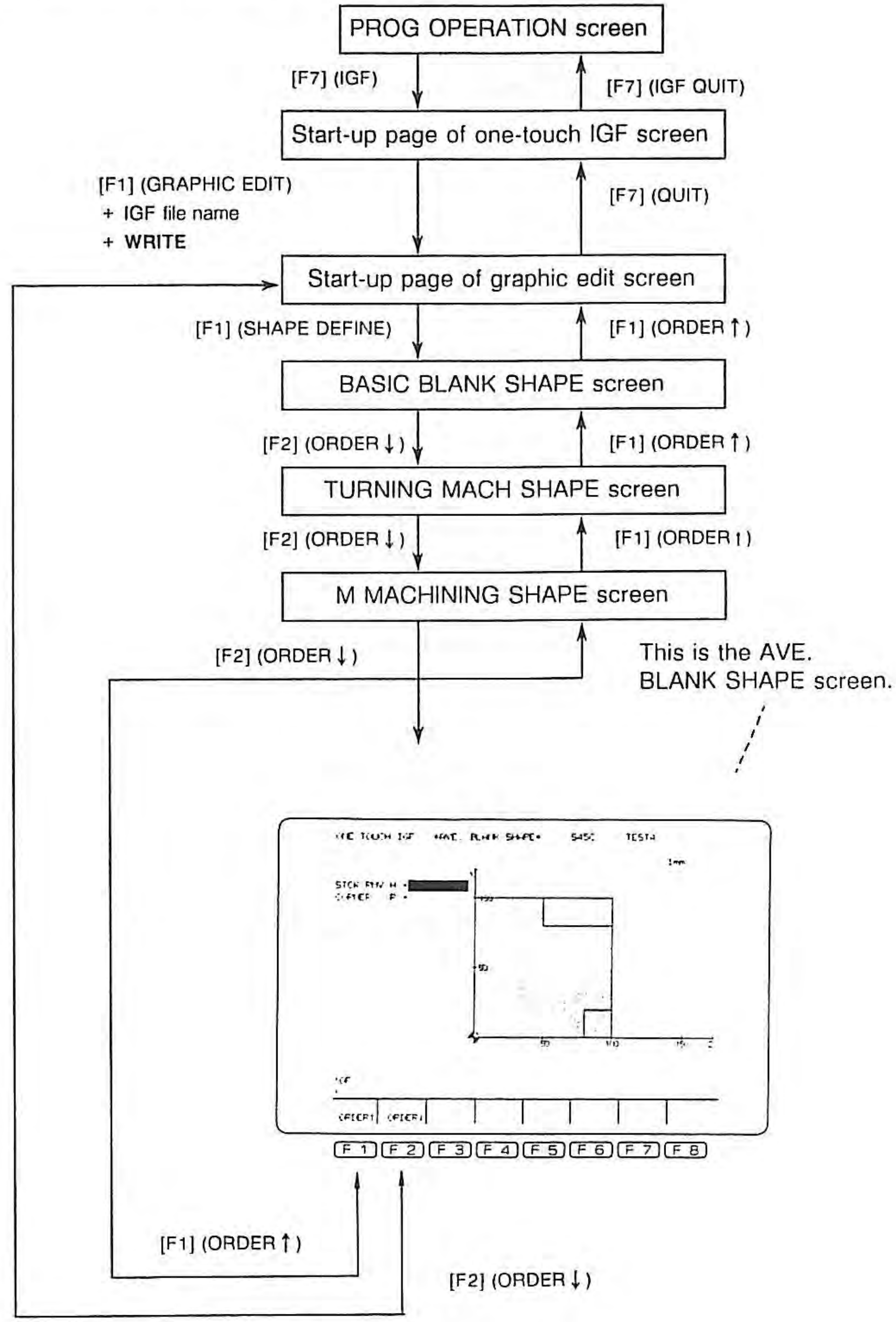
After the completion of AVE. BLANK SHAPE definition, press the function key [F2] (ORDER ↓). All shape definition processes are terminated. --------------- ⇩ [F2] (ORDER ↓)

Completion of shape definition
To SETUP and process determination

Screen Transfer:

The AVE. BLANK SHAPE screen is called from the PROG OPERATION screen as diagramed below.

Optional specification to be selected: Multi-machining specification

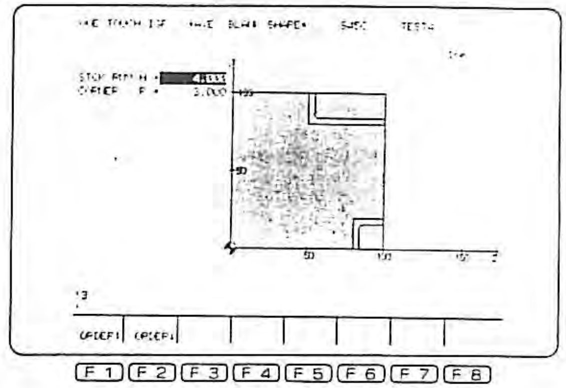


a) Data to be Designated

STCK RMV H
CORNER R

b) How to Designate

Move the cursor to the desired data column using the cursor control keys and enter the data.



STCK RMV H

Stock removal of the blank material shape is designated.

CORNER R

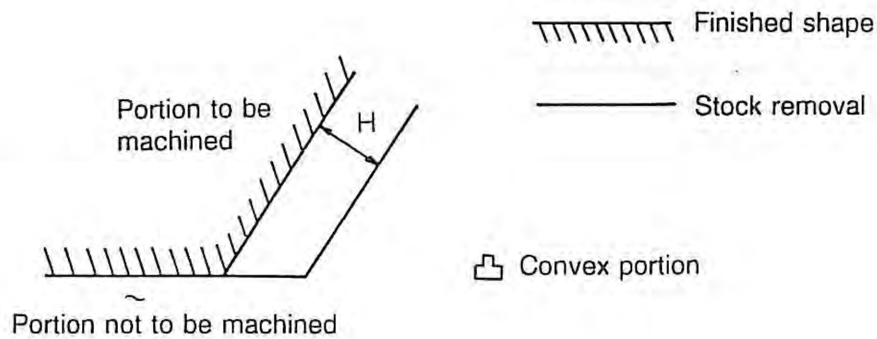
The radius of R at the corner of blank material is designated.

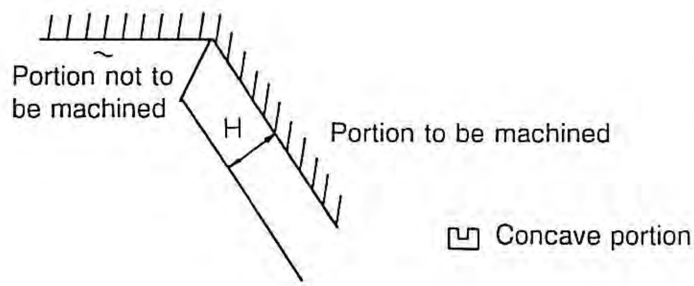
Display the AVE. BLANK SHAPE screen after the completion of TURNING MACH SHAPE and M MACHINING SHAPE definition. On this screen, the shape of basic blank material is displayed in yellow, and on the basic blank material shape, the shape defined through the shape element data entered on the TURNING MACH SHAPE and M MACHINING SHAPE screen is displayed in light blue.

When the data for STCK RMV H and CORNER R has been entered, processing is started and the blank material shape with uniform stock removal is displayed in blue.

There are rules when providing the blank material with stock removal.

- No stock removal is provided for portions whose shape has been defined by jump or designation of surface roughness.
- Stock removal is provided irrespective of the basic blank material shape.
- The optimal R is provided regarding the entered R value as the maximum value.





After the completion of data entry on the AVE. BLANK SHAPE screen, make sure that the blank material shape is displayed on the screen. Press the function key [F2] (ORDER ↓). The start-up page of the graphic edit screen will be displayed.

When the function key [F1] (ORDER ↑) is pressed at this stage, the TURNING MACH SHAPE screen or M MACHINING SHAPE screen (when the optional multi-machining specification is selected) will be displayed.

Note 1: *To partly correct the shape defined on the AVE. BLANK SHAPE definition process, for example to change the stock removal at certain position, use the BLANK SHAPE screen.*

Note 2: *The shape defined on the AVE. BLANK SHAPE definition process is simulated in yellow on the CRT as the blank material shape in the program TEST mode.*

Note 3: *To correct the finished shape after the completion of finished shape definition and AVE. BLANK SHAPE definition, enter the data for STCK RMV H and CORNER R again.*

3. BLANK SHAPE Definition

The explanation that follows is the procedure to define the arbitrary-shaped blank material.

In the BLANK SHAPE definition process, the procedure to define the detailed shape of arbitrary-shaped blank material (multi-diameter blank material, for example), other than round bar material.

The following functions are provided to define the arbitrary-shaped blank material shape.

(1) START PT (START PT, DEFINE DIR.)

(2) FACE

(3) TAPER

(4) LONG

(5) CCW ARC (optional:contour generating machining specification)

(6) CW ARC (optional:contour generating machining specification)

(7) R-CHF (optional:contour generating machining specification)

(8) CHAMFER

(9) JUMP

(10) Insertion of shape element

(11) Deletion of shape element

(12) Re-display of shape

In the BLANK SHAPE shape definition process, first designate the shape definition starting point and then enter the shape elements (2) to (9) sequentially according to the finished shape. The functions (2) to (9) are assigned to the function keys.

Note: *To define the arbitrary-shaped blank material, enter FREE SHAPE for BLANK SIZE in the BASIC BLANK SHAPE definition process.*

The relationship between the sequence of shape definition and the BLANK SHAPE definition is diagramed below.

Startup of shape definition

⇩

Designate AVERAGE STICK for BLANK SIZE. ------ BASIC BLANK SHAPE definition

⇩

TURNING MACH SHAPE definition

* Optional:
Multi-machining specification,
Contour generating machining specification

⇩

M MACHINING SHAPE definition
SIDE GENERATING definition
FACE GENERATING definition

⇩ [F2] (ORDER ↓)

BLANK SHAPE definition

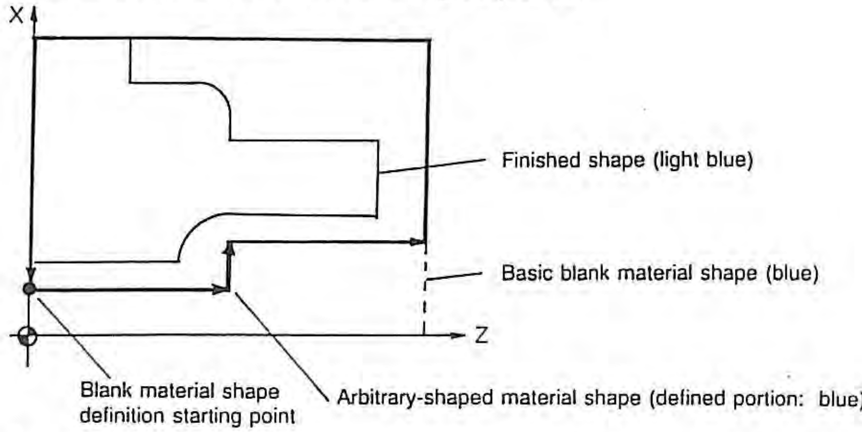
- START PT
- FACE
- TAPER
- LONG
- CCW ARC
- CW ARC
- R-CHF
- CHAMFER
- JUMP
- Insertion of shape element
- Deletion of shape element
- Re-display of shape

After the completion of BLANK SHAPE definition, press the function key [F2] (ORDER ↓). All shape definition processes are terminated. --------------- ⇩ [F2] (ORDER ↓)

Completion of shape definition
To SETUP and process determination

BLANK SHAPE definition is carried out in the following sequence.

Example:



1) Enter FREE SHAPE for BLANK SIZE in the BASIC BLANK SHAPE definition process.

2) After the completion of basic blank material shape definition, press the function key [F2] (ORDER ↓) to advance to the TURNING MACH SHAPE screen.

3) Enter the data for START PT, DEFINE DIR.

4) Select the shape element and define the blank material shape.

   Necessary shape elements: LONG, FACE, LONG, JUMP

   Start blank material shape definition from the starting point and define the blank material shape in such a manner that the starting point and the end point will agree each other.

   Definition with a single stroke

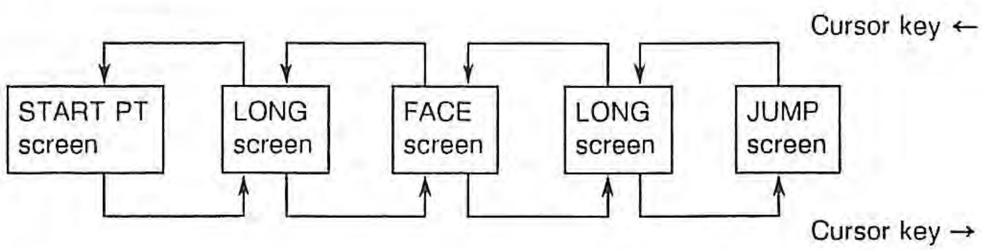
5) After the completion of blank material shape definition, press the function key [F2] (ORDER ↓).

This completes all shape definition processes.

The one-touch IGF system will check the entered data, and an error will occur if there is faulty data.

Note 1: *Using the cursor control keys ← and →, it is possible to display the shape element designation screen for the turning shape which has been already defined. Change of the set data is also possible.*

For the example on the previous page, the following screens can be displayed sequentially.



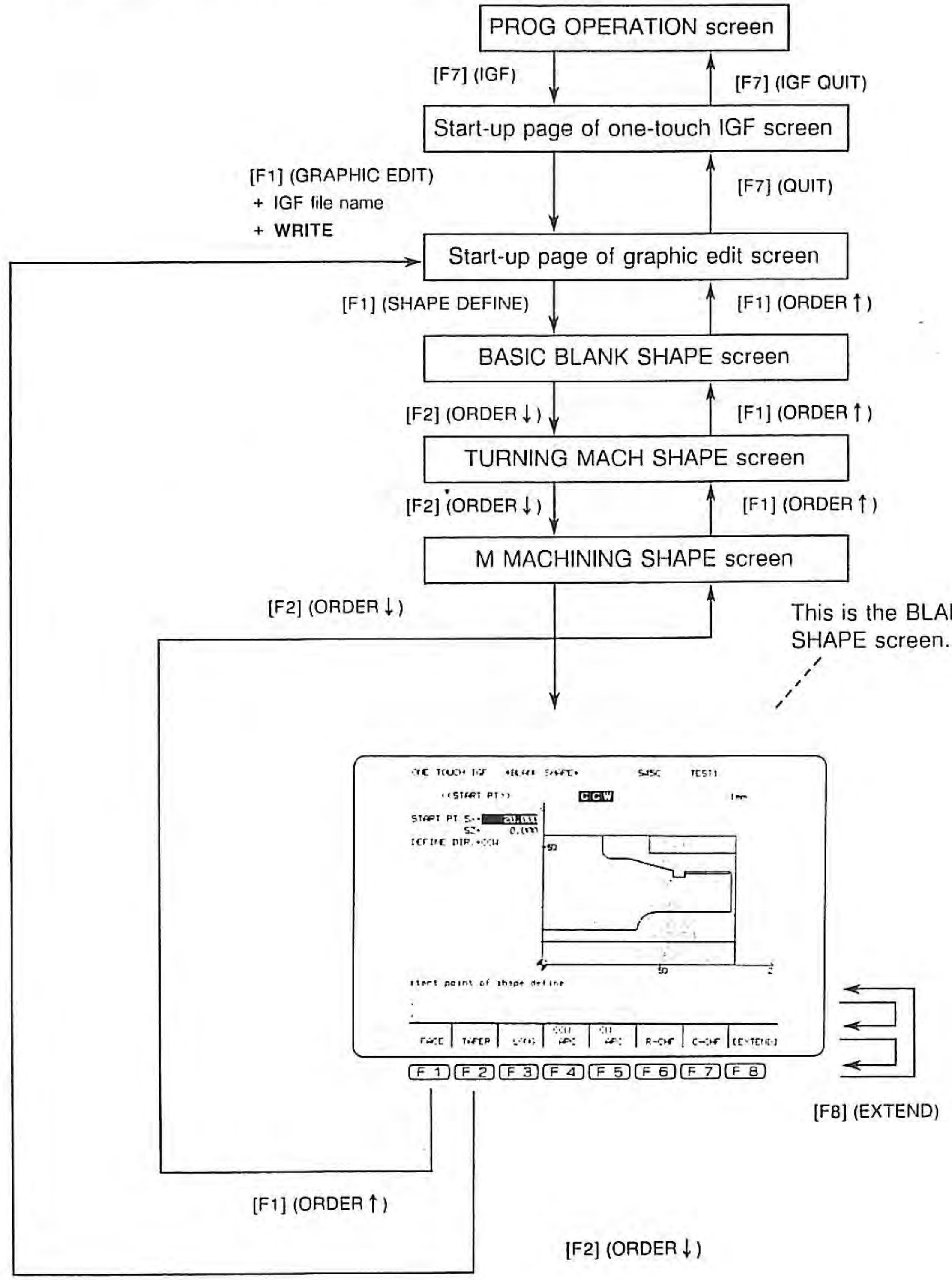
The (magenta) marker indicates where the finished shape (blue) is defined on the displayed shape element designation screen. Use the cursor control keys to move the marker.

Note 2: *The blank material shape defined in the BLANK SHAPE definition process is simulated in yellow on the CRT in the program TEST mode.*

Screen Transfer:

The BLANK SHAPE screen is called from the PROG OPERATION screen as diagramed below.

Optional specification to be selected: Multi-machining specification



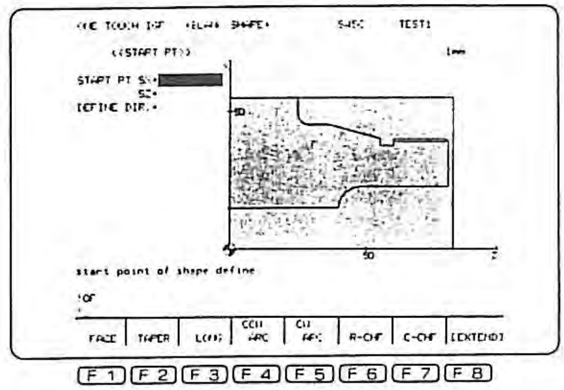
(1) **Designation of Starting Point (START PT)**

a) Function

START PT and DEFINE DIR. for BLANK SHAPE is designated.

b) Data to be Designated

START PT SX
SZ
DEFINE DIR.



c) How to Designate

Enter FREE SHAPE for BLANK SIZE in the BASIC BLANK SHAPE definition process.

Press the function key [F2] (ORDER ↓) after the completion of finished shape definition. The START PT screen for BLANK SHAPE definition will be displayed. Enter the data as follows.

START PT SX, SZ

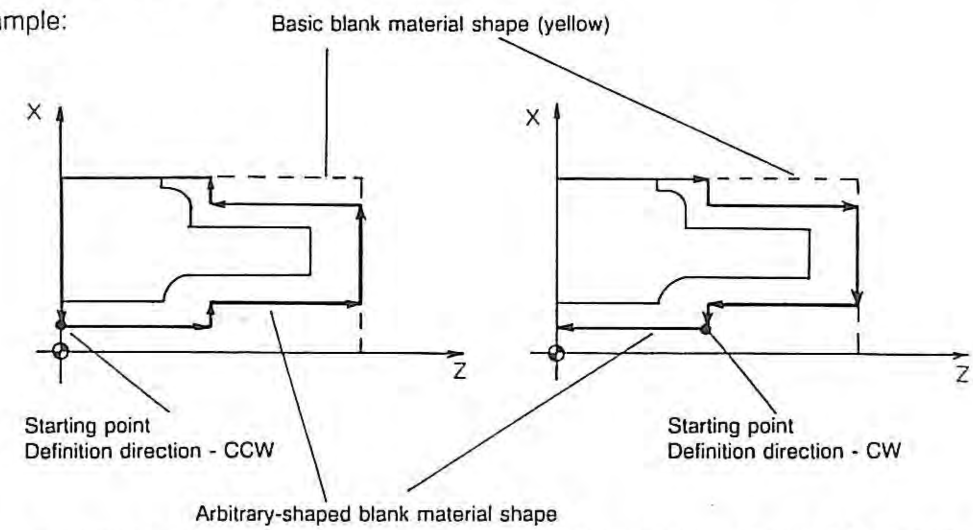
The X- (SX-) and Z- (SZ-) coordinates of the blank material shape starting point for BLANK SHAPE are designated.

The shape definition starting point may be located at any place on the circumference and inside of the basic blank material shape.

DEFINE DIR.

Shape definition direction for BLANK SHAPE is designated. As the prompt "1 = CCW, 2 = CW" is displayed, enter "1" or "2" accordingly.

Example:



In the arbitrary-shaped blank material shape definition process, shape definition is started from the shape definition starting point. Therefore, designate the shape definition starting point (SX, SZ) and definition direction. Otherwise the process does not proceed.

(2) **FACE**

a) Function

The shape of blank material end face is designated.

b) Data to be Designated

END PT X
Z
NEXT TAN TN

c) How to Designate

Press the function key [F8] (EXTEND) until the function "FACE" is assigned to the function key [F1] .

Press the function key [F1] (FACE). The FACE screen for BLANK SHAPE will be displayed.

The data setting procedure is the same as that for end face in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (2) "FACE" on page 88.

d) Conditions for Shape Determination

Refer to the explanation in Section 6, 3. (2) "FACE" on page 88.

(3) **TAPER**

a) Function

The shape of taper of the blank material is designated.

b) Data to be Designated

END PT X
Z
NEXT TAN TN
Z ANGLE A

c) How to Designate

Press the function key [F8] (EXTEND) until the function "TAPER" is assigned to the function key [F2] .

Press the function key [F2] (TAPER). The TAPER screen will be displayed.

The data setting procedure is the same as that for taper in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (3) "TAPER" on page 91.

d) Conditions for Shape Determination

Refer to Section 6, 3. (3) "TAPER" on page 91.

(4) **LONG**

a) Function

The shape in the longitudinal direction of the blank material is designated.

b) Data to be Designated

END PT	X
	Z
NEXT TAN	TN

c) How to Designate

Press the function key [F8] (EXTEND) until the function "LONG" is assigned to the function key [F3].

Press the function key [F3] (LONG). The LONG screen will be displayed.

The data setting procedure is the same as that for longitudinal portion in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (4) "LONG" on page 95.

d) Conditions for Shape Determination

Refer to the explanation in Section 6, 3. (4) "LONG" on page 95.

(5) **CCW ARC**

(6) **CW ARC**

a) Function

The shape of CCW arc or CW arc of the blank material is designated.

b) Data to be Designated

END PT	X
	Z
NEXT TAN	TN
RADIUS	R
CENTER	CX
	CZ

c) How to Designate

Press the function key [F8] (EXTEND) until the function "CCW ARC" is assigned to the function key [F4] and "CW ARC" to [F5].

Press the function key [F4] (CCW ARC). The CCW ARC screen will be displayed.

Press the function key [F5] (CW ARC). The CW ARC screen will be displayed.

The data setting procedure is the same as that for arc in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (5) "CCW ARC" or (6) "CW ARC" on page 97.

d) Conditions for Shape Determination

Refer to Section 6, 3. (5) "CCW ARC" or (6) "CW ARC" on page 97.

(7) R-CHF

a) Function

The shape of rounding R of the blank material is designated.

b) Data to be Designated

RADIUS R

c) How to Designate

Press the function key [F8] (EXTEND) until the function "R-CHF" is assigned to the function key [F6].

Press the function key [F6] (R-CHF). The R-CHF screen will be displayed.

The data setting procedure is the same as that for rounding R in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (7) "R-CHF" on page 101.

d) Conditions for Shape Determination

Refer to Section 6, 3. (7) "R-CHF" on page 101.

Note: *The following shapes cannot be designated preceding or following rounding R.*

Jump J

Rounding R

Chamfer C

Chamfer C cannot be designated as the first shape definition starting point or the last shape preceding definition end point.

(8) CHAMFER

a) Function

The shape of chamfer of the blank material is designated.

b) Data to be Designated

CHF SIZE C

c) How to Designate

Press the function key [F8] (EXTEND) until the function "C-CHF" is assigned to the function key [F7].

Press the function key [F7] (C-CHF). The C-CHF screen will be displayed.

The data setting procedure is the same as that for chamfer in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (8) "CHAMFER" on page 103.

d) Conditions for Shape Determination

Refer to Section 6, 3. (8) "CHAMFER" on page 103.

Note: *The following shapes cannot be designated preceding or following chamfer C.*

Jump J

Rounding R

Chamfer C

Rounding R cannot be designated as the first shape the shape definition starting point or the last shape preceding the shape definition end point.

(9) **JUMP**

a) Function

The portion whose shape is identical to the basic blank material shape is designated.

b) Data to be Designated

END PT X
Z

c) How to Designate

Press the function key [F8] (EXTEND) until the function "J JUMP" is assigned to the function key [F4].

Press the function key [F4] (J JUMP). The JUMP screen will be displayed.

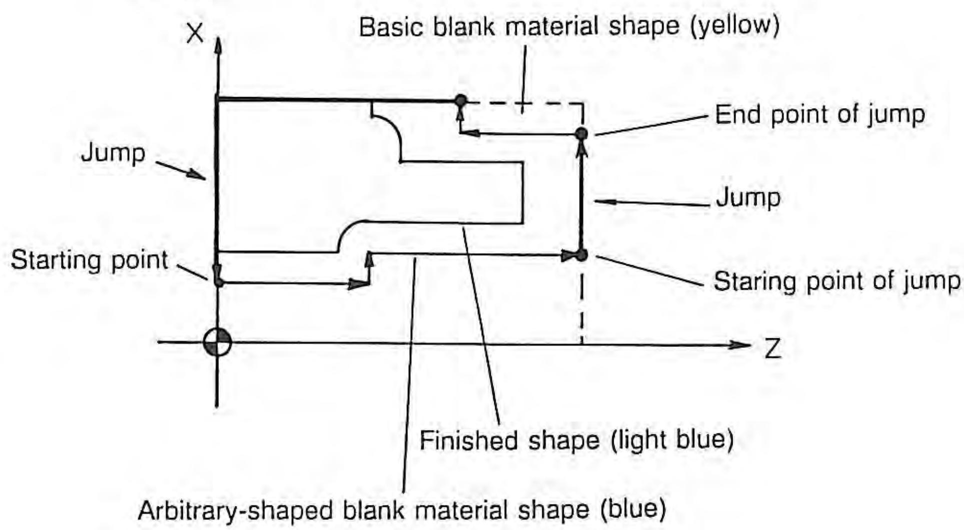
The data setting procedure is the same as that for jump in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (12) "JUMP" on page 115.

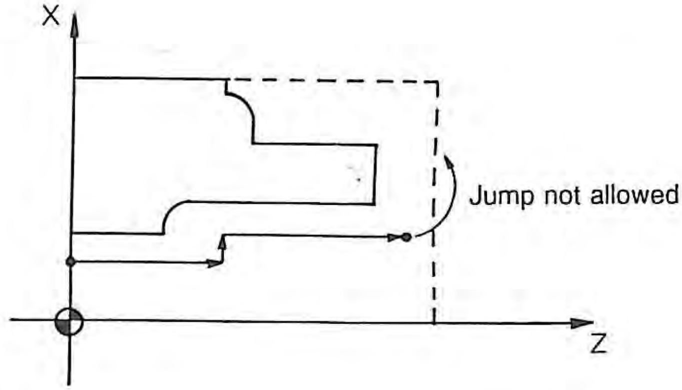
d) Conditions for Shape Determination

Refer to Section 6, 3. (12) "JUMP" on page 115.

Note: *The shape of jump can only be designated on the contour of basic blank material shape. (on the contour of the shape displayed in yellow on the screen.) In other words, the starting and end points of jump must be located on the contour of basic blank material shape.*

Example:





(10) **Insertion of Shape Element**

a) Function

Shape element is inserted between designated two shape elements.

b) How to Designate

Shape element is inserted in the same manner as in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (14) "Insertion of Shape Element" on page 118.

(11) **Deletion of Shape Element**

a) Function

The designated shape element is deleted.

b) How to Designate

Shape element is deleted in the same manner as in TURNING MACH SHAPE definition. Refer to the explanation in Section 6, 3. (15) "Deletion of Shape Element" on page 120.

(12) **Re-display of Shape**

a) Function

The shape defined by entered data is re-displayed.

b) How to Designate

Press the function key [F8] (EXTEND) until the function "SHAPE RE-DIS." is assigned to the function key [F5].

Press the function key [F5] (SHAPE RE-DIS.).

The shape defined by the entered data will be re-displayed. Refer to the explanation in Section 6, 3. (16) "Re-display of shape" on page 121.

(13) **Termination of BLANK SHAPE Definition**

When BLANK SHAPE definition beginning from the starting point in such a manner as single-stroke drawing has been completed, press the function key [F2] (ORDER ↓). The BLANK SHAPE definition process is terminated. The CRT will return to the start-up page of the graphic edit screen.

Remark: The end point of the last shape element is identical to the shape definition starting point. Therefore, it is possible to enter "E" instead of inputting the end point coordinate. Proceed as follows.

i) Display the shape element designation screen of the last shape element.

ii) Move the cursor to END PT X = (or END PT Z =).

iii) Enter "E" and press the **WRITE** key.

(14) **Shape Referencing Function**

Refer to Section 6, 3. (18) "Shape Referencing Function" on page 122.

SECTION 8 SETUP INPUT

1. What is the Function of "Setup Input"?

This is the function to designate the availability of chuck and tailstock center, and dimensions of chuck and tailstock center so that the shape of chuck or tailstock center can be displayed on the screen. This function is also used to set the data to output the bar feeder (puller) controlling M codes with the bar feeder (puller) specification models.

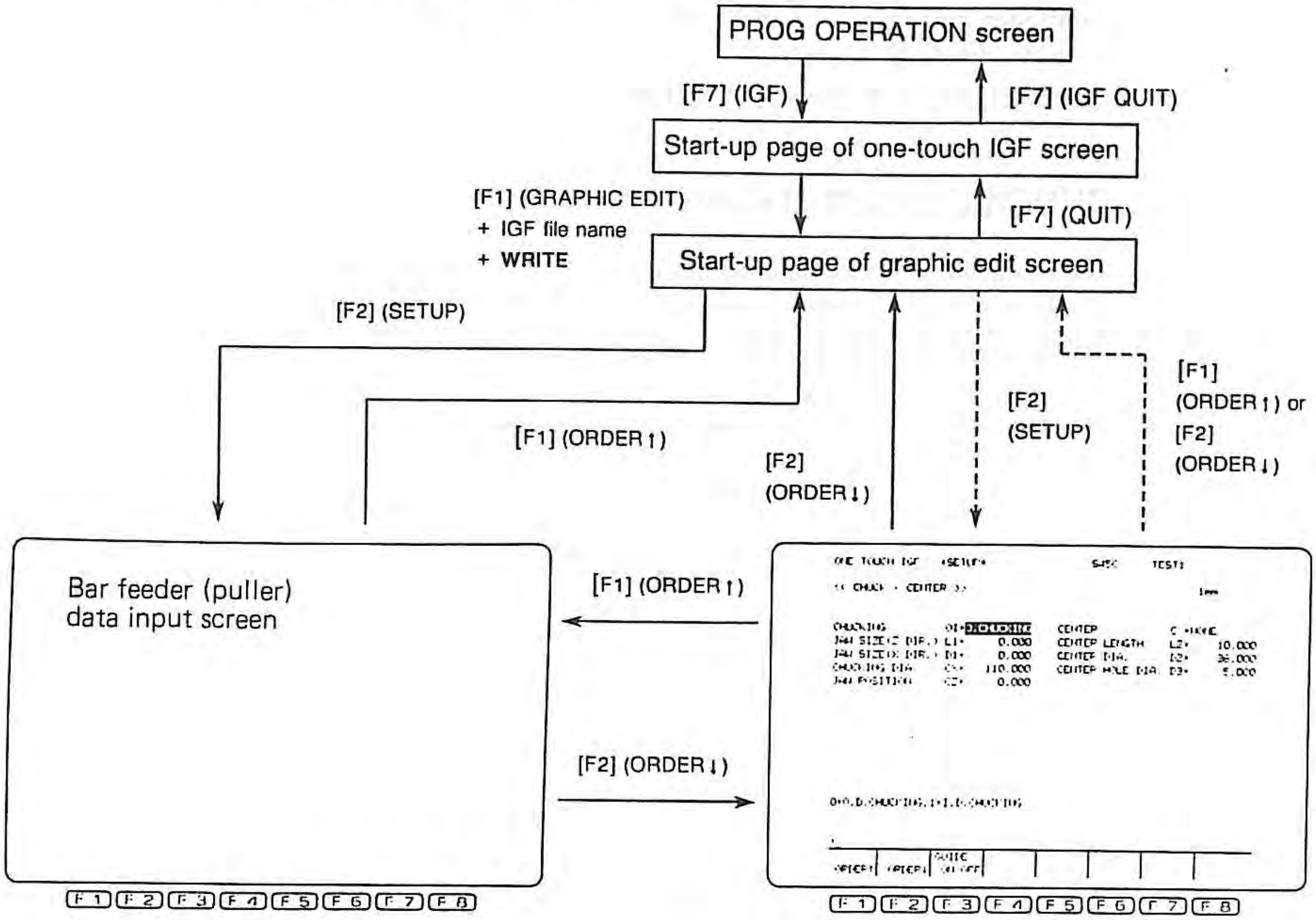
Note: *Before entering the data to output the bar feeder (puller) controlling M codes, set the data for the bar material preparation parameter.*

Refer to Section 19, "Bar Feeder and Bar Puller Function".

Screen Transfer:

The CHUCK·CENTER screen is called from the PROG OPERATION screen as diagramed below.

Optional specification to be selected: Multi-machining specification



This is the bar feeder (puller) data input screen. (Three screens are available according to the data set at the bar material preparation parameter.)

This is the CHUCK·CENTER screen.

Note 1: *The screen transfer varies depending on the setting of parameter:*

⟶ Bar material preparation parameter No. 1 BAR FEEDER/PULLER OPERATION PATTERN is set at "1", "2", "3", or "4", and No. 2 BAR FEEDER/PULLER PROGRAM (0 = OFF, 1 = ON) is set at "1"

----> Either bar material preparation parameter No. 1 BAR FEEDER/PULLER OPERATION PATTERN is set at "0", or No. 2 BAR FEEDER/PULLER PROGRAM (0 = OFF,1 = ON) is set at "0", or the bar feeder (puller) specification is not selected

Note 2: *When the function key [F1] (ORDER↑) or [F2] (ORDER↓) is pressed, the entered data is checked and the screen will be changed.*

2. Designating Bar Material Preparation (bar feeder (puller) specification)

The program for controlling bar feeder (puller) can be made by setting the data for bar material preparation parameter. To make this program, it is necessary to set the data of bar feeder (bar puller) on the SETUP screen.

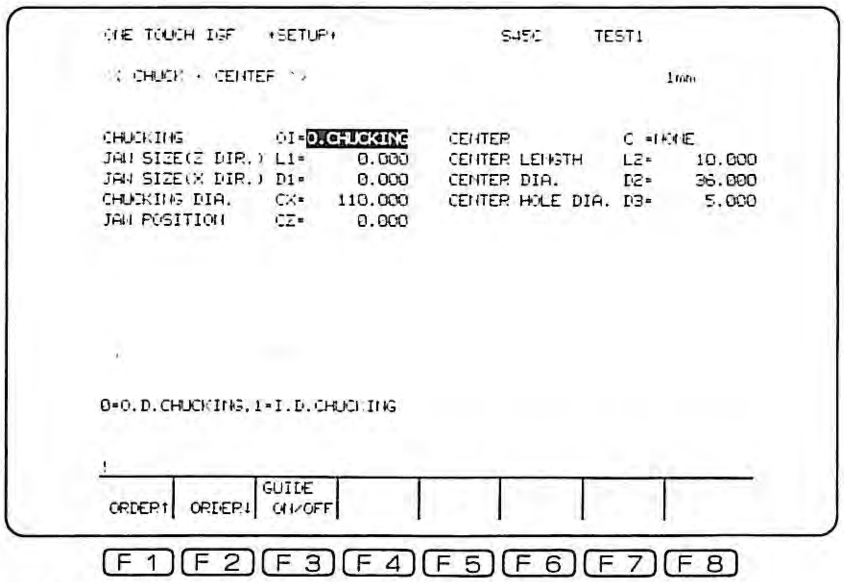
For details, refer to Section 20, "Bar Feeder and Bar Puller Function".

3. CHUCK·CENTER Definition

The availability of chuck and tailstock center, and dimensions of chuck and tailstock center so that the shape of chuck or tailstock center can be displayed on the screen.

Screen to be displayed and data to be set are indicated below.

(1) CHUCK·CENTER Screen



CHUCKING OI

Enter "0" for OD gripping, and "1" for ID gripping.

As "0" is set as the default, no data input is required in the case of OD gripping.

JAW SIZE L1

JAW SIZE D1

CHUCKING DIA. CX

For CHUCKING DIA. CX, the same value as set for OUT DIA. OD on the BASIC BLANK SHAPE screen is initially set. This value can however be used only in the case of OD gripping. In the case of ID gripping, change the value.

JAW POSITION CZ

For JAW POSITION CZ, the value calculated on the basis of PZ SHIFT ZO designated on the BASIC BLANK SHAPE screen is initially set.

The relationship between JAW POSITION CZ and PZ SHIFT is as indicated below.

JAW POSITION CZ = – (PZ SHIFT ZO)

CENTER C

The availability of tailstock center is designated. Enter "0" when tailstock center is not used, and "0" when tailstock center is used. As "0" is set as the default, no data input is required when tailstock center is not used.

CENTER LENGTH L2

The value set at dimension parameter No. 9 CENTER LENGTH L2 is initially set.

CENTER DIA. D2

The value set at dimension parameter No. 10 CENTER DIA. D2 is initially set.

CENTER HOLE DIA. D3

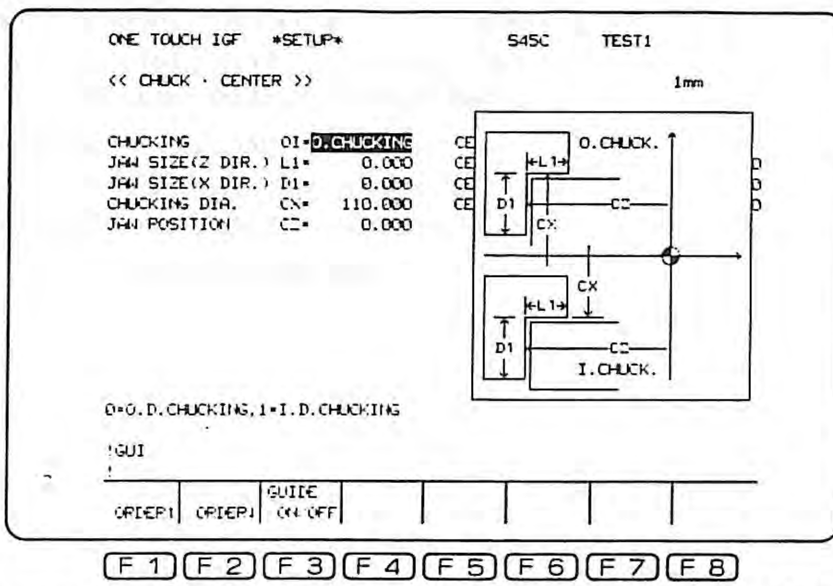
The value set at dimension parameter No. 11 CENTER HOLE DIA. D3 is initially set.

Remark: Guide Drawing Display

When designating the shape elements of chuck/tailstock center, it is possible to display the guide drawing for data input.

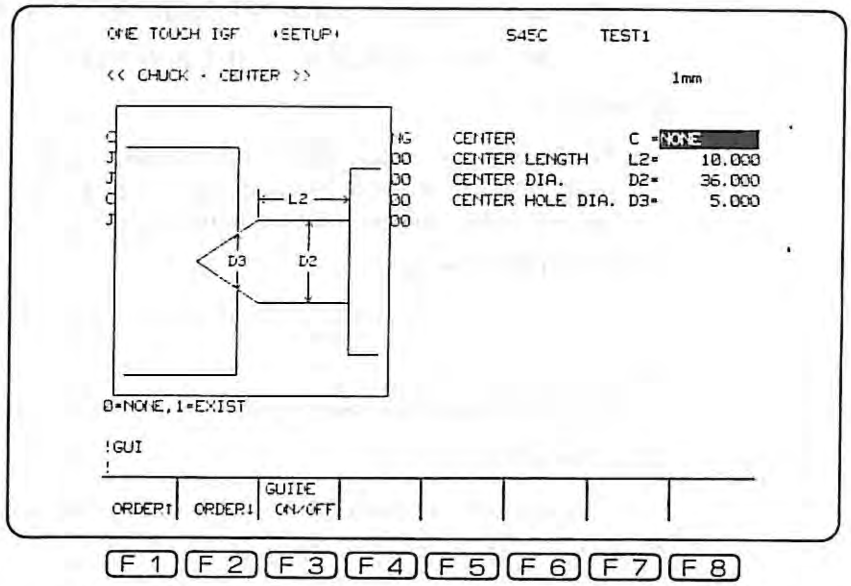
Which of the input data corresponds to what part of the chuck/tailstock center is displayed in this drawing.

Press the function key [F3] (GUIDE ON/OFF) while the cursor is located at any data column of the chuck shape data. The guide drawing for chuck will be displayed.



To erase the guide drawing, press the function key [F3] (GUIDE ON/OFF) again or advance the cursor to the data column of chuck data.

Press the function key [F3] (GUIDE ON/OFF) while the cursor is placed at any data column of the tailstock center shape data. The guide drawing for tailstock center will be displayed.



To erase the guide drawing, press the function key [F3] (GUIDE ON/OFF) again or advance the cursor to any of the data column.

After the completion data setting, press the function key [F2] (ORDER ↓). The one-touch IGF system will check the entered data and the start-up page of the one-touch IGF will appear when the data has been correctly entered. An error will occur if there is faulty data.

Note: *The screen called by pressing the function key [F1] (ORDER ↑) varies depending on the setting of the bar material preparation parameter.*

1 BAR FEEDER / PULLER OPERATION PATTERN . . . . 1, 2, 3, or 4
2 BAR FEEDER / PULLER PROGRAM (0 = OFF, 1 = ON) 1

→ BAR MATERIAL PREPARATION PARAMETER screen

1 BAR FEEDER / PULLER OPERATION PATTERN . . . . 0 or
2 BAR FEEDER / PULLER PROGRAM (0 = OFF, 1 = ON) 0

→ Start-up page of one-touch IGF

SECTION 9 AUTOMATIC PROCESS DETERMINATION

1. What is the Function of "Automatic Process Determination"?

The one-touch IGF system automatically determines the area, method, and sequence of machining through assigning the blank material data and finished shape data defined in the shape definition process to various rules for automatic determination as indicated below.

Automatic Determination of Machining Process

- **Determination of machining area**

  Divides the area to be machined into several portions according to the blank material shape and finished material shape, and determines machining process for each machining area.

  - **Finished shape data**

    Indicates the shape at the completion of each machining process.

  - **Blank material shape data**

    Used for rough machining. The area surrounded by contours of blank material shape and finished shape is the area to be machined.

- **Determination of machining method**

  Determines machining method for each machining process.

  - **Machining kind**

    Indicates how to machine the area to be machined.

  - **Tools to be used**

    Indicates tools to be used.

  - **Cutting conditions**

    Indicates cutting conditions such as cutting feedrate, rapid feedrate, or infeed amount.

- **Determination of machining sequence**

  Determines machining method for each machining process.

Fig. 9-1

Supplement:

The automatic process determination function determines the machining process from the data such as input engineering drawing, standard hole data, material data, one-touch IGF parameter, and tool data. Therefore, the possible machining process patterns is infinite and we can not guess the machining process determined by the one-touch IGF for individual engineering drawings. If the machining process determined by the one-touch IGF is not satisfactory, either retry automatic process determination after changing the conditions, or correct the machining process data using the process edit function.

Reference:

- Changing conditions for automatic process determination
- Changing conditions for cutting tool selection

  → Section 9, 4. Role of the Condition Setting Function

- Modifying the machined shape, cutting tool, or cutting conditions
- Modifying the type of machining or sequence of machining

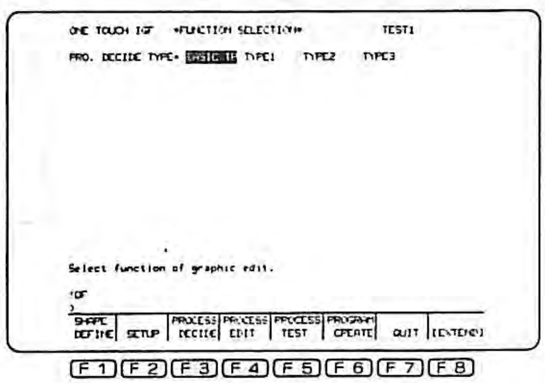
  → Section 11 Process Edit

* The conditions for automatic process determination can also be changed by modifying the standard hole data, material data, one-touch IGF parameter, or tool data.

Screen Transfer:

Start-up page of one-touch IGF screen

[F1] (GRAPHIC EDIT) + IGF file name + WRITE ↓ ↑ [F7] (QUIT)



(Start-up page of graphic edit screen)

Press the function key [F3] (PROCESS DECIDE) on the start-up page of one-touch IGF. This automatically detërmines the machining process.

After the completion of process determination, the prompt ""PROCESS DECIDE" finished" will be displayed on the screen.

Reference:

Designating the BASIC TP, TYPE 1, TYPE 2, and TYPE 3

→ Section 9, 6. Automatic Process Determination Using the Pattern

Note 1: *When process determination is attempted for the following IGF files, the machining processes previously created are all deleted:*

IGF file whose machining process is already determined
IGF file whose machining process is corrected in the process edit mode

For example, when process determination is attempted again for the IGF file whose data is corrected in the process edit mode after the completion of process determination, the corrected data will be all deleted.

When the function key [F3] (PROCESS DECIDE) is pressed for the IGF file whose machining process is already determined, the following message will be displayed on the screen:

process exist (Y = delet, N = decide cancel)

To delete the machining process previously determined and to newly determine the machining process, key in [Y] and press the **WRITE** *key.*

To cancel automatic process determination, key in [N] and press the **WRITE** *key.*

Note 2: *In the case of the optional multi-machining specification, the machining processes previously created are all deleted when process determination is attempted for the following IGF files:*

IGF file whose machining process is already determined
IGF file whose machining process is corrected in the process edit mode

When the function key [F3] (PROCESS DECIDE) is pressed for the IGF files indicated above, the following message will be displayed on the screen:

process exist (Y = delete, M = M append, N = decide cancel)

To delete the machining process previously determined and to newly determine the machining process, key in [Y] and press the **WRITE** *key.*

To determine only the multi-machining process, key in [M] and press the **WRITE** *key.*

To cancel automatic process determination, key in [N] and press the **WRITE** *key.*

When [M] is keyed in and the **WRITE** *key is pressed, the machining process is automatically determined only for the multi-machining portion of the finished shape which has been determined in shape definition. The multi-machining process created here is added at the end of machining process already created. Therefore, to correct/amend the multi-machining shape or to add a new multi-machining process, first delete the unnecessary process in the process edit mode, correct/amend the multi-machining shape in shape definition, and carry out process determination.*

2. Determination of Machining Area

Machining area for each machining process while dividing the area to be machined into several portions on the basis of blank material shape and finished material shape.

The area to be machined is divided into several portions according to the blank material shape and finished material shape, while taking the method and sequence of machining into consideration.

When ∇∇, or ∇∇∇∇ is designated for surface roughness, the finished shape in rough machining can be obtained through adding finishing stock (0.2 mm: can be set by material data).

For rough machining, both the blank material shape and the finished shape must be designated as the area surrounded by contours of blank material shape and finished shape is machined.

For finish machining, determination of blank material shape is not necessary as only finishing stock is machined and the tool therefore moves along the finished shape.

Fig. 9-2 indicates the cases where ∇ is designated for surface roughness and ∇∇ is designated for surface roughness.

In the case of ∇, rough OD← process is generated to obtain the finished shape defined in the shape definition process as no finish machining process is necessary. In the case of ∇∇, two machining processes, rough OD← and finish OD← are generated.

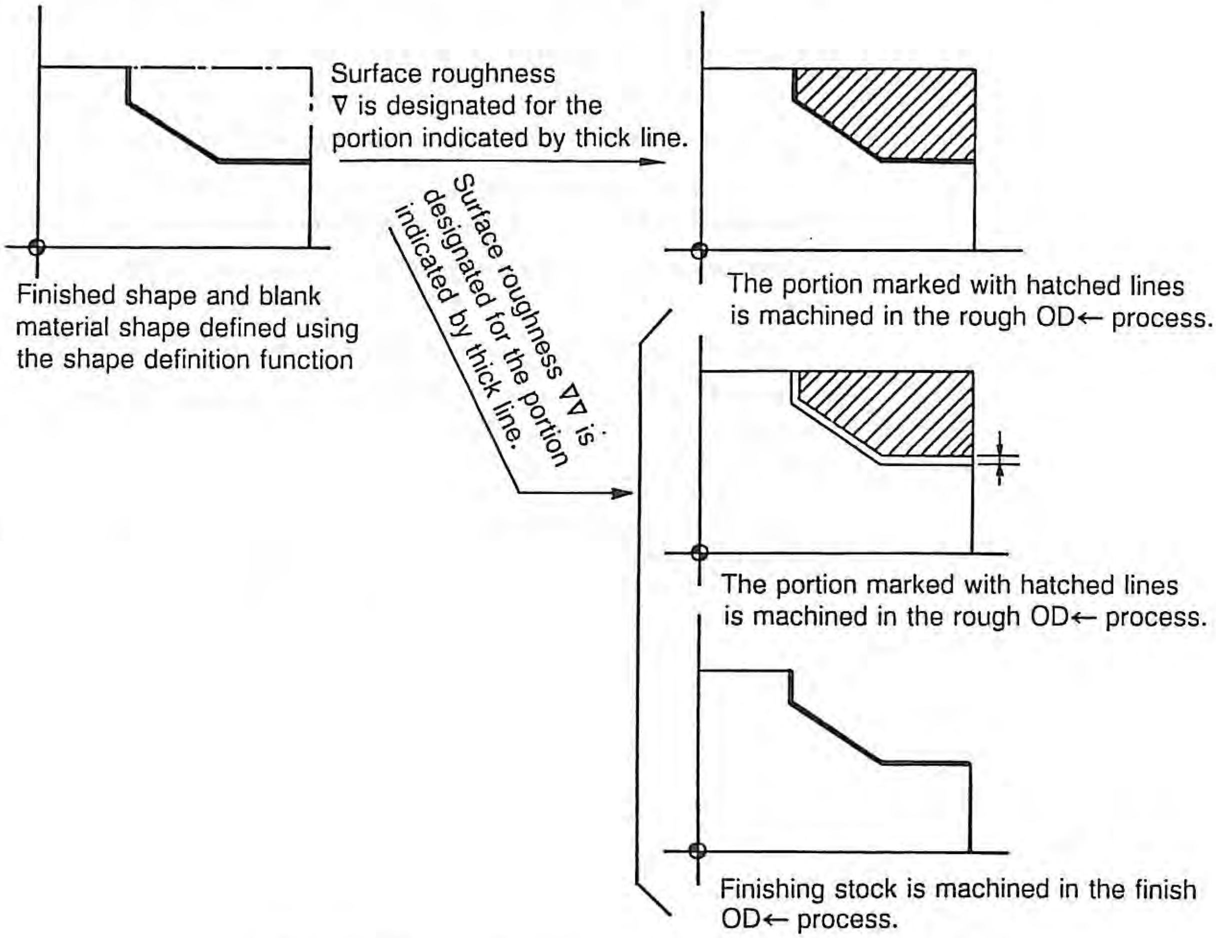


Fig. 9-2 Surface Roughness Designation and Finishing Stock

3. Determination of Machining Method

How to machine the automatically determined machining area is determined. The machining method is determined by the following three items.

Kind of machining
Tools to be used
Cutting conditions

(1) Machining Kind

"Machining kind" refers to how to machine the area to be machined. Refer to Table 9-1. For example, when "ROUGH OD←" refers to the machining explained below.

1) ROUGH : round bar turning cycle of LAP

2) OD : outside diameter

3) ← : from right to left (in the Z-axis negative direction)

Fig. 9-3 shows the machining cycle for ROUGH OD←.

In the same manner, machining cycle for FINISH OD← is as indicated in Fig. 9-4.

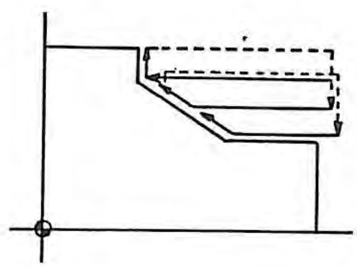


Fig. 9-3 ROUGH OD←

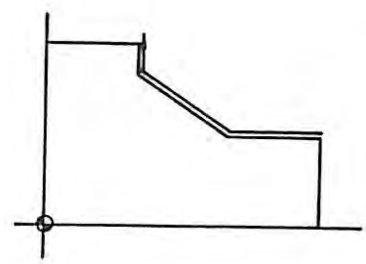
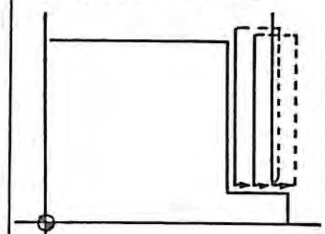
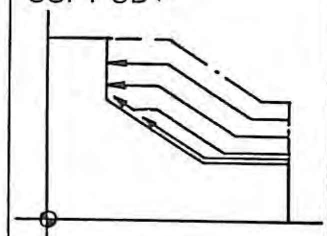
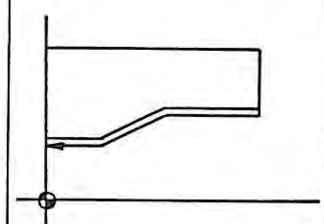
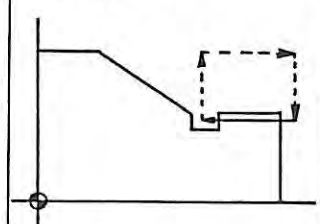
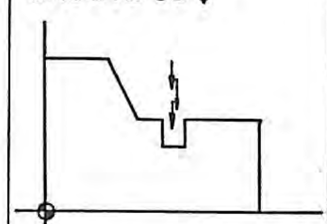
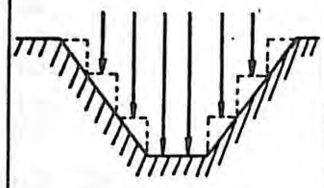
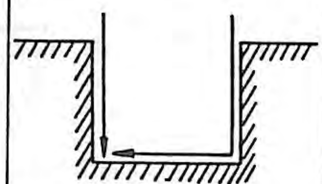
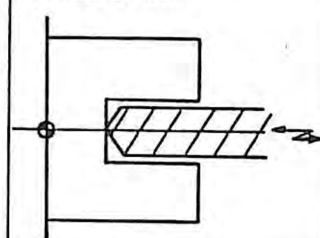
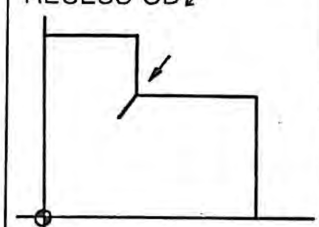
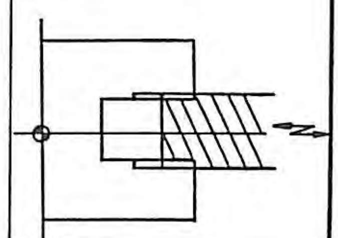
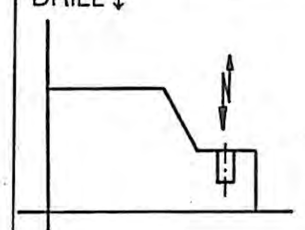


Fig. 9-4 FINISH OD←

Table 9-1

Machining Kind	Characteristics of Machining	Motion
ROUGH OD←, ROUGH ID←, ROUGH O. FACE↓, ROUGH I. FACE↑, ROUGH M. OD←, ROUGH M. ID←, ROUGH M. FACE↓, ROUGH M. OD→, ROUGH M. ID→, ROUGH M. FACE↑, ROUGH OD→, ROUGH ID→	Machining in the round bar turning cycle of LAP	ROUGH O. FACE↓ 
COPY OD←, COPY ID←, COPY O. FACE↓, COPY I. FACE↑, COPY M. OD←, COPY M. ID←, COPY M. FACE↓, COPY M. OD→, COPY M. ID→, COPY M. FACE↑, COPY OD→, COPY ID→	Machining in the copy turning cycle of LAP	COPY OD← 
FIN. OD←, FIN. ID←, FIN. O. FACE↓, FIN. I. FACE↑, FIN. M. OD←, FIN. M. ID←, FIN. M. FACE↓, FIN M. OD→, FIN. M. ID→, FIN. M. FACE↑, FIN. OD→, FIN. ID→	Machining along the finished shape defined in shape definition	FIN. ID← 
THREAD OD←, THREAD ID←, THREAD FACE↓, THREAD OD→, THREAD ID→, THREAD FACE↑	Machining in the thread cutting cycle	THREAD OD← 
GROOVE OD↓, GROOVE ID↑, GROOVE FACE←	Machining in the single grooving cycle	GROOVE OD↓ 

Machining Kind	Characteristics of Machining	Motion
R. W. GRV OD↓, R. W. GRV ID↑, R. W. GRV FC←, R. V GRV OD↓, R. V GRV ID↑, R. V GRV FC←	The tool moves along the groove shape and finishes wide groove.	R. V GRV OD↓ 
F. W. GRV OD↓, F. W. GRV ID↑, F. W. GRV FC←, F. V GRV OD↓, F. V GRV ID↑, F. V GRV FC←	The tool moves along the groove shape and finishes wide groove.	F. W.GRV OD↓ 
DRILL BLIND (HSS, CRBD) DRILL THRU (HSS, CRBD) DRILL CENTR	Machining in the drilling cycle	DRILL BLIND 
RECESS OD↙, RECESS ID↖, RECESS OD↘, RECESS ID↗	Recessing	RECESS OD↙ 
R. ENDMILL F. ENDMILL TAP	Endmilling or tapping at workpiece center	TAP 
Multiple machining DRILL↓, BORING↓, TAP↓, DRILL←, BORING←, TAP←, REAMING↓, REAMING←, FACE KEY, SIDE KEY, ENDMILL↓, ENDMILL←, FACE GEN. R, FACE GEN. F, SIDE GEN. R, SIDE GEN. F	Multiple machining	DRILL↓ 

(2) Tools to be Used

Tools to be used to machine the defined machining area are determined.

Cutting tools to be used are determined according to the area and kind of machining. As far as necessary tools are registered in the tool data, tools are selected from the tool data. If necessary tools are not registered, the data of such tools are automatically generated as non-registered tool data. Here, the tool generated as non-registered tool is a generally used standard tool. Therefore, to machine the part using the part program created by the one-touch IGF, mount the same tool as generated as non-registered tool to the designated turret position.

(3) Cutting Conditions

Cutting conditions to be applied for machining the defined area are determined.

Cutting conditions include items such as cutting feedrate, infeed amount, etc., and they are determined according to the kind of blank material and machining method. The values for these items are registered as material data, and desired cutting conditions can be determined by changing the setting of material data.

4. Role of the Condition Setting Function

With the one-touch IGF, the machining process is automatically determined from the following data:

Finished workpiece shape data, blank material shape data, standard hole data, tool data, material data, one-touch IGF parameter data

With the condition setting function. automatic process determination can be executed in multiple patterns by partly changing the tool data and the one-touch IGF parameter.

4-1. Functions

(1) Prior Tool Designation

From the tools registered in the tool data, a tool which is used for the intended machining prior to other tools can be designated.

(2) Process Determination Condition Change

Conditions to determine the machining process can be changed by changing the face/long deciding data, inside machining data, or shape/tool position data.

(3) Data Setting for Each Process Determination Pattern

The data set for (1) and (2) above can be registered in four patterns (BASIC TP, TYPE1, TYPE2, TYPE3). Register the data according to usage, and designate the pattern as needed. Automatic process determination can be executed. (condition selection)

Supplement:

Before registering the tool as a prior tool, store the tool data of this tool to the one-touch IGF memory using the tool data setting function.

In data setting using the automatic process determination pattern, when the data for BASIC is changed, the corresponding one-touch IGF parameter data is also changed. In the same manner, when the one-touch IGF parameter data is changed, the data for BASIC is also changed.

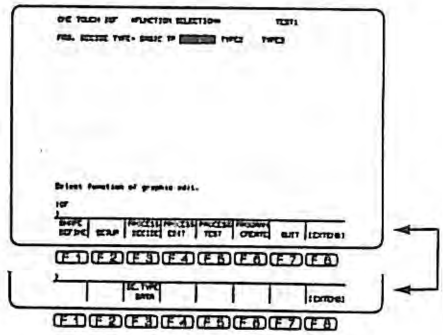
Screen Transfer:

The DE.DEC.TYPE DATA screen is called from the one-touch IGF screen as diagramed below.

Start-up page of one-touch IGF screen

[F1] (GRAPHIC EDIT)
+ IGF file name
+ **WRITE**

[F7] (IGF QUIT)



[F8]
(EXTEND)

[F3] (PROCESS DECIDE)

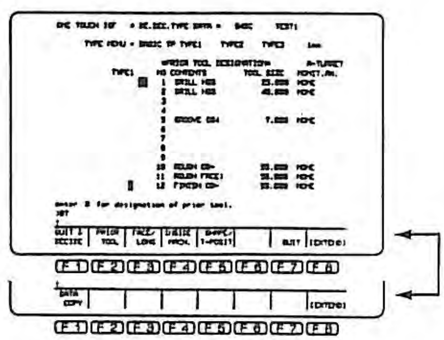
The automatic process determination pattern can be selected using the cursor control keys ← and →.

[F3] (DE.TYPE DATA)

[F3]
(DE.TYPE DATA)

[F7] (QUIT)

[F1] (QUIT & DECIDE)



[F8]
(EXTEND)

[F2] (PRIOR TOOL)
[F3] (FACE/LONG)
[F4] (INSIDE MACH.)
[F5] (SHAPE/T-POSIT)

Press the function to display the desired condition setting screen.

[F1] (QUIT & DECIDE)

Ends condition setting operation and determines the machining process automatically.

[F7] (QUIT)

Ends condition setting operation.

[F1] (DATA COPY)

Switch the screen using the cursor control keys ← and → and set data for each individual process determination pattern.

[F2] (DESIG. GUIDE))

To display the PRIOR TOOL DESIGNATION screen, FACE/LONG DECIDING DATA screen, INSIDE MACHINING DATA screen, or SHAPE/TOOL POSITION DATA screen, press the function key [F3] (DE.TYPE DATA).

5. Data Setting Item

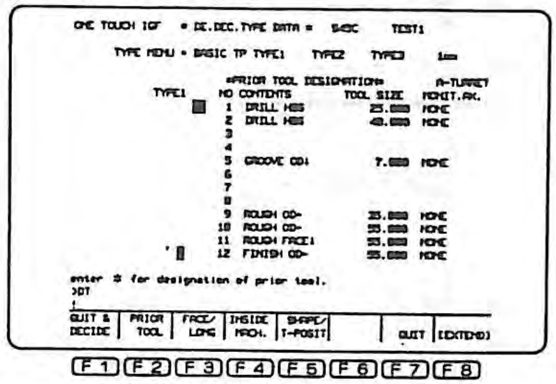
(1) PRIOR TOOL DESIGNATION

The tool which is used prior to other tools (called "prior tool") can be designated from the tools registered in the tool data. Input "*" preceding the tool number.

Operating Procedure:

1) Press the function key [F8] (EXTEND) on the start-up page of the graphic edit screen. Then, press the function key [F3] (DE. TYPE DATA).

2) Press the function key [F2] (PRIOR TOOL).

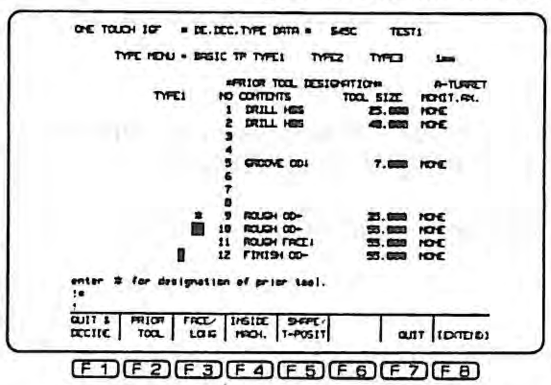


3) Using the cursor control keys, move the cursor to the tool to be registered as the prior tool. Key-in "*" and press the **WRITE** key.

To cancel prior tool designation, key-in "N" and press the **WRITE** key.

* Data for four patterns of automatic process determination can be set. Switch the screen using the cursor control keys ← and → and input the data as needed.

* When the function key [F3] (FACE/LONG), [F4] (INSIDE MACH.) or [F5] (SHAPE/T-POSIT) is pressed, the screen for setting other data will be displayed.



(PRIOR TOOL DESIGNATION screen)

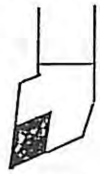
4) Press the function key [F1] (QUIT & DECIDE) or [F7] (QUIT).

Example:

Automatic process determination using the PRIOR TOOL DESIGNATION function

- Tool registered in the tool data

No. 1 ROUGH OD ←



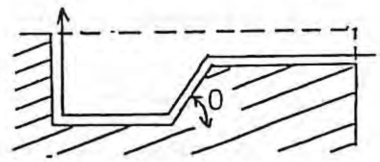
Tool angle = 55°
Cutting edge angle = 3°

No. 2 ROUGH OD ←

Tool angle = 35°
Cutting edge angle = 3°

* Assume that the data of other tools is not registered.

- Input the machining shape.



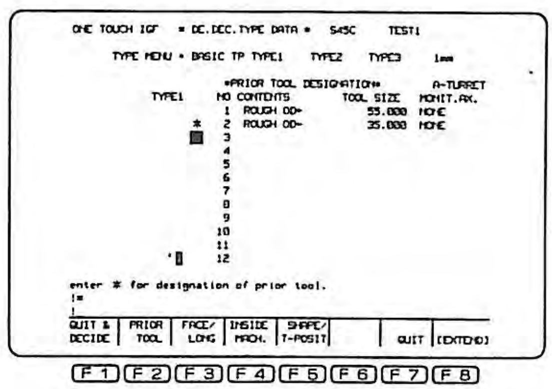
θ=40°

* Assume that only rough turning is executed.

When automatic process determination is attempted without prior tool designation, the following error occurs.

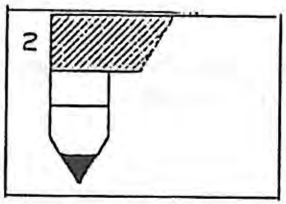
"Remained uncut area" error

When 2. ROUGH OD ← tool is designated as a prior tool, the OD rough machining process in which this tool is used is automatically generated.

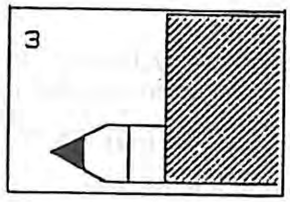


Note 1: *The prior tool must be registered in the tool data in advance.*

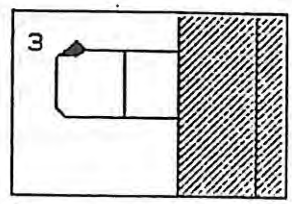
Note 2: *The following tools can not be registered as a prior tool. Designate them in process edit operation. The number in ( ) indicates the tool shape code number.*



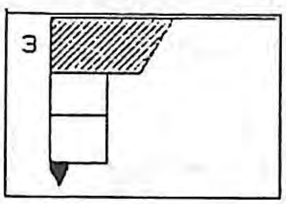
OD ← (2)
OD → (2)
GROOVE OD ↓ (2)



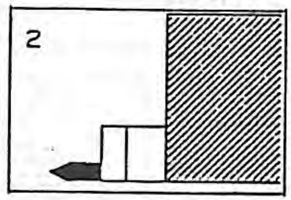
FACE ↓ (3)
FACE ↑ (3)



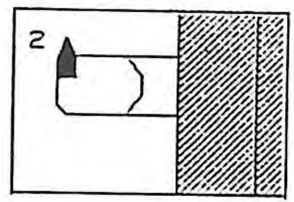
ID ← (3)
ID → (2)



GROOVE OD ↓ (3)



GROOVE FACE ← (2)



GROOVE ID ↑ (2)

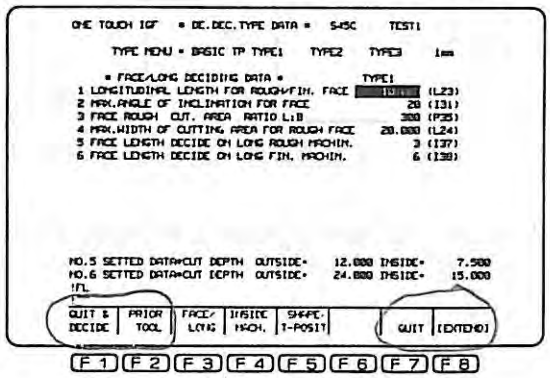
Note 3: *When the machining process in which the designated prior tool can not be used is being generated, the prior tool is not selected.*

(2) FACE/LONG DECIDING DATA

Whether the shape is an end face or a longitudinal portion is determined according to the data set on the FACE/LONG DECIDING DATA screen.

Operating Procedure:

1) Press the function key [F8] (EXTEND) on the start-up page of the graphic edit screen. Then, press the function key [F3] (DE. TYPE DATA).
2) Press the function key [F3] (FACE/LONG).
3) Using the cursor control keys ↓ and ↑, move the cursor to the data column and set the necessary data.
   * Data for four types of automatic process determination can be set. Switch the screen using the cursor control keys ← and →, and input data as needed.
   * When the function key [F2] (PRIOR TOOL), [F4] (INSIDE MACH.) or [F5] (SHAPE/T-POSIT) is pressed, the screen for setting other data will be displayed.
4) Press the function key [F1] (QUIT & DECIDE) or [F7] (QUIT).



(FACE/LONG DECIDING DATA screen)

Data to be designated:

- LONGITUDINAL LENGTH FOR ROUGH/FIN. FACE
  → Corresponds to the dimension parameter No. 23
- MAX.ANGLE OF INCLINATION FOR FACE
  → Corresponds to the integer parameter No. 31
- FACE ROUGH CUT. AREA RATIO L:B
  → Corresponds to the percent parameter No. 35
- MAX.WIDTH OF CUTTING AREA FOR ROUGH FACE
  → Corresponds to the dimension parameter No. 24
- FACE LENGTH DECIDE ON LONG ROUGH MACHIN.
  → Corresponds to the integer parameter No. 37
- FACE LENGTH DECIDE ON LONG FIN. MACHIN.
  → Corresponds to the integer parameter No. 38

According to the data set for the above items, the shape is judged to be a longitudinal portion or an end face.

Longitudinal portion/end face judgment

The shape is judged to be a longitudinal portion or an end face as shown by the flowchart on the following page.

The data numbers referenced in the flowchart indicate the following:

- Data No. 1

  → LONGITUDINAL LENGTH FOR ROUGH/FIN. FACE

- Data No. 2

  → MAX.ANGLE OF INCLINATION FOR FACE

- Data No. 3

  → FACE ROUGH CUT. AREA RATIO L:B

- Data No. 4

  → MAX.WIDTH OF CUTTING AREA FOR ROUGH FACE

- Data No. 5

  → FACE LENGTH DECIDE ON LONG ROUGH MACHIN.

- Data No. 6

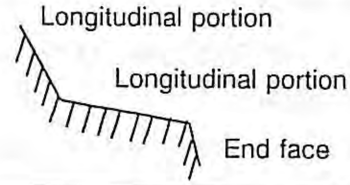
  → FACE LENGTH DECIDE ON LONG FIN. MACHIN.

- Depth of cut

  → Depth of cut for OD ← or OD → in roughing and copy turning in the material data.

Flowchart for longitudinal portion/end face judgment

Shape elements are classified into two groups, the longitudinal portion and the end face, by depending on the angle formed between the X-axis and Z-axis, set at Data No. 2.



End face ↓ / Longitudinal portion →

The end face whose dimension in the X-axis direction is smaller than the certain value is machined as a longitudinal portion.

Certain value:

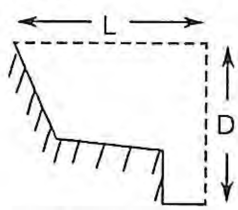
For roughing	Infeed amount × (Data No. 5)
For finishing	Infeed amount × (Data No. 6)

End face ↓ / Longitudinal portion →

The area is judged to be an end face and machined accordingly in the rough turning process when the blank material satisfies the following conditions:

The length of the stock removal in the Z-axis direction L < Data No. 4

Width (D)/length (L) ratio of blank material D/L ≧ Data No. 3



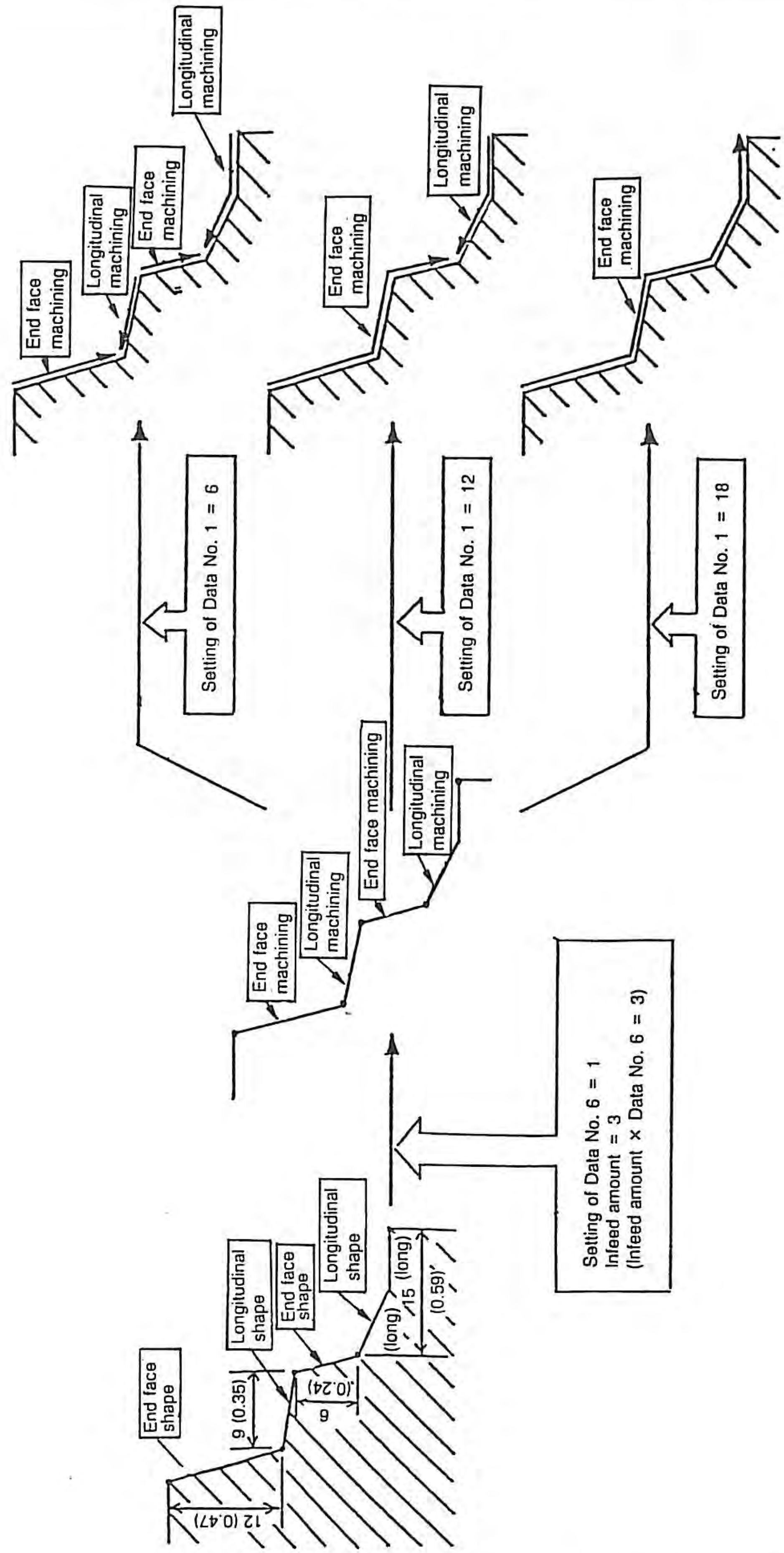
End face ↓ / Longitudinal portion →

When the dimension in the Z-axis direction of the longitudinal portion which is tangent to the end face is smaller than the setting of Data No. 1, this portion is judged to be an end face and machined accordingly.

End face ↓ / Longitudinal direction ↓

End face machining

Longitudinal machining


End face shape
Longitudinal shape
End face shape
Longitudinal shape
12 (0.47)
9 (0.35)
6 (0.24)
15 (0.59)
(long)
(long)
Setting of Data No. 6 = 1
Infeed amount = 3
(Infeed amount × Data No. 6 = 3)
End face machining
Longitudinal machining
End face machining
Longitudinal machining
Setting of Data No. 1 = 6
Setting of Data No. 1 = 12
Setting of Data No. 1 = 18
End face machining
Longitudinal machining
End face machining
Longitudinal machining
End face machining
Longitudinal machining
End face machining

(3) INSIDE MACHINING DATA

The data related with ID turning is set on the INSIDE MACHINING DATA screen.

Operating Procedure:

1) Press the function key [F8] (EXTEND) on the start-up page of the graphic edit screen. Then, press the function key [F3] (DE. TYPE DATA).

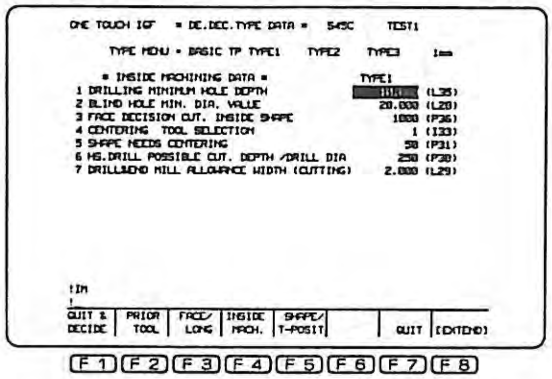
2) Press the function key [F4] (INSIDE MACH.).

3) Using the cursor control keys ↓ and ↑, move the cursor to the data column and set the necessary data.

   * Data for four types of automatic process determination can be set. Switch the screen using the cursor control keys← and → and input the data as needed.

   * When the function key [F2] (PRIOR TOOL), [F3] (FACE/LONG) or [F5] (SHAPE/T-POSIT) is pressed, the screen for setting other data will be displayed.

4) Press the function key [F1] (QUIT & DECIDE) or [F7] (QUIT).

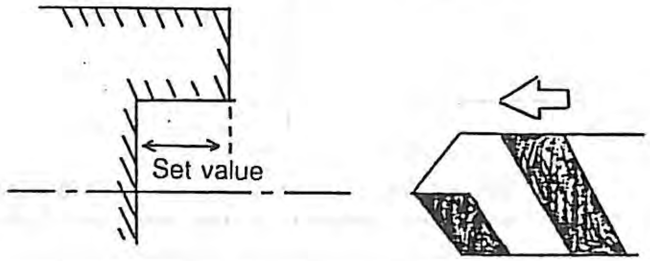


(INSIDE MACHINING DATA screen)

Data to be designated:

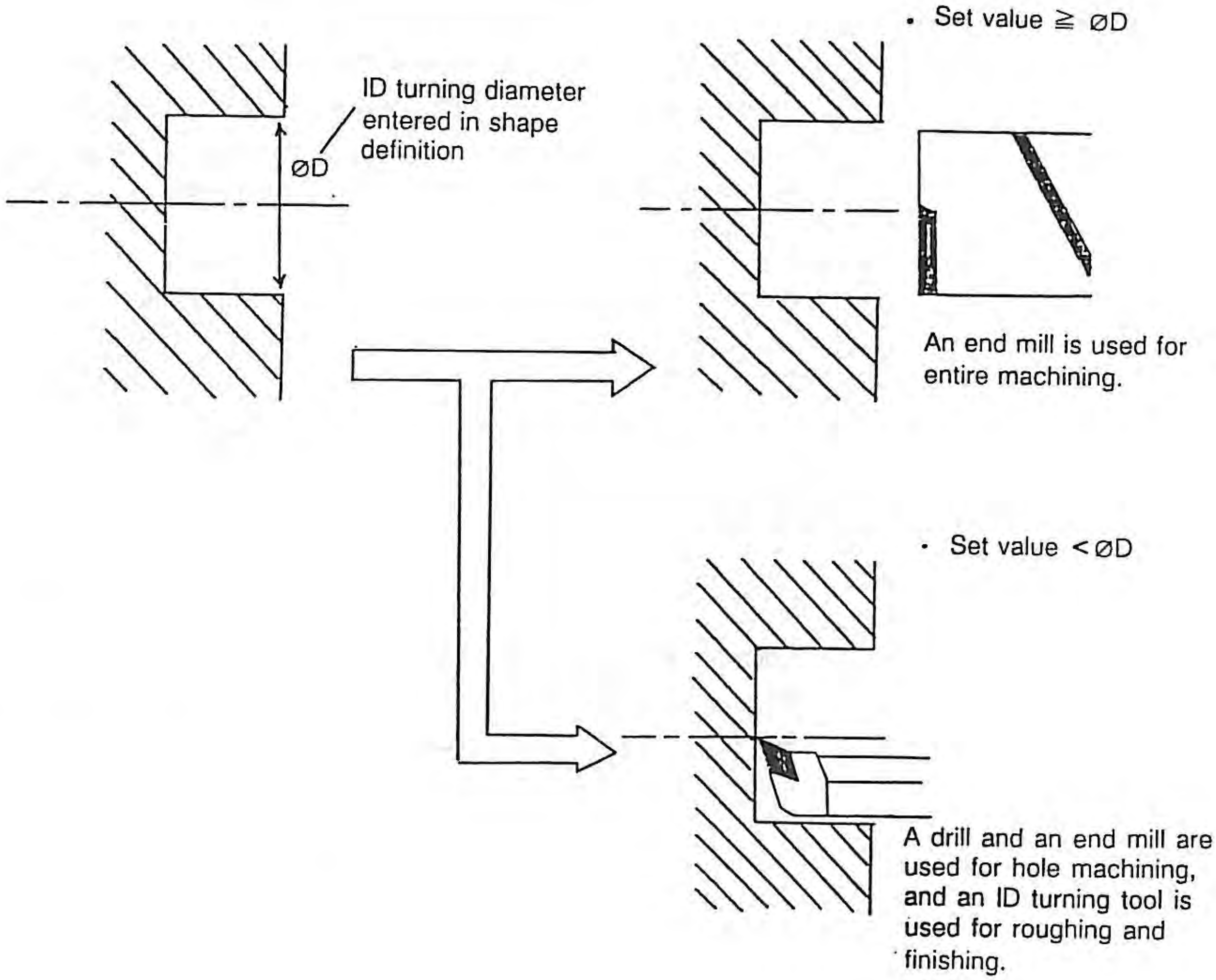
- DRILLING MINIMUM HOLE DEPTH → Corresponds to the dimension parameter No. 35

  The minimum stock value on the end face (cut depth) required for drilling a blind hole is set. When the actual value is smaller than the set value, endmilling is executed instead of drilling.

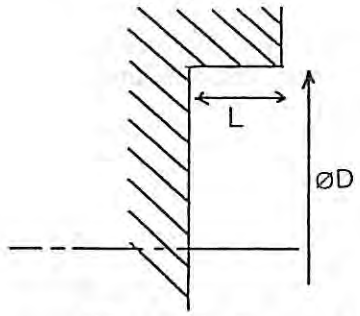


- BLIND HOLE MIN. DIA. VALUE → Corresponds to the dimension parameter No. 28

  The minimum blind hole diameter for which an ID turning tool can be used is set. Set the diameter of the smallest ID turning tools in hand.



- FACE DECISION CUT. INSIDE SHAPE → Corresponds to the percent parameter No. 36



* L and ØD are ID dimensions entered in shape definition

When ØD/L ≧ set value (%), ID turning is executed without using a drill. However, when L is too large and uncut portions are left when end face machining is executed, a drill or boring bar is also used. Enter a value in percent using an integer.

- CENTERING TOOL SELECTION (0: Center drill 1: Drill tool is used as center drill as well) → Corresponds to the integer parameter No. 33

  The tool used in centering is designated.

  0 : The center drilling process is generated automatically and centering is executed using a center drill

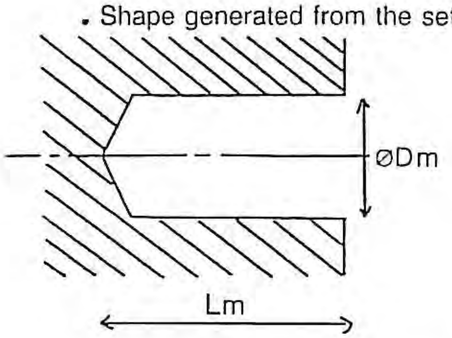
  1 : Centering is executed in the HSS drilling process.

- SHAPE NEEEDS CENTERING → Corresponds to the percent parameter No. 31

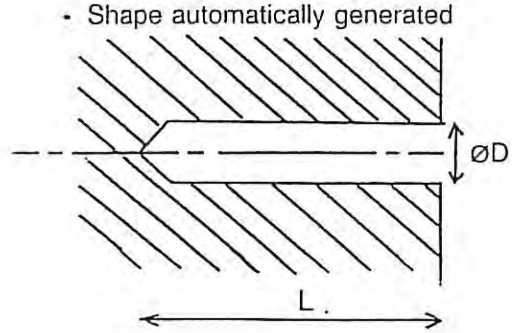
  Centering is executed when the shape in the automatically generated HSS drilling process is longer and narrower than the shape generated from the set data.

  Example:

• Shape generated from the set data



• Shape automatically generated



Calculate the value Lm/ØDm and set it.

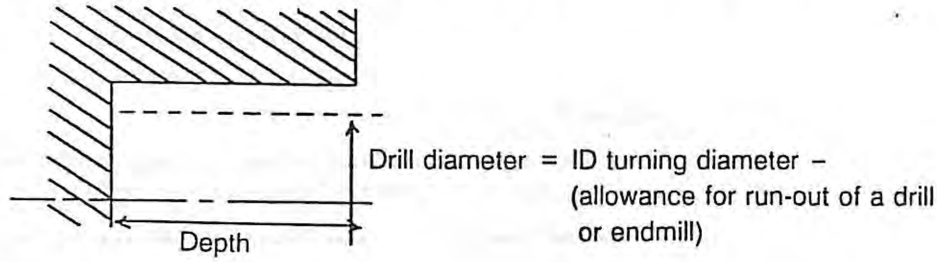
When (depth L)/(drill diameterØD) is equal to or larger than the set value (%), centering is executed.

Enter a value in percent using an integer.

- HS.DRILL POSSIBLE CUT. DEPTH/DRILL DIA → Corresponds to the percent parameter No. 30

  In automatic process determination, a carbide tool is selected for the shape which allows the use of a carbide tool.

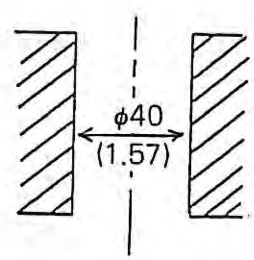
  When (depth)/(drill diameter) is equal to or smaller than the set value of the parameter (%), a carbide tool is selected.



- DRILL&END MILL ALLOWANCE WIDTH (CUTTING) → Corresponds to the dimension parameter No. 29

  Set the allowance taking the run-out of the tool in drilling into consideration.

  Example:



To machine the ID of 40 mm

↓

Set "2" for DRILL&END MILL ALLOWANCE WIDTH (CUTTING).

↓

The machining process is generated automatically in which drilling is executed using a drill within 38 mm (* Ø38 = Ø40 – Ø2) in diameter and roughing and finishing are executed using a boring bar.

(4) SHAPE/TOOL POSITION DATA

How to designate the finishing stock and the tool indexing position are set.

Operating Procedure:

1) Press the function key [F8] (EXTEND) on the start-up page of the graphic edit screen. Then, press the function key [F3] (DE. TYPE DATA).

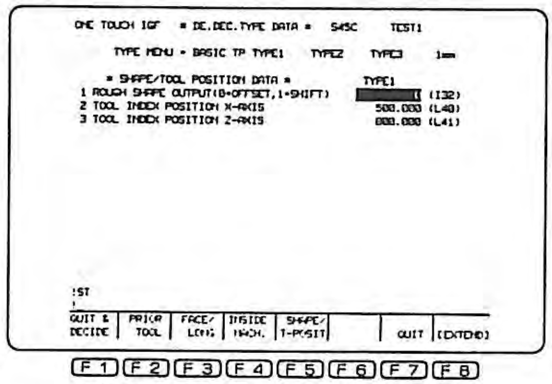
2) Press the function key [F5] (SHAPE/T-POSIT).

3) Using the cursor control keys ↓ and ↑, move the cursor to the data column and set the necessary data.

   * Data for four types of automatic process determination can be set. Switch the screen using the cursor control keys ← and → and input the data as needed.

   * When the function key [F2] (PRIOR TOOL), [F3] (FACE/LONG) or [F4] (INSIDE MACH.) is pressed, the screen for setting other data will be displayed.

4) Press the function key [F1] (QUIT & DECIDE) or [F7] (QUIT).



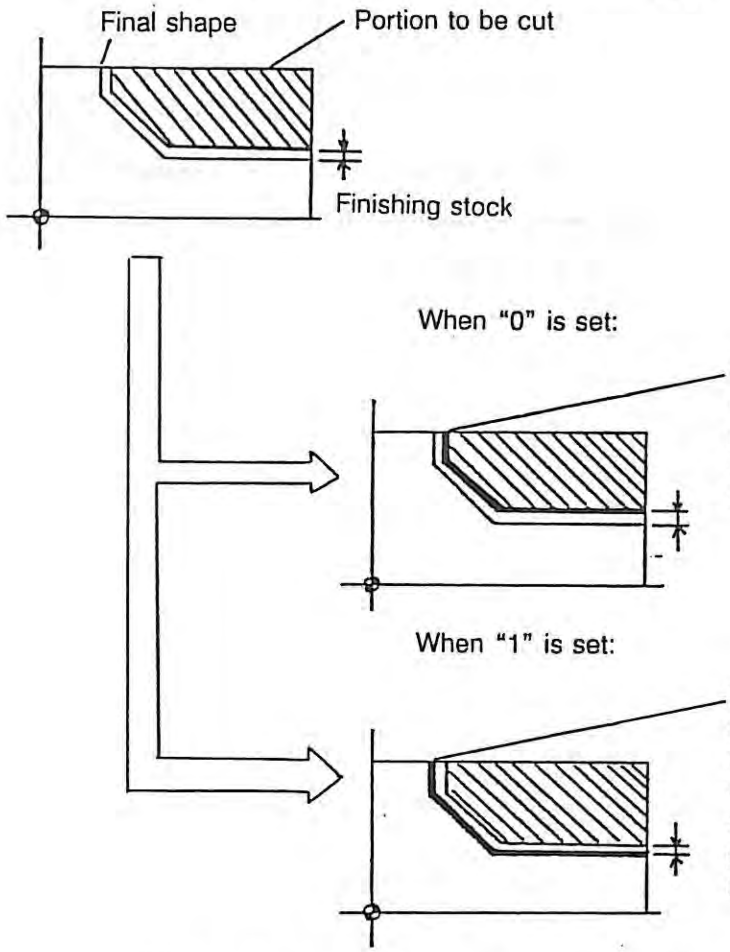
(SHAPE/TOOL POSITION DATA)

Data to be designated:

- ROUGH SHAPE OUTPUT (0=OFFSET, 1=SHIFT) → Corresponds to the integer parameter No. 32

  Whether or not the finishing stock which is defined for X-axis (LX) and Z-axis (LZ) in the material data setting is added to the finished shape in the rough machining process is set.

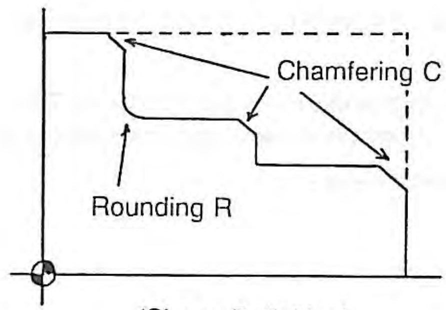
  The shape to be cut in the rough machining process



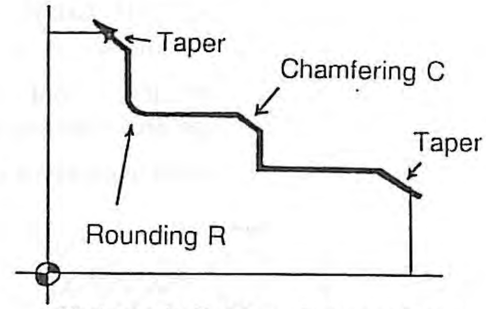
When "0" is set: The shape indicated by a bold line (▬) is determined automatically for the rough machining process. Finishing stock is added, outputting neither U nor W command to the program.

When "1" is set: The shape indicated by a bold line (▬) is determined automatically for the rough machining process. Finishing stock is not added. U and W commands are output to the program. The finishing process is generated also for the area where surface roughness "∇" is designated.

When "1" is set, the shape element which was designated as "rounding R" or "chamfering C" in shape definition is designated also as the shape element of "rounding R" or "chamfering C" in automatic process determination. However, when "rounding R" or "chamfering C" was defined as the starting or end shape element for the rough machining process, "ARC" is generated for "rounding R" and "TAPER" for "chamfering C" automatically. (This is to prevent uncut.)



(Shape Definition)



(Automatic Process Determination)

- TOOL INDEX POSITION X-AXIS → Corresponds to the dimension parameter No. 40
- TOOL INDEX POSITION Z-AXIS → Corresponds to the dimension parameter No. 41

Set the tool indexing position.

(5) Data Setting for Individual Process Determination Patterns

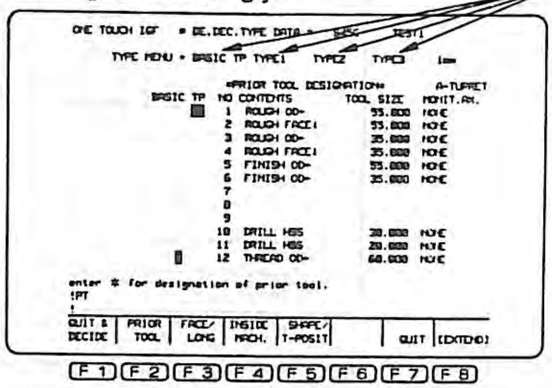
As the automatic process determination pattern (type), the following four patterns are available.

BASIC TP, TYPE1, TYPE2, TYPE3

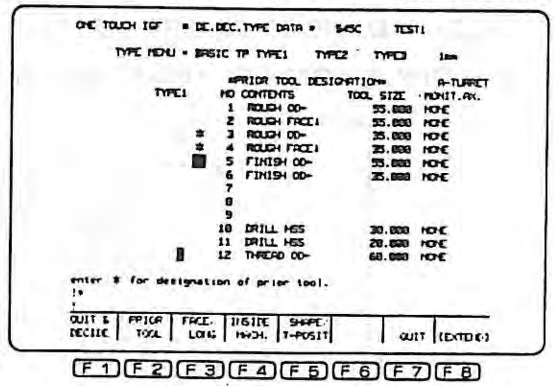
Set the prior tool data, face/long judgment data, ID turning data, and shape/tool indexing position data for these four patterns, and call the pattern that meets your requirements.

Operating Procedure:

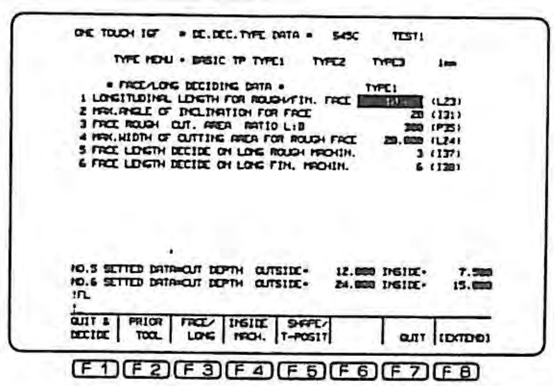
1) Each time the cursor control key ← or → is pressed on the PRIOR TOOL DESIGNATION screen, BASIC TP, TYPE1, TYPE2, and TYPE3 is displayed in yellow color sequentially, and the display page is changed accordingly.



2) Move the cursor to TYPE1 using the cursor control key ← or →. The PRIOR TOOL DESIGNATION screen for TYPE1 will be displayed. Input data as required.



3) When the function key [F3] (FACE/LONG), [F4] (INSIDE MACH.) or [F5] (SHAPE/T-POSIT) is pressed, the screen for setting other data of TYPE1 will be displayed. Input data as required.



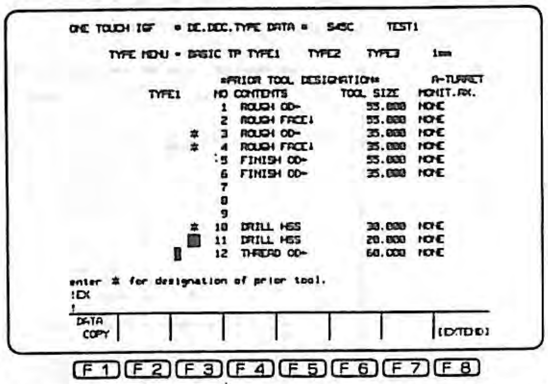
(6) Condition Copy

The data in the pattern can be copied to another pattern.

Operating Procedure:

This is the procedure to copy the prior tool designation data in TYPE1 to TYPE3.

1) Move the cursor to TYPE1 using the cursor control key ← or →, and press the function key [F2] (PRIOR TOOL). The PRIOR TOOL DESIGNATION screen for TYPE1 will be displayed. Input the data as required.

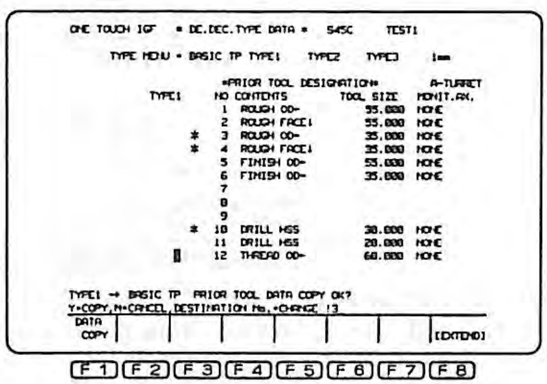


2) Press the function keys [F8] (EXTEND) and [F1] (DATA COPY) in this order. The following prompt will be displayed.

"MENU No. 0 = BASIC TYPE, 1 = TYPE1, 2 = TYPE2, 3 = TYPE3

TYPE1 → BASIC TP PRIOR TOOL DATA COPY OK?

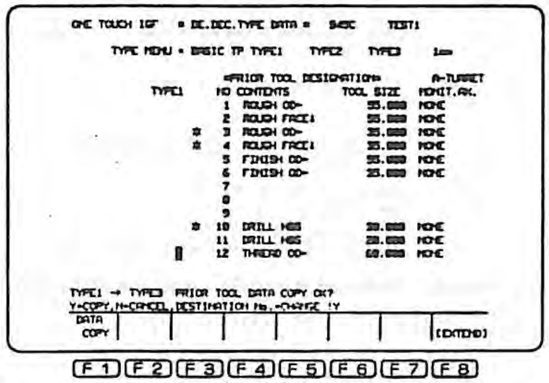
Y = COPY, N = CANCEL, DESIGNATION No. = CHANGE"



3) Since the data is required to be copied to pattern 3, key in "3" and press the **WRITE** key. The following prompts will be displayed.

"TYPE1 → TYPE3 PRIOR TOOL DATA COPY OK?

Y = COPY, N = CANCEL, DESTINATION No. = CHANGE"



4) Key in Y and press the **WRITE** key. The data will be copied.

Supplement:

- To copy the data to BASIC TP, key in "Y" and press the WRITE key in step 3).
- To cancel copy operation, key in "N" and press the WRITE key in step 3).

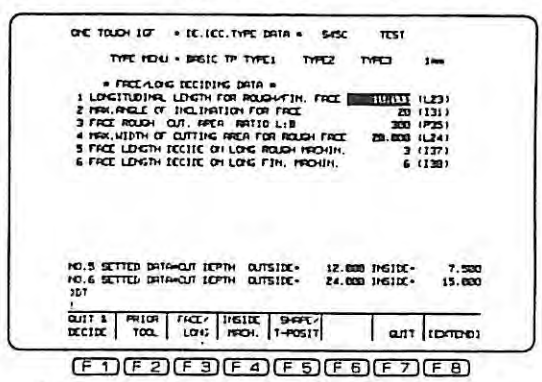
(7) Guide Setting

The guide indication can be entered when the condition setting data is set for each automatic process determination pattern. The entered guide indication is displayed on the condition setting screens and on the start-up page of the graphic edit screen.

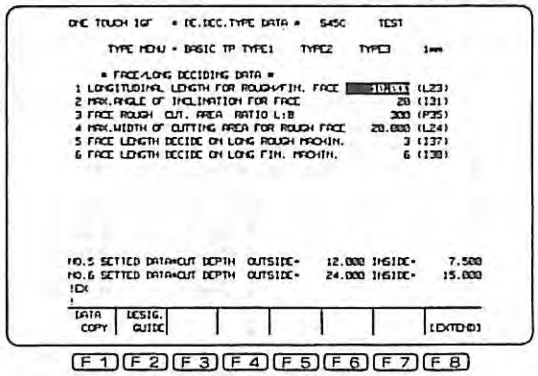
Operating Procedure:

This is the procedure to display guide indication "ROUGH <L>" for TYPE 3.

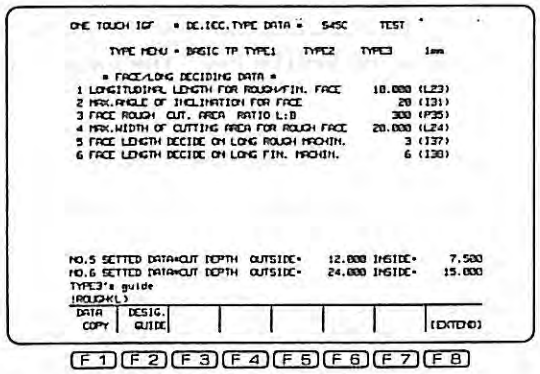
1) High light "TYPE 3" in yellow color using the cursor control keys ← and →.



2) Press the function keys [F8] (EXTEND) and [F2] (DESIG. GUIDE).



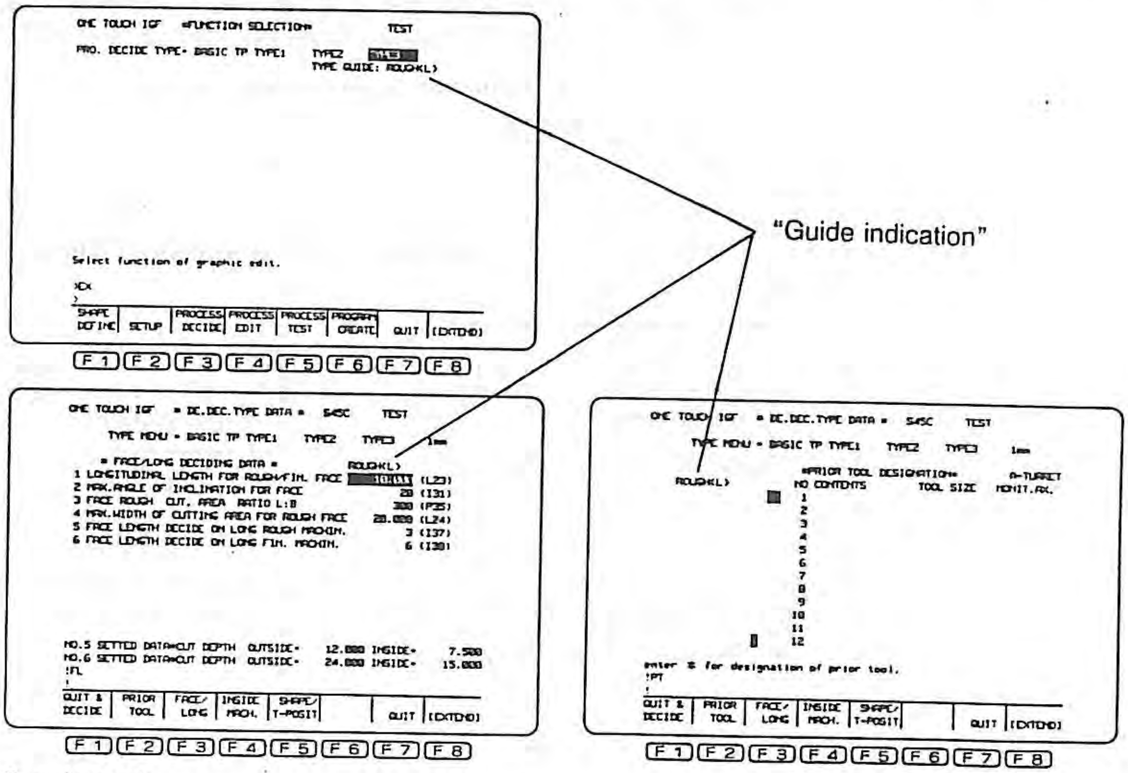
3) The prompt "TYPE 3's guide" will be displayed. Key in [R], [O], [U], [G], [H], [<], [L], [>], and press the **WRITE** key.



4) This completes guide setting.

Guide indication on each screen

The entered guide indication is displayed as illustrated below. When the automatic process determination pattern is switched using the cursor control keys ← and →, the guide indication changes accordingly.



Supplement:

The following characters and symbols can be used for the guide indication:

A - Z, 0 - 9, (space), !, ", #, $, (,), @, &, ', ?, %, ¥, ^, [,], -, ;, :, <, >, /, *, +, −, =, , (comma), and . (period).

The number of characters and symbols in the guide indication must not exceed 16.

The following cannot be used as the guide indication:

CO, GU, EX, QD, PT, FL, IM, ST, Q, and (space only)

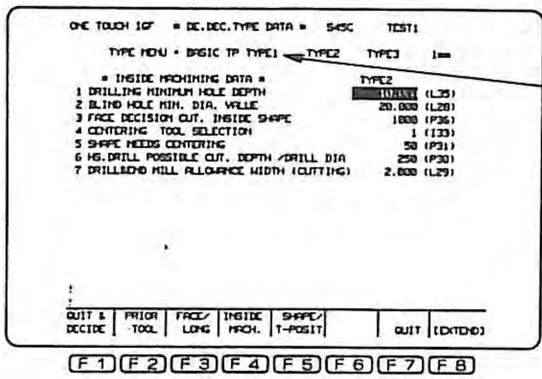
(8) Ending Condition Setting

After the condition setting data is set, press the function key [F7] (QUIT). This ends condition setting operation. The data registered in the patterns is saved in the bubble memory, and the one-touch IGF data which corresponds to the process determination condition change data is changed accordingly so that the parameter data coincides with the BASIC TP data.

(9) Ending Condition Setting and Determining Machining Process

After the condition setting data is set, press the function key [F1] (QUIT & DECIDE). This ends condition setting operation and determines the machining process automatically.

The pattern which is selected when the function key [F1] (QUIT & DECIDE) is pressed (the cursor-located pattern) is designated as an automatic process determination pattern.



Move the cursor to the pattern which is designated as an automatic process determination pattern, using the cursor control key ← or →, and press the function key [F1] (QUIT & DECIDE). This ends condition setting operation and determines the machining process automatically.

6. Automatic Process Determination Using the Condition Setting Function

The machining process can be determined automatically using the automatic process determination pattern (BASIC TP, TYPE1, TYPE2, or TYPE3) that is preset using the condition setting function.

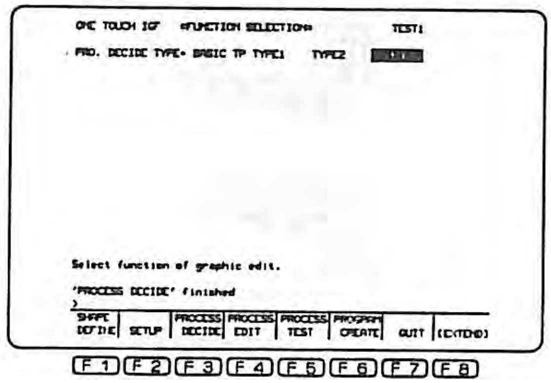
Screen Transfer:

Start-up page of one-touch IGF screen

[F1] (GRAPHIC EDIT)
+ IGF file name
+ **WRITE**

[F7] (IGF QUIT)

1) On the start-up page of the graphic edit screen, move the cursor to the desired pattern using the cursor control keys ← and →.
2) Press the function key [F3] (PROCESS DECIDE).
3) When automatic process determination is complete, the prompt "'PROCESS DECIDE finished'" will be displayed.



Supplement:

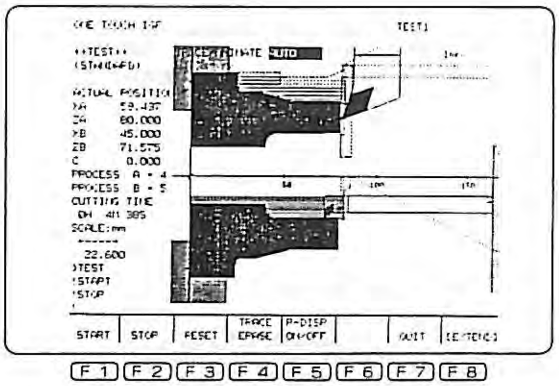
To determine the machining process from the standard hole data, material data, one-touch IGF parameter data, and tool data without changing the automatic process determination conditions, skip condition setting operation and select "BASIC TP" as the automatic process determination pattern.

SECTION 10 PROCESS TEST

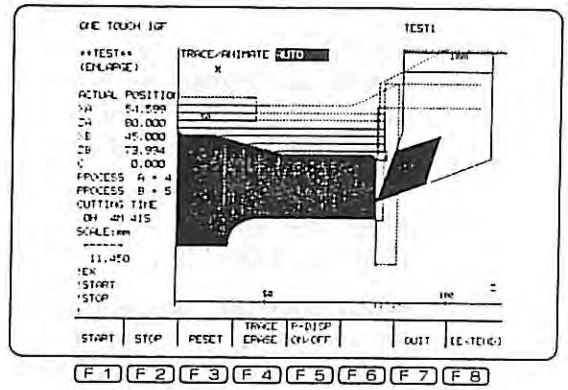
1. What is the Function of "Process Test"?

This is the function to simulate the part program created through the one-touch IGF on the CRT using animation.

Two types of screen are available to check the part program. One is the standard screen on which the entire blank material shape can be displayed, and the other is the enlarged screen on which a part of blank material shape is enlarged and displayed.



Standard screen



Enlarged screen

On individual screens, the actual position of tool, process number, and cutting time are displayed in accordance with the progress of test operation.

As for tool path, cutting feed is displayed by solid lines, and rapid feed is displayed by dotted lines.

2. Test

The following functions are provided for process test operation.

(1) START

(2) STOP

(3) RESET

(4) TRACE ERASE

(5) P-DISP ON/OFF

(6) STD./ENLARGE

(7) ENLARGE SCALE

(8) TRACE/ANIMATE

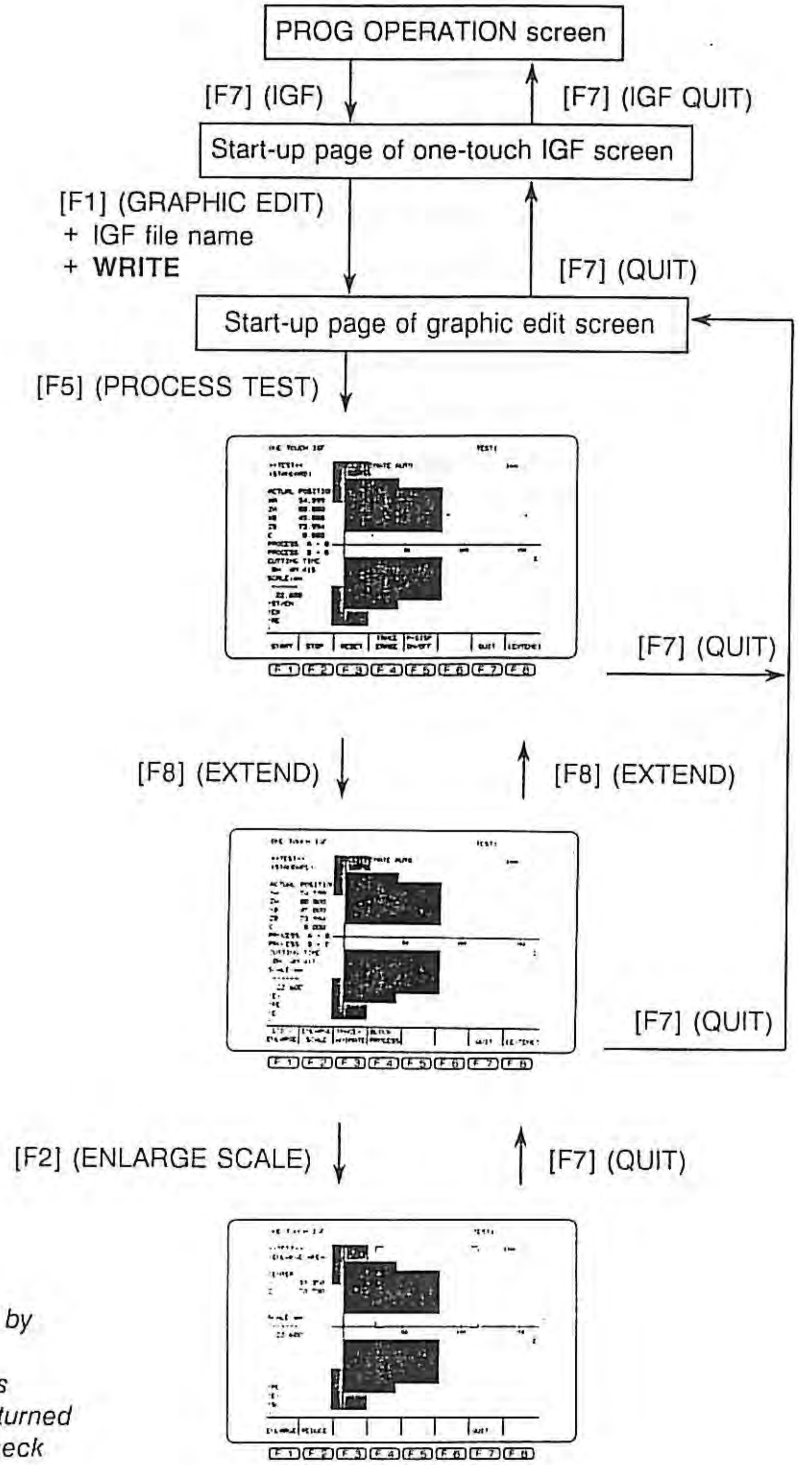
(9) BLOCK/PROCESS

(10) SIDE/FRONT

Functions from (1) to (9) are assigned to the function keys. As for (10), switching between side and front is carried out using page keys **PAGE ↑** and **PAGE ↓**.

Screen Transfer:

The CHUCK·CENTER screen is called from the PROG OPERATION screen as diagramed below.



Following functions are assigned:

[F1] (START)

[F2] (STOP)

[F3] (RESET)

[F4] (TRACE ERASE)

[F5] (P-DISP ON/OFF)

Following functions are assigned:

[F1] (STD./ENLARGE)

[F2] (ENLARGE SCALE)

[F3] (TRACE/ANIMATE)

[F4] (BLOCK/PROCESS)

Note: When the TEST screen is called by pressing the function key [F3] (PROCESS TEST) in the process editing mode, the CRT will be returned to the corresponding process check screen.

(1) **START**

a) Function

Process test is started.

b) How to Designate

Press the function key [F1] (START). Process test is started.

Note 1: *With the two-saddle specification model, when the integer parameter No. 26 TOOL INTERFERENCE CHECK M-CODE OUTPUT is set at "1", interference of tools on the A- and B-turrets is checked during the execution of process test. If tool interference has occurs, an error will occur and process test is suspended. To resume process test, press the function key [F1] (START) again.*

Note 2: *When the integer parameter No. 23 CHUCK BARRIER CHECK M-CODE OUTPUT is set at "1", tool edge entered the chuck barrier is checked during the execution of process test. If the tool edge has entered the chuck barrier, an error will occur and process test is suspended. To resume process test, press the function key [F1] (START) again.*

Note 3: *When the integer parameter No. 22 TAILSTOCK BARRIER CHECK M-CODE OUTPUT is set at "1", tool edge entered the tailstock barrier is checked during the execution of test operation. If the tool edge has entered the tailstock barrier, an error will occur and process test is suspended. To resume process test, press the function key [F1] (START) again.*

(2) **STOP**

a) Function

Process test is temporarily suspended.

This function is used to check the progress of machining in the suspended state.

b) How to Designate

Press the function key [F2] (STOP). Process test is temporarily suspended.

Press the function key [F1] (START) to resume process test from the position where it has been suspended.

Note: *Counting of cutting time is suspended during the STOP function is activated.*

(3) **RESET**

a) Function

Process test is returned to the initial state where only the shape of blank material, chuck, and tailstock center is displayed.

This function is used to return process test to the start after the completion of process test or during the execution of process test.

b) How to Designate

Press the function key [F3] (RESET). Process test is returned to the initial state.

(4) TRACE ERASE

a) Function

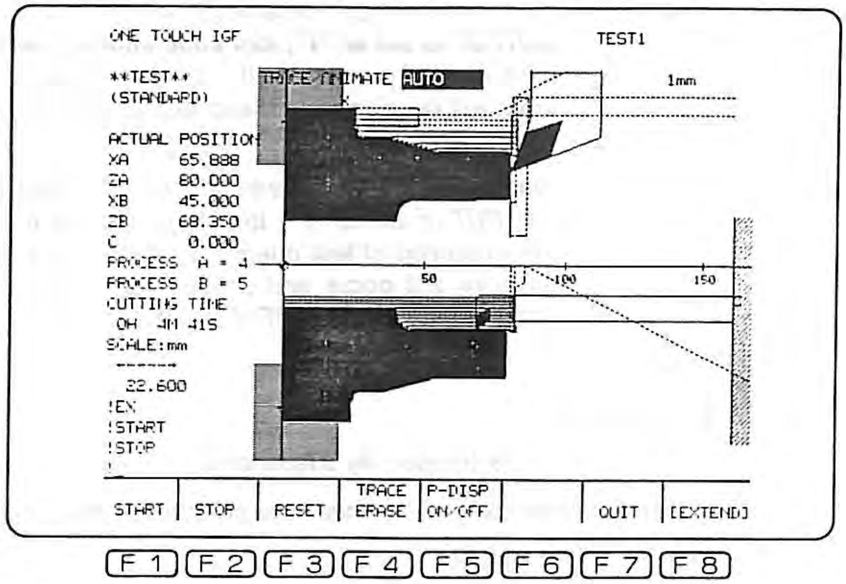
Tool paths are erased from the TEST screen.

This function is used when tools paths interrupt the view of the workpiece on the screen.

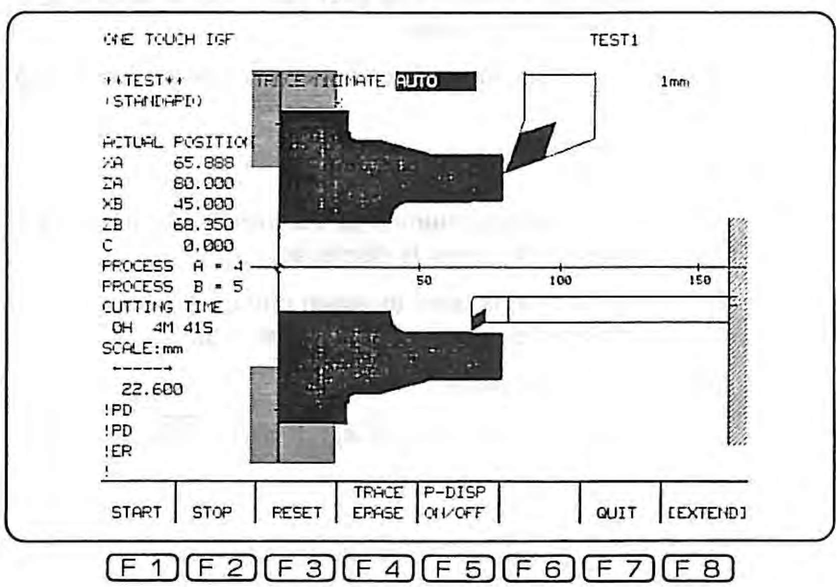
b) How to Designate

Press the function key [F4] (TRACE ERASE). Tool paths which have been drawn before the pressing of the function key will be erased. Display of tool paths is resumed after that.

Note: *Pressing the function key [F4] (TRACE ERASE) has no effect unless the TRACE function is selected. (Refer to (8) "TRACE/ANIMATE" on page 242.)*



[F4] (TRACE ERASE)



(5) **P-DISP ON/OFF**

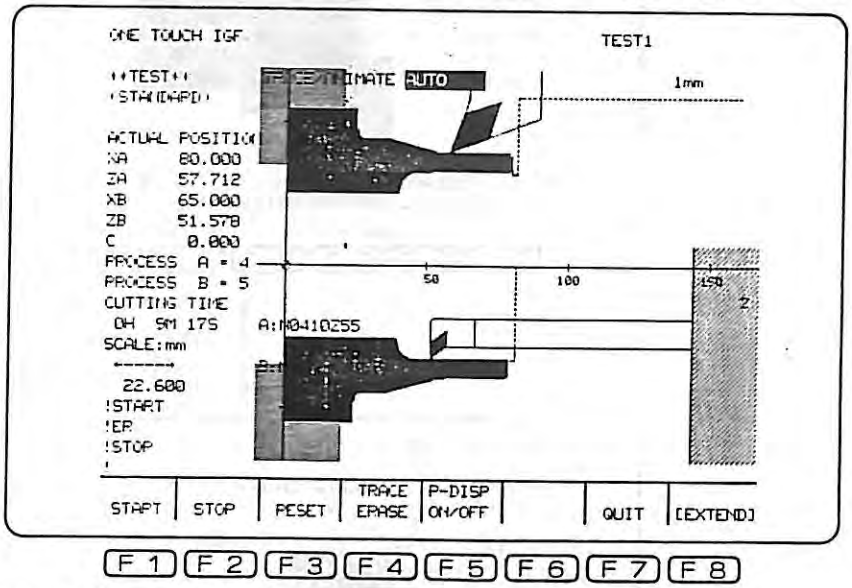
a) Function

The block of program currently executed during process test is displayed on the CRT screen.

This function is used to check the content of the program currently executed.

b) How to Designate

Display is turned on and off each time the function key [F5] (P-DISP ON/OFF) is pressed.



(6) **STD./ENLARGE**

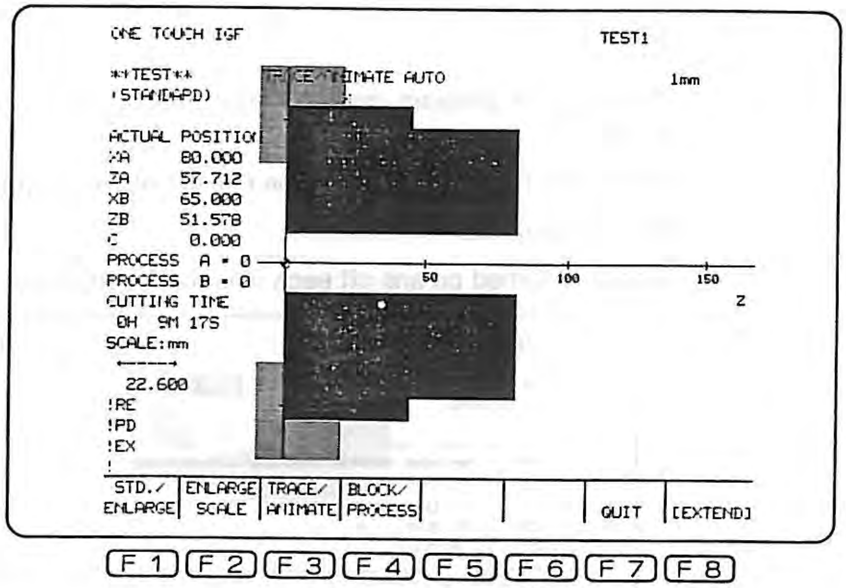
a) Function

Switching between the standard and enlarged screen modes is executed. Standard screen refers to the screen on which the entire blank material shape can be displayed, and enlarged screen refers to the screen on which a part of blank material shape is enlarged and displayed.

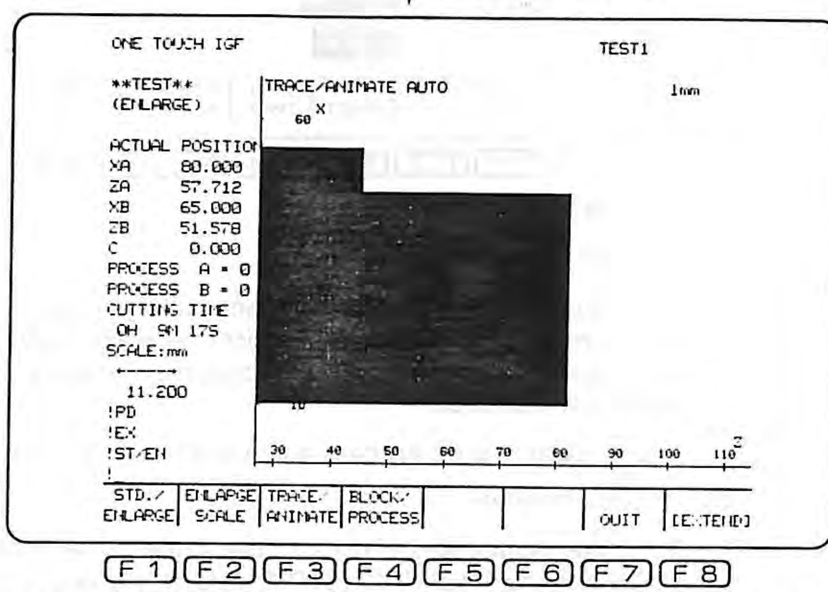
This function is used to check a part of machining in the enlarged size.

b) How to Designate

Press the function key [F1] (STD./ENLARGE) when process test is not in progress. Each time the function key [F1] (STD./ENLARGE) is pressed, the screen is switched between the standard and enlarged display modes.



[F1]
(STD./ENLARGE)



Note 1: *Switching between the standard and enlarged screen modes is not possible when process test is in progress.*

Note 2: *Display of the enlarged screen is not possible unless ENLARGE SCALE is set. For the procedure to set the enlarge scale, refer to (7) "ENLARGE SCALE" on page. 241.*

(7) ENLARGE SCALE

a) Function

The size of display on the ENLARGE screen and the portion to be displayed are designated.

This function is used to check a part of machining in the enlarged size.

b) How to Designate

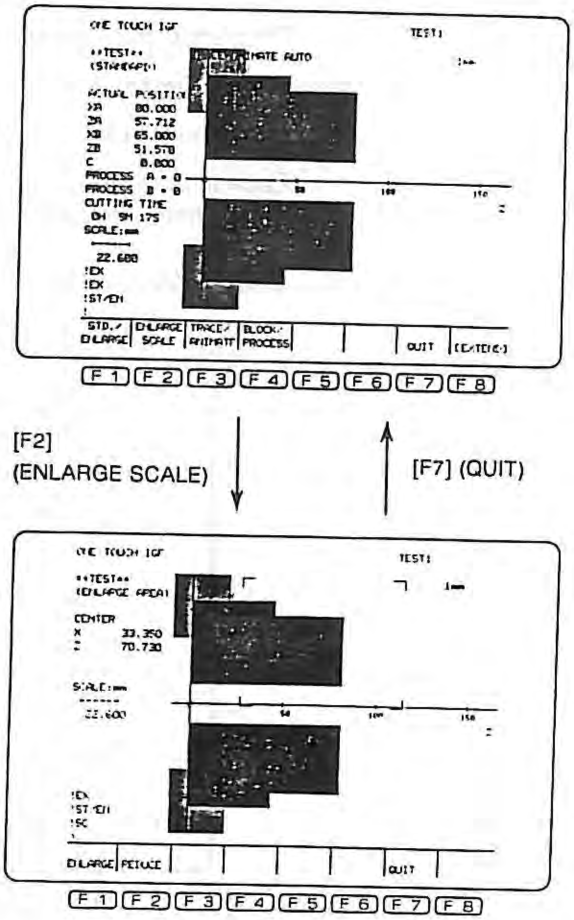
Press the function key [F2] (ENLARGE SCALE) when process test is not in progress. The ENLARGE screen will be displayed.

Each time the function key [F1] (ENLARGE) is pressed, the frame area is reduced. (Display on the ENLARGE screen is enlarged.)

Each time the function key [F2] (REDUCE) is pressed, the frame area is enlarged. (Display on the ENLARGE screen is reduced.)

Each time the cursor control key ↑, ↓, ←, or → is pressed, the frame is shifted in the direction specified by the cursor control key pressed, and the frame area is changed.

Press the function key [F7] (QUIT). The size of display on the ENLARGE screen and the portion to be displayed are defined.



Note 1: *Display of the figure smaller than the standard screen is not possible.*

Note 2: *Area of frame to be enlarged, reduced, or shifted is limited.*

Note 3: *When the function key [F2] (ENLARGE SCALE) is pressed while the STANDARD screen is displayed, enlargement for the figure on the STANDARD screen is enabled.*

Note 4: *When the function key [F2] (ENLARGE SCALE) is pressed while the ENLARGE screen is displayed, enlargement for the figure on the ENLARGE screen is enabled.*

Note 5: *Switching between side view and front view is not possible on the ENLARGE AREA screen. (Multi-machining specification: optional)*

(8) TRACE/ANIMATE

a) Function

The graphic display mode for process test includes the ANIMATE (cut portion of workpiece by tools), TRACE (tool edge path), and TRACE/ANIMATE modes.

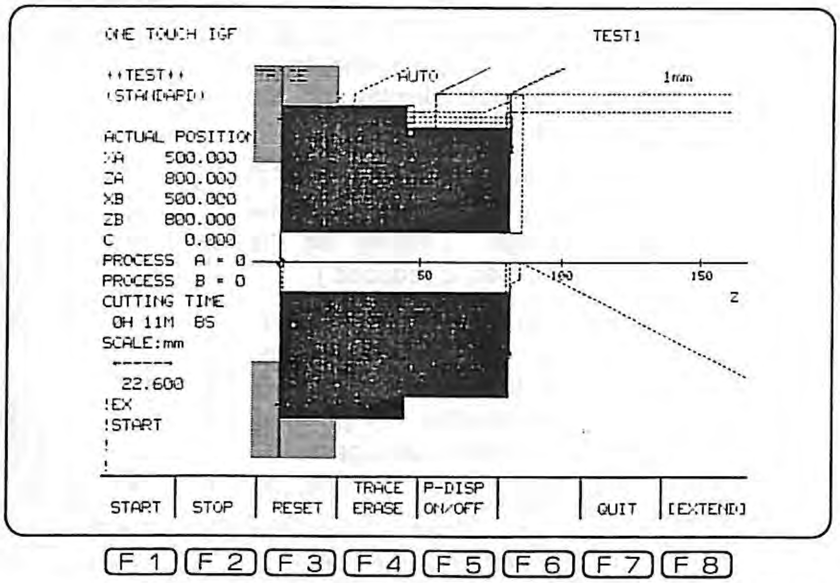
This function is used to select the mode from the above three modes.

b) How to Designate

Press the function key [F3] (TRACE/ANIMATE) when process test is not in progress.

Each time the function key [F3] (TRACE/ANIMATE) is pressed, the graphic display mode is switched in the following sequence.

TRACE/ANIMATE→TRACE→ANIMATE



Note 1: *Switching between TRACE, ANIMATE, and TRACE/ANIMATE is not possible on the ENLARGE AREA screen.*

Note 2: *As for tool path, cutting feed is displayed by solid lines, and rapid feed is displayed by dotted lines.*

(9) BLOCK/PROCESS

a) Function

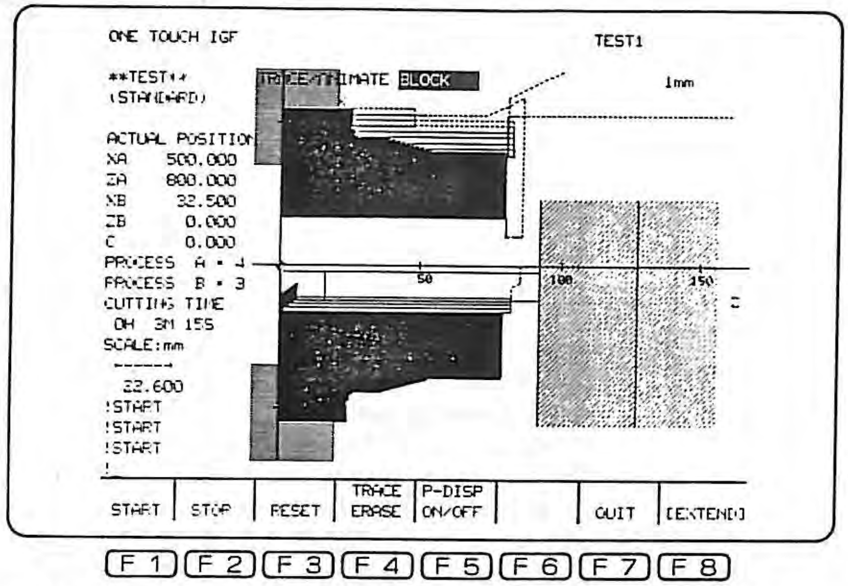
The process test mode includes the following three modes:

AUTO	Process test is executed up to the end of the program.
BLOCK STOP	Process test is executed in units of blocks.
PROCESS-TOP	Process test is executed in units of processes.

b) How to Designate

Each time the function key [F4] (BLOCK/PROCESS) is pressed, the process test mode is switched in the following sequence.

AUTO→BLOCK STOP→PROCESS STOP



Restart of process test in the block-stop or process-stop mode is possible by pressing the function key [F1] (START).

Note: *Switching between BLOCK and PROCESS is possible when process test is in progress.*

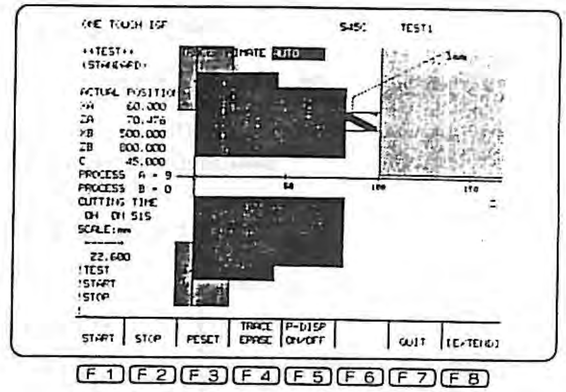
(10) **SIDE/FRONT** (multi-machining specification: optional)

a) Function

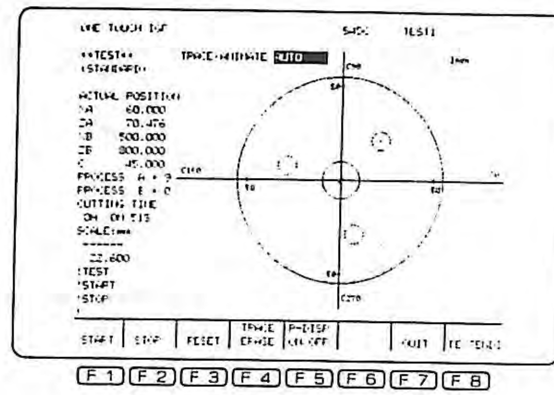
The direction of graphic display, front view or side view, is designated.

b) How to Designate

Each time the page key **PAGE ↑** or **PAGE ↓** is pressed, switching between the front view and side view occurs.



PAGE ↑ ↕ **PAGE ↓**



Note 1: *Differing from the process test for turning, cut portion is not removed from the blank material in the process test for multiple machining.*

Note 2: *The front view is effectively used to check the hole positions in drilling or to check the tool path in contour generating machining.*

SECTION 11 PROCESS EDIT

1. What is the Function of "Process Edit"?

Process edit is the function to enable the editing of the processes which have been automatically determined by the one-touch IGF, so that the know-how of the user can be reflected on the part program. Process edit includes change, addition, and deletion of the processes.

Checking the Contents of Process (process check screen)

- The IGF data (finished shape, machining method, cutting time, etc.) determined in the process definition process can be checked on the process check screen.
- The IGF data displayed on the process check screen are edited.

Editing in Units of Process

- This is the function to modify the machining process data that is automatically determined in units of process, through inserting, deleting, replacing, or copying the process.
- This function can also be used to make the user-specific IGF data and to use it to create another IGF data.

Editing the Contents of Process

- This is the function to make corrections on the finished shape or machining method (from LONG to TAPER, changing groove width, or increasing feedrate, etc.) of the process designated.
- The screen changes automatically according to the kind of process to be corrected.
- It is also possible to directly correct the NC program.

Process Test

- The result of process editing operation can be checked immediately in the animation mode.
- Process test is executed in the same manner as explained in Section 10.

Note: *The following message will be designated at the completion of process edit operation when it is attempted in the process edit mode to correct the IGF file which caused an error in process definition.*

Have you removed all decide errors from this file? (Y/N)

*Key in "Y" and press the **WRITE** key when the error was corrected in the process edit mode.*

*Otherwise key in "N" and press the **WRITE** key.*

Operating Procedure for Process Edit:

(1) Open the IGF file for process edit operation.

(2) Establish the PROCESS EDITING mode.

(3) Move the cursor (reverse display) to the process number for which process editing operation is executed, using the cursor control keys ↑, ↓, ←, and →, and page keys **PAGE↑** and **PAGE↓**.

(4) Press the function keys (INFORMATION↓), (INFORMATION↑) and check the contents of the process on the process check screen.

Contents to be checked:

- Finished shape
- Machining method
    Process kind, cutting direction, tool data, cutting conditions
- Cutting time, total time

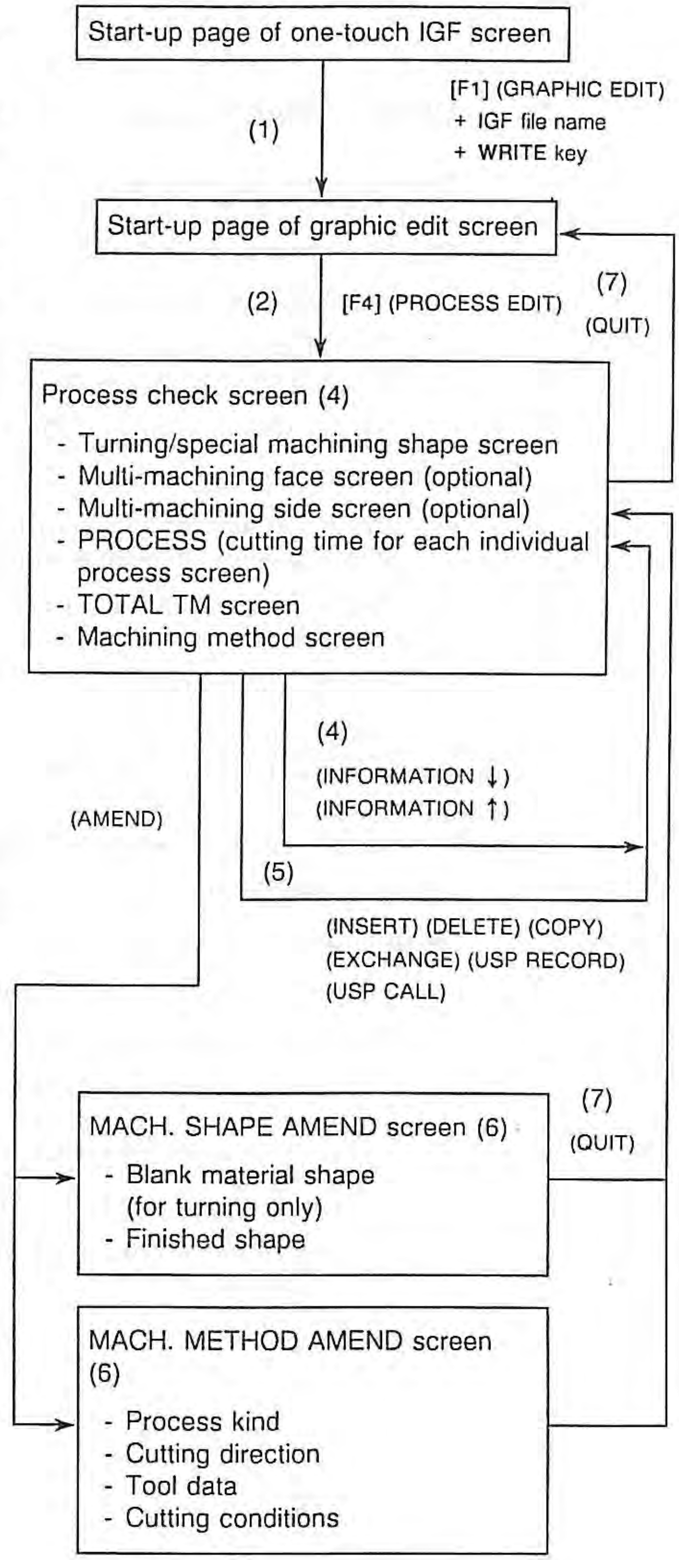
(5) Editing in units of process

Change the sequence of process or delete the process using the function keys (INSERT), (DELETE), (EXCHANGE), (COPY), (USP RECORD), and (USP CALL).

(6) Editing the contents of process

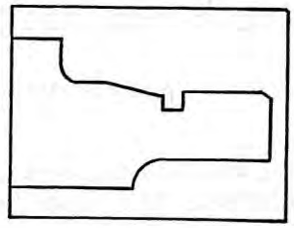
Change the finished shape, machining method, etc. using the function key (AMEND).

(7) Press the function key (QUIT) and define the result of editing.



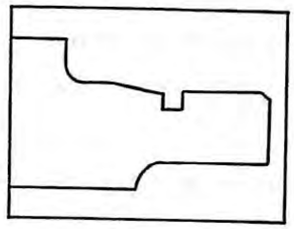
Example of Process Editing:

Shape definition



Finished shape definition

Automatic process determination

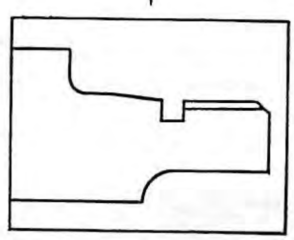


1	DRILL THRU
2	ROUGH O. FACE ↓
3	ROUGH OD←
4	ROUGH ID←
5	FIN. O. FACE ↓
6	FIN. OD←
7	FIN. ID←
8	R. W. GRV OD ↓
9	F. W. GRV OD ↓

Process editing

(Editing the contents of process)
(Addition of GROOVE OD← process)

(Editing the process in unit)
(Deletion of F. W. GRV OD ↓)



1	DRILL THRU
2	ROUGH O. FACE ↓
3	ROUGH OD←
4	ROUGH ID←
5	FIN. O. FACE ↓
6	FIN. OD←
7	FIN. ID←
8	R. W. GRV OD ↓
10	GROOVE OD←

2. Checking the Contents of Process (Process Check Screen)

The explanation that follows is for the process check screen on which the IGF data is displayed to enable process edit operation.

To edit the contents of process, move the cursor to the process to be corrected on the screen, and press the function key [F1] (AMEND). The screen for the process to be edited will be displayed.

For example, to correct the shape created in the ROUGH OD← cycle, display the turning/special machining shape screen, move the cursor (reverse display) to ROUGH OD←, and press the function key [F1] (AMEND). The MACH. SHAPE AMEND screen for ROUGH OD← will be displayed.

Following process check screens are provided.

(1) Turning/special machining shape screen

(2) Multi-machining face shape screen (optional: multi-machining specification)

(3) Multi-machining side shape screen (optional: multi-machining specification)

(4) PROCESS (cycle time for each individual process) screen

(5) TOTAL TM screen

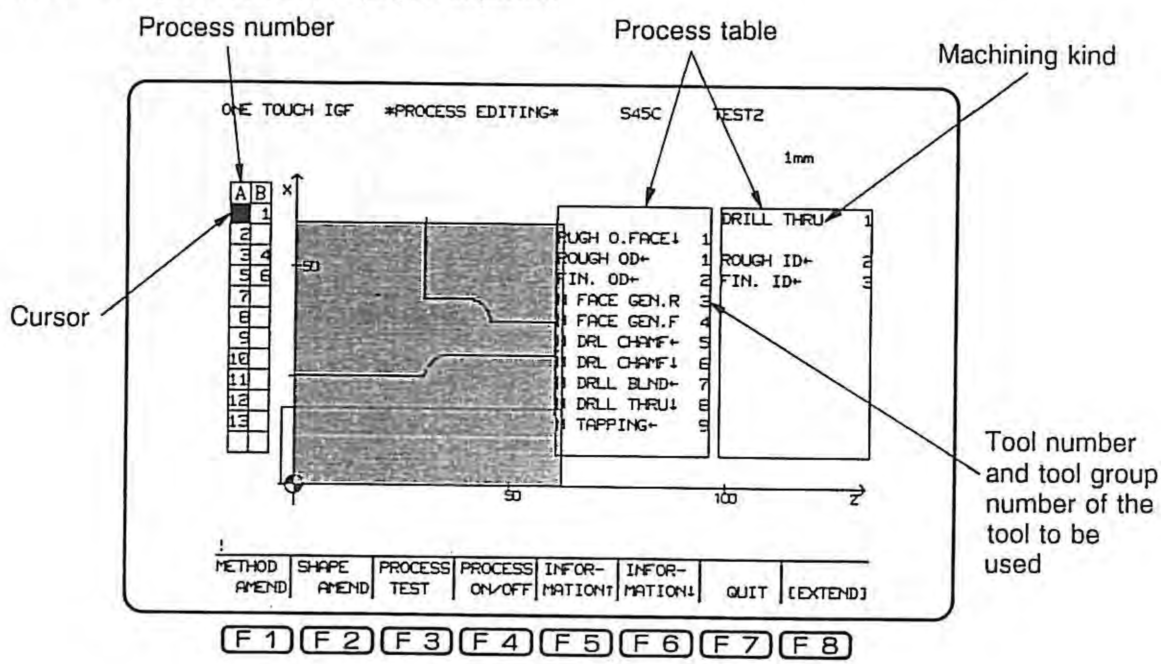
(6) Process list screen

(7) Machining method screen

Each time the function key [F6] (INFORMATION↓) is pressed, the screen will be changed in the sequence of (1)→(2)→(3)→(4)→(5)→(6)→(7)→(1)...

Each time the function key [F5] (INFORMATION↑) is pressed, the screen will be changed in the sequence of (7)→(6)→(5)→(4)→(3)→(2)→(1)...

The process numbers displayed on the left columns are displayed in light blue. With the two-saddle specification, the balance cut process is displayed in white. The process number of the process which is being corrected is displayed in yellow.



Process Check Screen (Turning/special machining screen)

Machining processes are displayed on the process check screen with the process number and the tool number of the tool to be used. If the number of process is large, the process check screen will be divided into several display pages. Switch the display page as needed using the page keys **PAGE ↑** and **PAGE ↓**.

How to check the contents of the process is summarized below.

1) Move the cursor to the process number of the process be corrected on the screen, using the cursor control keys ↑, ↓, ←, → and the page keys **PAGE ↑** and **PAGE ↓**.

2) Press the function key [F5] (INFORMATION ↑) or [F6] (INFORMATION ↓) to display the required process check screen.

   Turning/special machining screen, multi-machining face shape screen, multi-machining side shape screen

   → The finished shape (blue) is displayed on the blank material shape (yellow). At the same time, the finished shape created in other processes is displayed.

   PROCESS (cycle time for each individual process) screen, TOTAL TM screen

   → Cutting time (blue) and non-cutting time (white) are displayed for all processes.

   Process list screen

   → Cutting tools and cutting conditions for all machining processes are displayed.

   Machining method screen

   → Machining method of the cursor located process is displayed.

3) Advance to the next step such as process editing.

Process Check Screens:

In the process editing mode, screens on which the contents of the IGF data created by the one-touch IGF can be checked are available.

(1) Turning/special machining shape screen

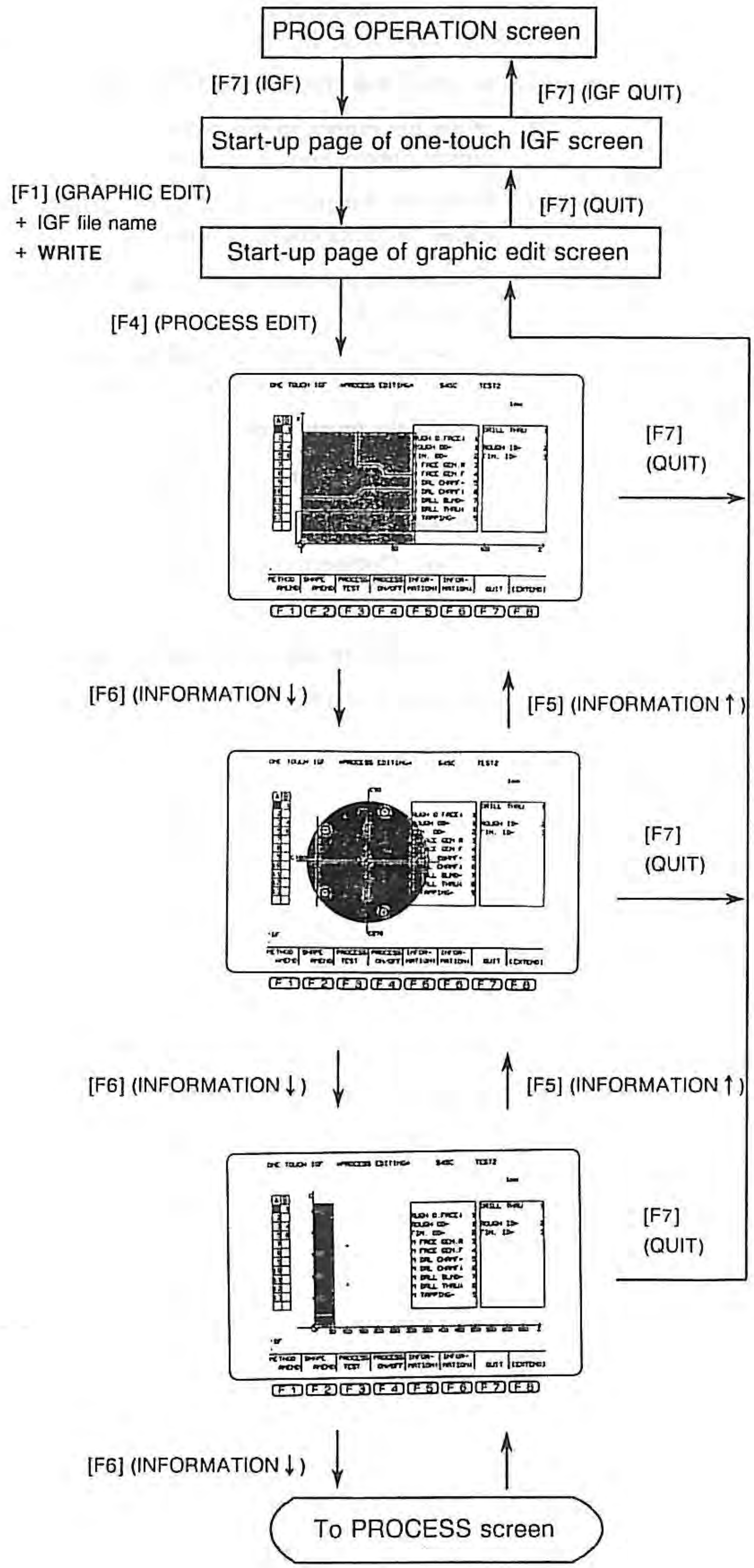
The finished shape (blank material shape) created in the graphic edit mode can be checked.

(2) Multi-machining face shape screen

The finished shape (face) related in the graphic edit mode can be checked.
(This screen is not available unless the multi-machining specification is selected.)

(3) Multi-machining side shape screen

The finished shape (side) created in the graphic edit mode can be checked.
(This screen is not available unless the multi-machining specification is selected.)



(4) PROCESS screen

The cycle time of each process created in the graphic edit mode can be checked.
(The graph is not displayed unless process test is executed.)

(5) TOTAL TM screen

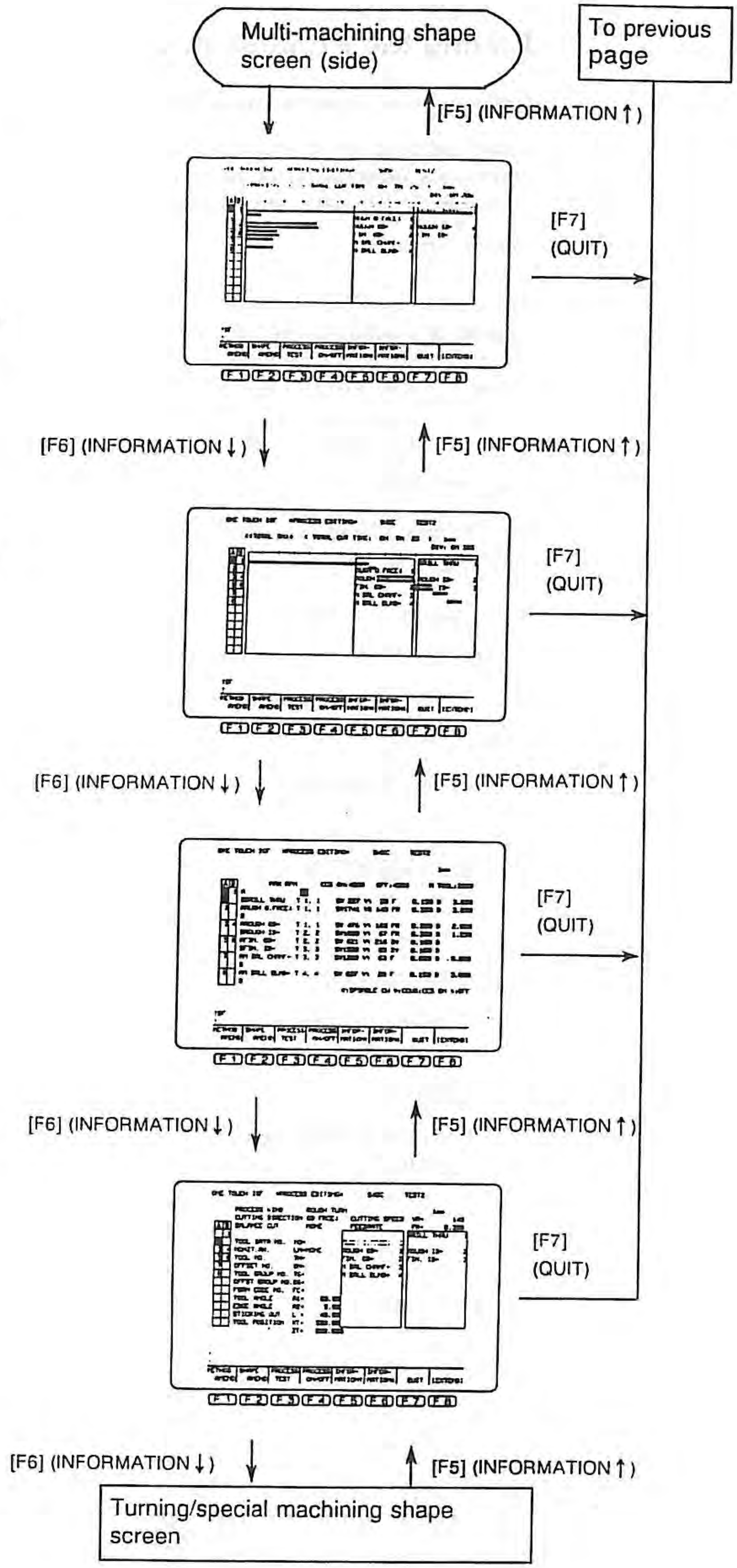
The total of cycle time of all processes in the IGF data can be checked.
(The graph is not displayed unless process test is executed.)

(6) Process list screen

Cutting tools and cutting conditions for all machining processes are displayed.

(7) Machining method screen

The machining method (process kind, cutting tool, cutting conditions) created in the graphic edit mode can be checked.



3. Editing the Process in Unit

Editing in units of process is explained hereafter.

Using this function, operations such as change of process sequence, copy or deletion of process, continuous designation of process (with the two-saddle specification), or registration and call of user-specific IGF data, are possible.

Screen Transfer:

The screen on which editing in units of process is executed (process check screen) is called from the PROG OPERATION screen as diagramed to the right.

[F3] (INSERT)

Inserts a blank process.

[F4] (DELETE)

Deletes a process.

[F6] (EXCHANGE)

Replaces a process.

[F5] (COPY)

Copies a process.

[F3] (USP RECORD)

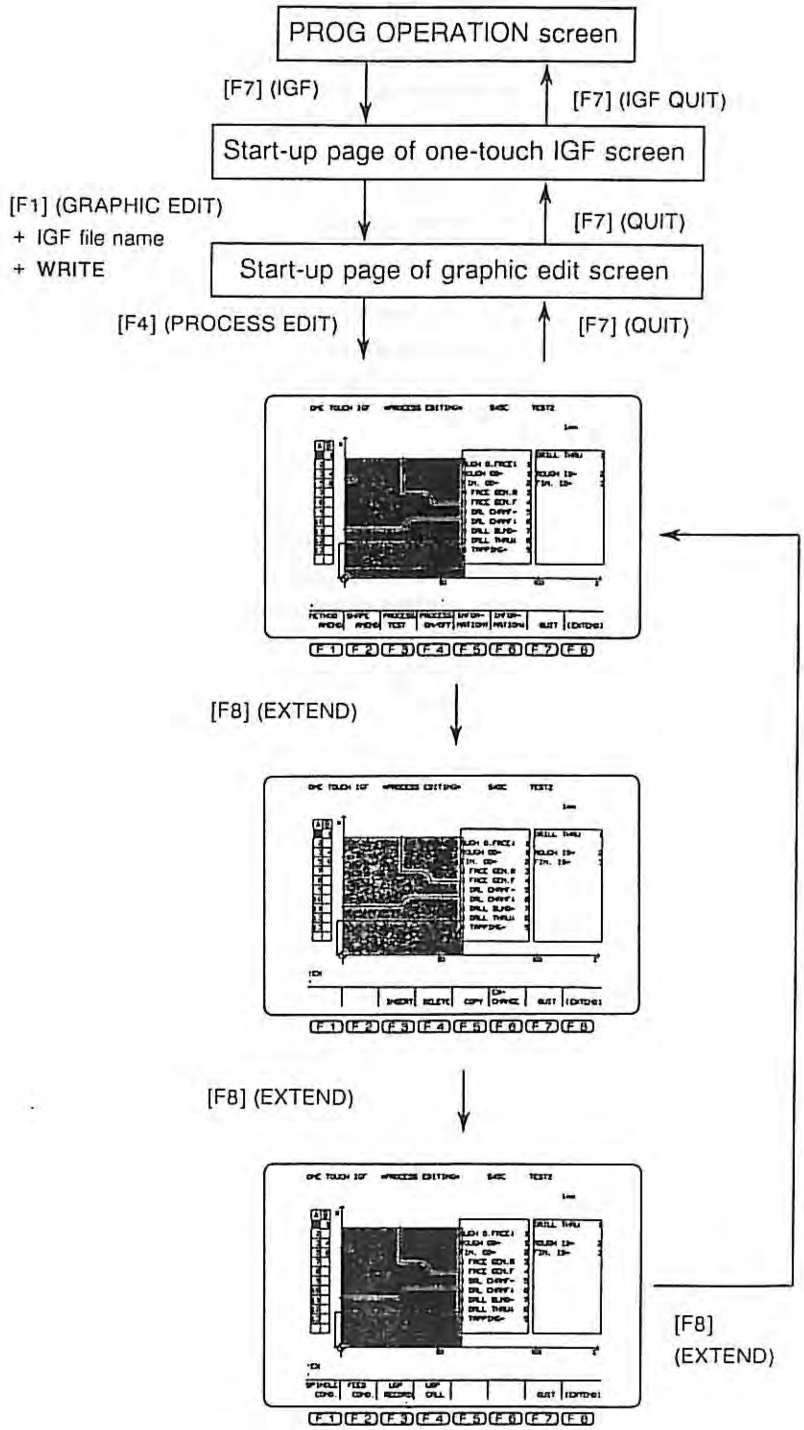
Registers the user-specific IGF file.

[F4] (USP CALL)

Calls the user-specific IGF file.

[F7] (QUIT)

Ends the process edit operation.



(1) INSERT

a) Function

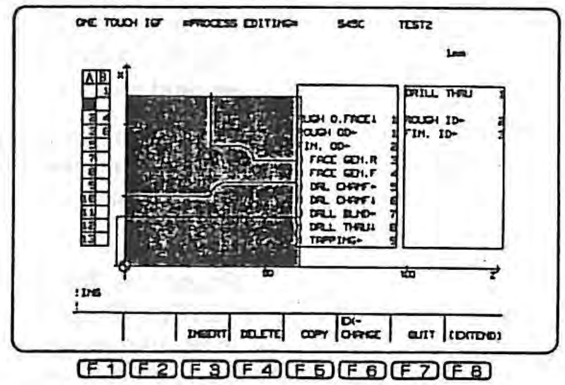
The processes following the designated process are shifted downward by one, leaving the designated process blank.

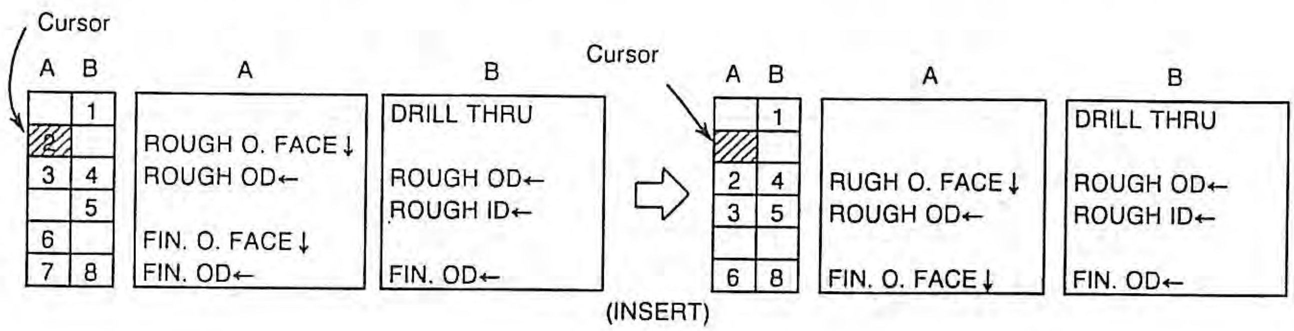
b) How to Designate

Move the cursor to the process number before which the blank process is inserted using the cursor control keys ↑, ↓, →, and ←.

Press the function key [F8] (EXTEND) until the function "INSERT" is assigned to the function key [F3].

Press the function key [F3] (INSERT). The processes following the cursor-located column shift downward, and the cursor-located column is left blank.





Note 1: *The same procedure applies to one-saddle specification models.*

Note 2: *In the case of two-saddle specification model with the balance cut function activated, the position of balance cut is disordered when process insertion operation is executed before the balance cut process. Therefore, it is recommended to insert a blank process also for the other turret. When a part program is created or process test is executed with the disordered balance cut position, each individual process is recognized as an independent process.*

(2) DELETE

a) Function

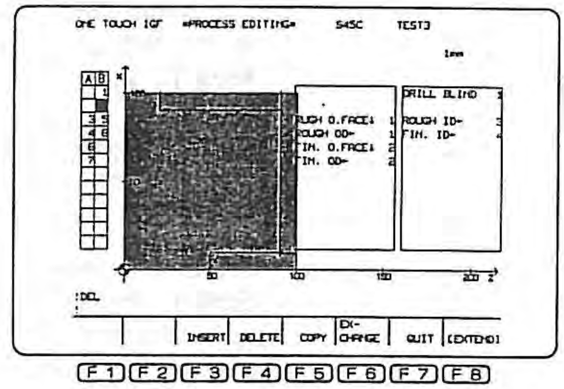
The process designated is deleted and the processes follow are shifted upward to fill the deleted process. This function is used to delete the unnecessary process.

b) How to Designate

Move the cursor to the process number of the process to be deleted using the cursor control keys ↑, ↓, →, and ←.

Press the function key [F8] (EXTEND) until the function "DELETE" is assigned to the function key [F4].

Press the function key [F4] (DELETE). The cursor-located process is deleted and the processes that follow shift upward.



Cursor

A	B	A	B
	1		DRILL THRU
			ROUGH ENDMILL←
3		ROUGH O. FACE↓	
4	5	ROUGH OD←	ROUGH OD←
	6		ROUGH ID←
7		FIN. O. FACE↓	

⇨

Cursor

A	B	A	B
	1		DRILL THRU
3	5	RUGH O. FACE↓	ROUGH OD←
4	6	ROUGH OD←	ROUGH ID←
7	8	FIN. O. FACE↓	FIN. OD←

(DELETE)

Note 1: *The same procedure applies to one-saddle specification models.*

Note 2: *In the case of two-saddle specification model with the balance cut function activated, the position of balance cut is disordered when process insertion operation is executed before the balance cut process (the 4th and 5th processes in figure above). Therefore, it is recommended to insert a blank process to the column of the deleted process. Even when blank columns are generated for both A- and B-turrets due to the insertion of blank process, it is no problem as the one-touch IGF will ignore these columns at the completion of process edit.*

Note 3: *If deletion is attempted with the cursor located at the blank column, the processes that follow will also be shifted upward.*

(3) EXCHANGE

a) Function

The cursor-located process is replaced with the process whose process number is designated.

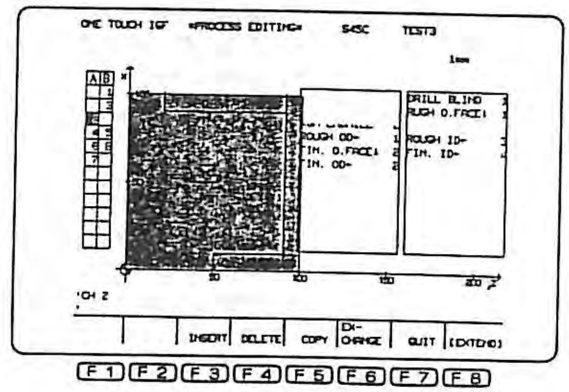
This function is used to replace the process of A-turret with that of B-turret, or to change the sequence of processes.

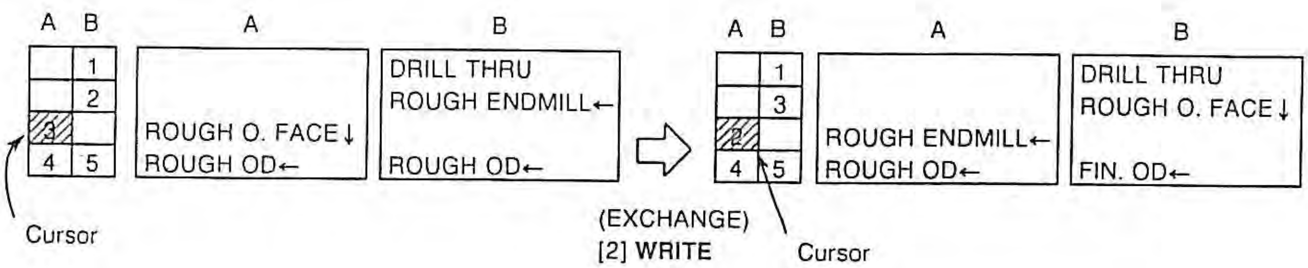
b) How to Designate

Move the cursor to the process number of the process to be replaced using the cursor control keys ↑, ↓, →, and ←.

Press the function key [F8] (EXTEND) until the function "EXCHANGE" is assigned to the function key [F6].

Press the function key [F6] (EXCHANGE), key in the process number of the process which is replaced with the cursor-located process, and press the **WRITE** key. The cursor-located process is replaced with process designated.





Note 1: *The same procedure applies to one-saddle specification models.*

Note 2: *If a process number not present in the process table is designated, replacement operation is not executed.*

Note 3: *The process is replaced without changing the tool number of the tools, tool offset number, or machined shape in that process. Therefore, it might be necessary to correct them depending on the process to be replaced by.*

(4) COPY

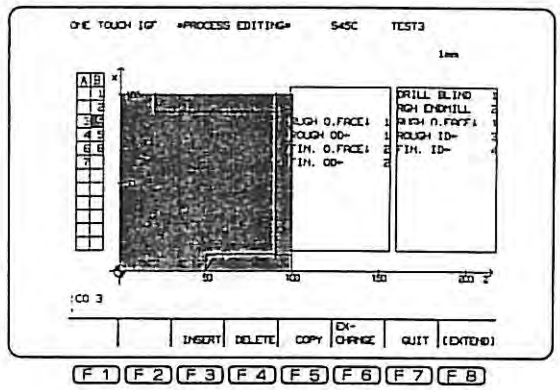
a) Function

The process data of the designated process is copied to the cursor-located blank column.

This function is used to create a same or similar process from the currently existing process.

b) How to Designate

Move the cursor to the blank column where the designated process is to be copied using the cursor control keys ↑, ↓, →, and ←.

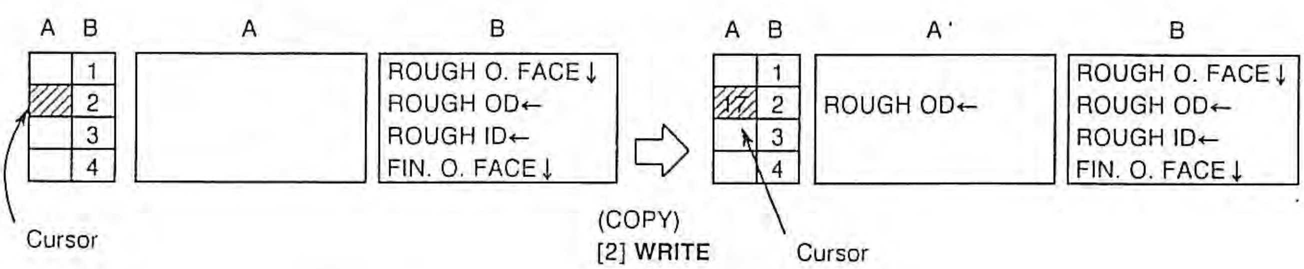


Press the function key [F8] (EXTEND) until the function "COPY" is assigned to the function key [F5].

Press the function key [F5] (COPY), key in the process to be copied, and press the **WRITE** key. The designated process is copied to the cursor-located blank column.

The process number calculated in the formula below is assigned to the newly created process.

Process number =
Largest process number registered + 1



Note 1: *The same procedure applies to one-saddle specification models.*

Note 2: *The copy function is not effective unless the cursor is located at the blank column.*

Note 3: *When the maximum process number (96) is used, the smallest number of the unused process numbers is assigned to the newly created process.*

Note 4: *When the process is copied, the tool number is also copied. This means different tools could have the same tool number in the case of two-saddle specification models.*

(5) USP RECORD/USP CALL

a) Function

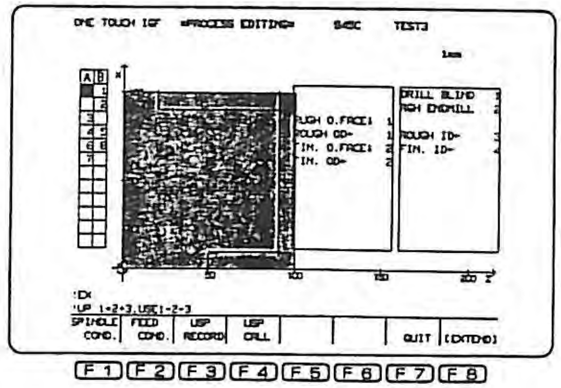
A part of the IGF data created in the automatic process determination mode or process edit mode is saved (registered) so that it can be called when creating another IGF file. This function is used to create a user-specific IGF file from the existing IGF file.

b) How to Designate

i) To register or call the user-specific file (USP), it is necessary to assign a file name to the user-specific file. Assign a file name while observing the rules below.

The extension name (following ".") is "USP".

The main file name must begin with an alphabetic character followed by up to 15 alphanumeric characters including a hyphen.



Example:

RAKURAKU.USP ○

RH250-DT200R-MTX.USP ○

Main file name Extension name

AT775-CC680K-TT573P.USP ×

(The number of characters in the main file name exceeds 16.)

777RAKURAKU.USP ×

(The main file name does not begin with an alphabetic character.)

RAKURAKU.USS ×

(The extension name is not USP.)

ii) USP RECORD

To register a user-specific file (USP), follow the procedure below.

1) Press the function key [F3] (USP RECORD).
2) Key in the process to be registered.
3) Press the "," key.
4) Key in the sector device name, main file name, "." , and extension name.
5) Press the **WRITE** key.
6) This completes user-specific file (USP) registration.

Note 1: *If the number of process to be registered is plural, combine the processes using "+".*

Example:

To register process numbers 1, 7, 11, 12, and 13, designate as follows:

1+7+11+12+13

Note 2: *Sector device name refers to the place where the user-specific file is saved.*

Example:

FD0: (floppy disk)

BB1: (bubble memory)

Note 3: *When the designation of sector device name is omitted, the user-specific file will be saved in BB1:.*

Note 4: *When the designation of main file name is omitted, the main file name "1" will be automatically assigned.*

Note 5: *Entry of "." and extension name can be omitted.*

Example:

Key operations to save the process numbers 3, 4, 7, and 8 to the floppy disk under the user-specific file name USE3-4-7-8.USP:

[F3] + (3+4+7+8) + FD0: + USP3-4-7-8 + ***WRITE***

Note 6: *When an extension name other than "USP" is designated, user-specific file registration is disabled.*

Note 7: *If an attempt to register the USP file is made when a file having the same name as the USP file is already present in the designated sector device, the following message will be displayed.*

file exist overwrite? (Y/N) _

Key in [Y] and press the ***WRITE*** *key to erase the old USP file and to register the new USP file.*

Key in [N] and press the ***WRITE*** *key to cancel registration of the new USP file.*

Note 8: *In the case of two-saddle specification models, when balance cut process is present in the processes to be registered and yet only the process of one turret present, this process is recognized as an independent process.*

Note 9: *With the in-process gauging specification when the gauging process is present yet the process for which this gauging process is defined is not present, an error occurs and the gauging process is not registered as a user-specific file.*

iii) USP CALL

To call a user-specific file (USP), follow the procedure below.

1) Press the function key [F4] (USP CALL).
2) Key in the sector device name, main file name, "." , and extension name.
3) Press the **WRITE** key.
4) This calls the user-specific file (USP).

Note 1: *Sector device name refers to the place where the user-specific file is saved.*

Example:

FD0: *(floppy disk)*

BB1: *(bubble memory)*

Note 2: *When the designation of sector device name is omitted, the user-specific file will be called from BB1:.*

Note 3: *When the designation of main file name is omitted, the main file name "1" will be automatically assigned.*

Note 4: *Entry of "." and extension name can be omitted.*

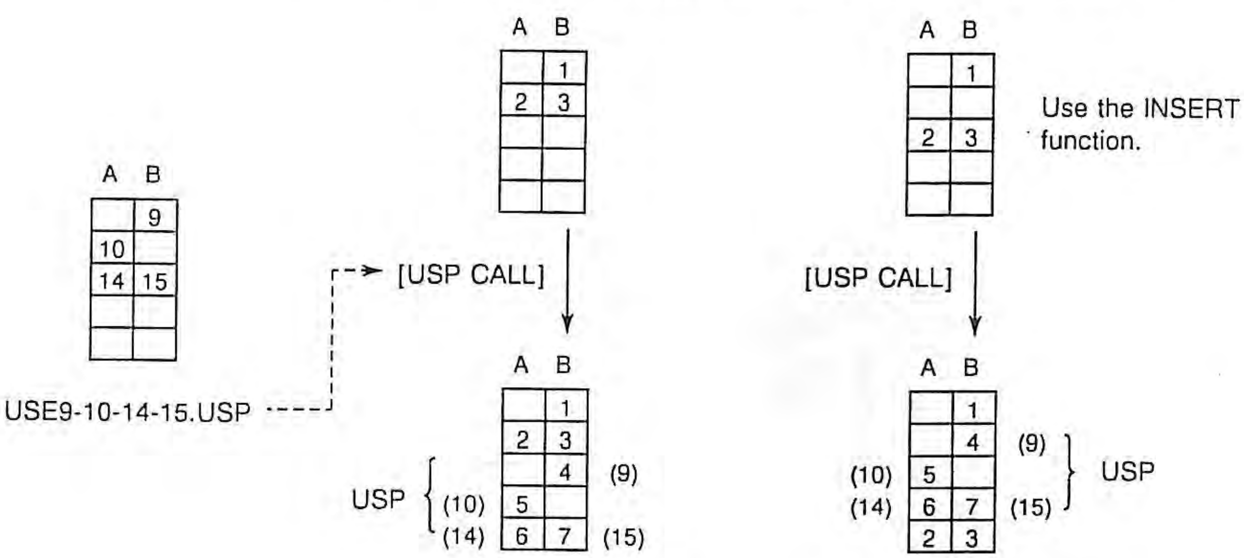
Example:

Key operations to call the USP file "USE3-4-7-8-.USP" from BB1.

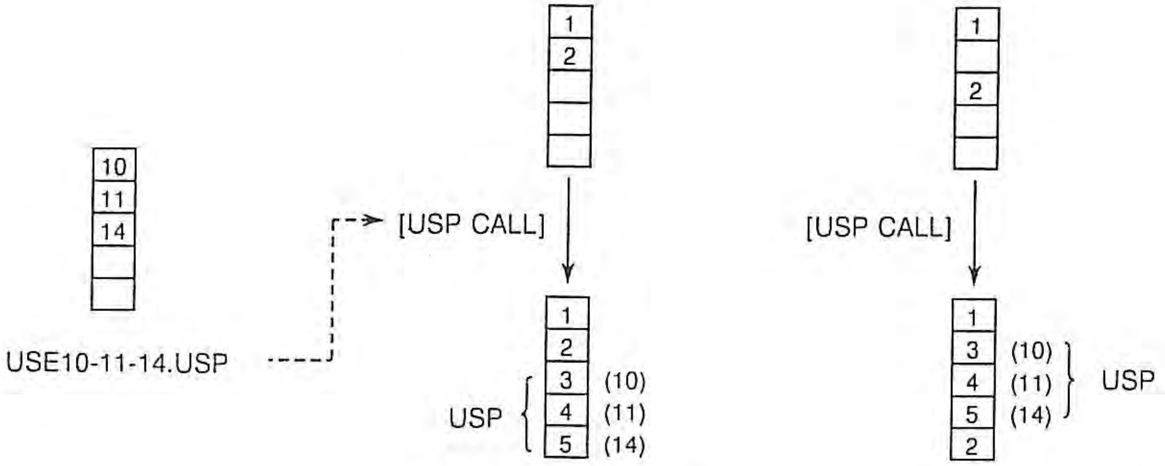
[F4] + USE3-4-7-8 + WRITE

The user-specific file called is registered to the first blank process in the IGF file. The process number calculated in the formula below is assigned to the user-specific file:

Number of processes already registered in the IGF file + 1



Numbers in ( ) are process numbers within the USP file.



Numbers in ( ) are process numbers within the USP file.

Note 5: *Keying in "*" or "?" instead of main file name will display the USP FILE SELECTION screen, and the desired USP file can be selected from the USP file names displayed on this screen.*

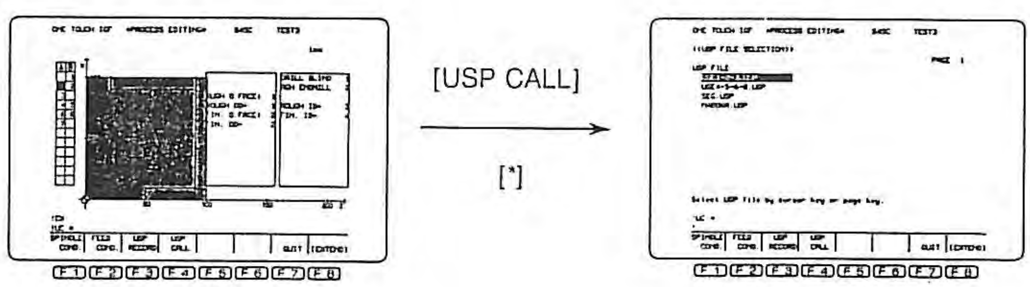
Example:

To call a USP file stored in the floppy disk

Press the function key [F4] (USP CALL), key in the device name (FD0:) and "", and press the* ***WRITE*** *key.*

The USP FILE SELECTION screen will be displayed. Search for the USP file name to call using the page keys ***PAGE*** ↑ *and* ***PAGE*** ↓.

Move the cursor (reverse display) to the USP file name to be selected using the cursor control keys ↑ and ↓, and press the ***WRITE*** *key. This selects the desired USP file.*



Note 6: *When an extension name other than "USP" is designated, the user-specific file cannot be called.*

Note 7: *When the maximum process number (96) is used, the smallest number of the unused process numbers is assigned to the USP file.*

Note 8: *"?" can be used in the main file name as one character.*

Example:

To call one of the USP files beginning with "USE" followed four alphanumeric characters in the bubble memory

$$[F4] + BB1: + \underbrace{[U][S][E][?][?][?][?]}_{\text{7 characters}} + WRITE$$

This displays the USP file name(s) meeting the conditions.

(6) **Designation of Continuous Execution of Process** (two-saddle specification)

a) Function

This function is used to execute a single process continuously on one turret while executing various processes on the other turret.

b) How to Designate

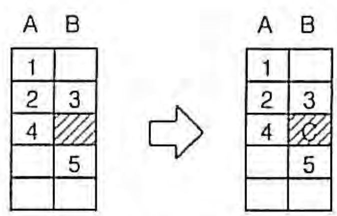
Move the cursor to the blank column following the process number to be continuously executed.

Key in [C] and press the **WRITE** key. "C" will be displayed on the cursor-located column indicating continuous execution of the process.

Example: To execute the 2nd and 4th processes of the A-turret and the 3rd process of the B-turret continuously

Key in "C" to the blank column under the 3rd process.

The 2nd and 3rd processes are started simultaneously, and after the completion of the 2nd process, the execution of the 4th process will be started on the A-turret.



[C] **WRITE**

Note: If the continuous designation of process is designated when the cursor (reverse display) is not blank, an error occurs and "C" is not set.

(7) **Simultaneous Machining by Two Saddles**

With the one-touch IGF, whether or not the two-saddle simultaneous machining process is automatically determined can be set by the one-touch IGF integer parameter.

One-touch IGF integer parameter No. 24 AUTO COMBINED 4-AXIS PROGRAMMING (A + B).

Setting:

0 . . . . Simultaneous machining process not present

1 . . . . Simultaneous machining process present (default)

(8) **PROCESS ON/OFF**

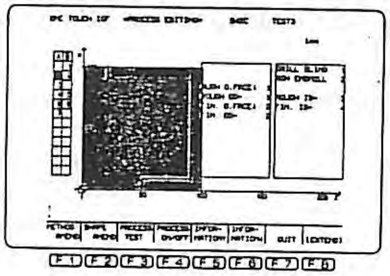
a) Function

The display of the process table indicating process names for each process (table at the right section of the screen) is turned on and off.

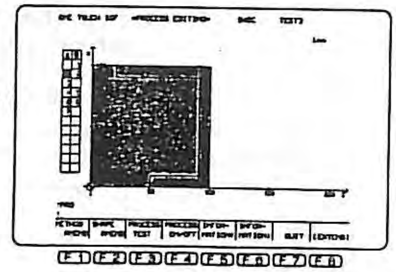
This function is used to display the portion of the workpiece which is behind the process table or to check the contents of process table.

b) How to Designate

Press the function key [F4] (PROCESS ON/OFF). The process table is erased. To display the process table, press the function key [F4] (PROCESS ON/OFF). In other words, each time the function key [F4] (PROCESS ON/OFF) is pressed, the display of process table is turned on and off.



[F4]
(PROCESS ON/OFF)



4. Editing the Machining Method (turning/special machining)

The procedure to edit the machining method for each process is explained. Machining method includes machining kind, tool data of cutting tools to be used, and cutting conditions.

(1) Selection of Process to be Edited

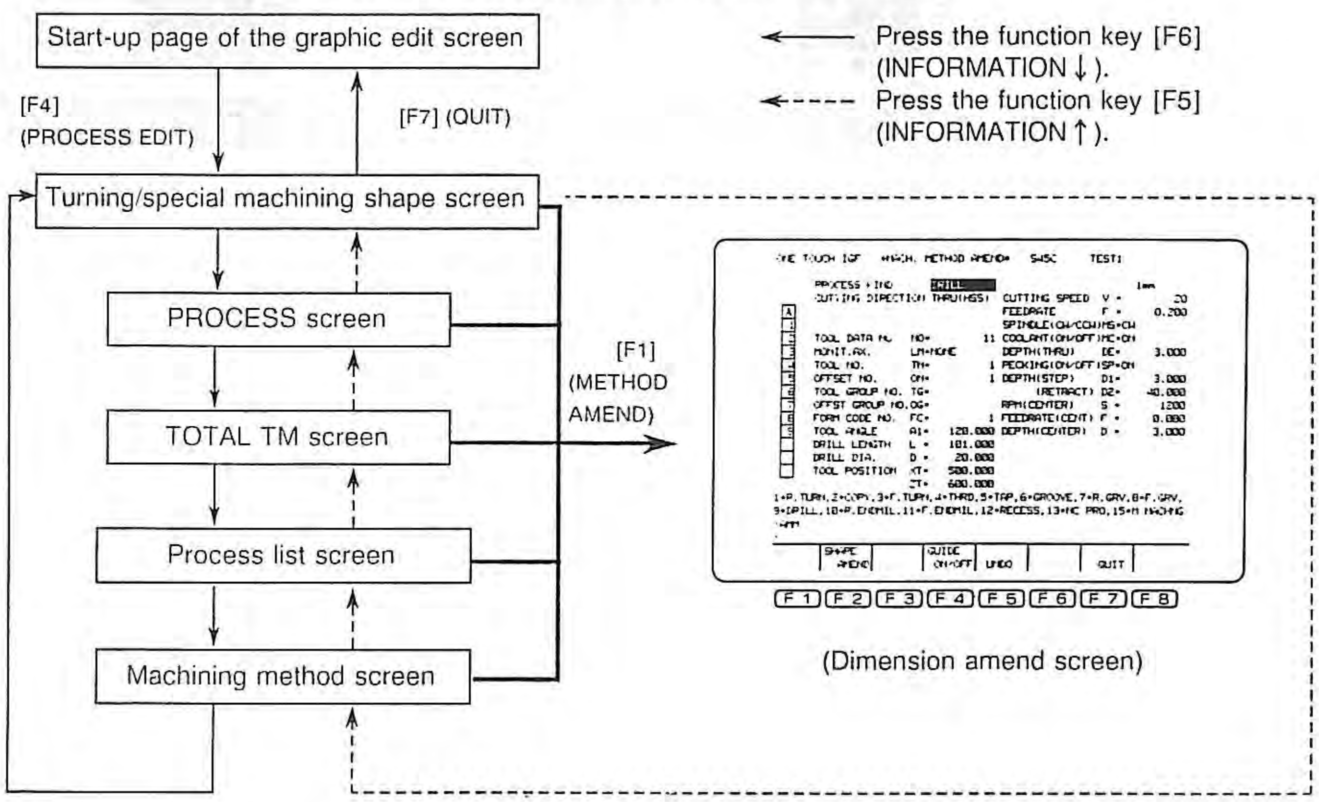
To select the process to be edited, move the cursor to the desired process.

a) How to Designate

Select the display page on which the desired process is present using the page keys **PAGE↑** and **PAGE↓**. Move the cursor to the process number of the process to be edited using the cursor control keys ↑, ↓, ←, and →. To newly make a process, move the cursor to the blank column where no process number is entered (blank process). When the cursor is located, the name of machining kind flickers.

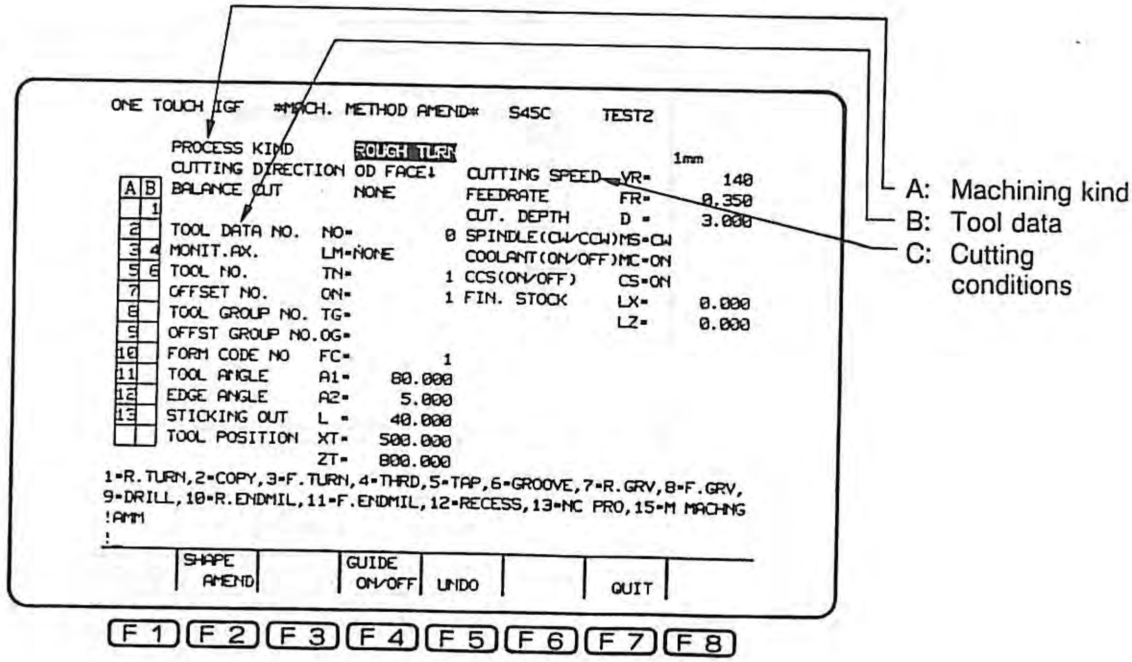
Press the function key [F1] (METHOD AMEND). The MACH. METHOD AMEND screen will be displayed.

In case no blank process is available, use the INSERT function.



(2) **Data Setting**

On the MACH. METHOD AMEND screen, PROCESS KIND (machining kind), TOOL DATA NO. (tool data), and cutting conditions are changed.



PROCESS KIND

a) Function

Machining kind is changed. For example, single groove is changed to wide groove, or OD← is changed to OD→.

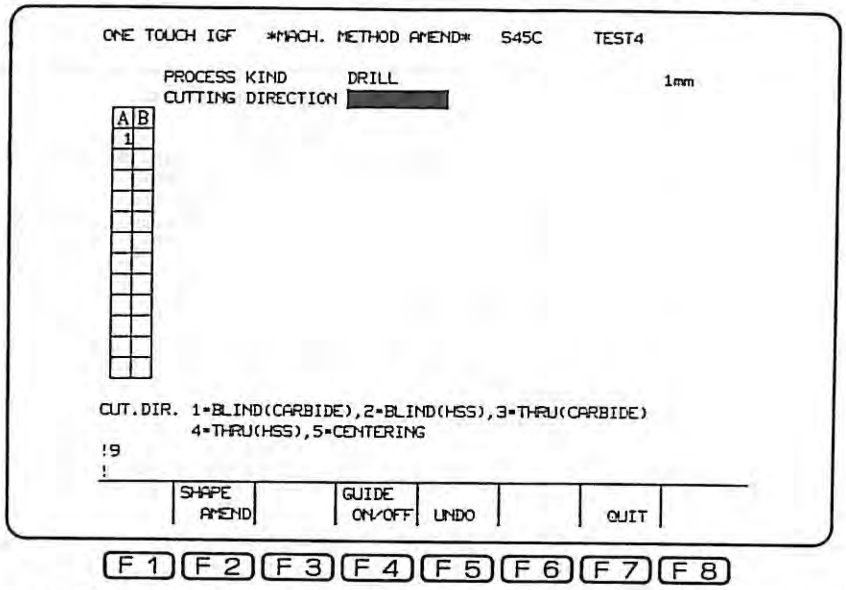
b) Data to be Designated

PROCESS KIND
CUTTING DIRECTION
BALANCE CUT

c) How to Designate

Move the cursor to the data column of the input item to be changed using the cursor control keys ↑, ↓, ←, and →. As the prompt is displayed at the bottom of the screen in yellow characters, enter the data accordingly.

Example: When the cursor is located to the data column of CUTTING DIRECTION, the prompt "**1 = BLIND(CARBIDE), 2 = BLIND(HSS), 3 = THRU(CARBIDE), 4 = THRU(HSS), 5 = CENTERING**" is displayed. Key in [3] and press the **WRITE** key. "3 = THRU(CARBIDE)" is selected for CUTTING DIRECTION.



Note 1: *When the data for PROCESS KIND is changed, the data (TOOL DATA NO., cutting conditions, finished shape, blank material shape) excluding the data which is automatically determined is cleared.*

Note 2: *When the data for CUTTING DIRECTION is changed, the data excluding the data which is automatically determined and that for PROCESS KIND is cleared.*

Note 3: *When the data for PROCESS KIND and CUTTING DIRECTION has been changed mistakenly, press the function key [F5] (UNDO). The CRT will be returned to the process check screen which has been displayed prior to the MACH. METHOD AMEND screen.*

Note 4: *When the function key [F4] (GUIDE ON/OFF) is pressed while the cursor (reverse display) is located either at PROCESS KIND or CUTTING DIRECTION, the guide drawing will be displayed. Each time the function key [F4] (GUIDE ON/OFF) is pressed, another guide drawing is displayed. The display of guide drawing disappears from the screen after all guide drawings have been displayed.*

Note 5: *For the combination between PROCESS KIND and CUTTING DIRECTION, refer to Section 11.*

Tool Data of Cutting Tools

a) Function

Tool data for each machining process (tool number, offset number, shape data, etc.) is changed.

b) Data to be Designated

TOOL DATA NO. NO

COMMAND DATA
  TOOL NO. TN
  OFFSET NO. ON (or O1, O2)

TOOL EDGE DATA
  FORM CODE NO. FC
  TOOL ANGLE A1
  TOOL ANGLE A2
  DRILL DIA. D

TOOL INDEX POSIT. XT, ZT

c) How to Designate

Move the cursor to the data column of input items using the cursor control keys ↑, ↓, ←, and →, and key in the numerical value.

With the one-touch IGF, machining process and the tools to be used are automatically determined. Register the tool data of the tools in hand on the TOOL DATA screens. The one-touch IGF system will automatically look for and find out the necessary tool. The tool data of the cutting tool used in each process is displayed on the screen.

When the necessary tool is not found in the tool data, the one-touch IGF will automatically set "0" for TOOL DATA NO., indicating that the tool is a non-registered tool.

If no data has been set for the following input items in the TOOL DATA setting mode, the one-touch IGF will automatically set the optimal data for them.

TOOL NO. TN
OFFSET NO. ON
NOSE RADIUS R (The same number as OFFSET NO. ON is set.)
STICKING OUT/HOLDER LENGTH/DRILL LENGTH L
HOLDER DIA. D (for ID turning tools only)

In other words, when the data has been set for the above input items in the TOOL DATA setting mode, the same data as entered is used in the MACH. METHOD AMEND mode. Therefore, it is a great convenience to set the tool data of the cutting tools used with high frequency. Note however that the cutting tool may cause an interference in process test when the data is set for STICKING OUT or HOLDER LENGTH.

Note 1: *When the function key [F4] (GUIDE ON/OFF) is pressed while the cursor (reverse display) is located either at TOOL DATA NO. or FORM CODE NO., the guide drawing will be displayed. Each time the function key [F4] (GUIDE ON/OFF) is pressed, another guide drawing is displayed. The display of guide drawing disappears from the screen after all guide drawings have been displayed.*

Note 2: *The guide drawing for TOOL DATA NO. is displayed as indicated below. For TOOL DATA NO., either "0" or the tool data number of the cutting tools displayed on the guide drawings can be set.*

No.	A1	A2	L	
2	80,000	5,000	20,000	
46	80,000	5,000	50,000	
102	75,000	5,000	50,000	

Tool numbers of the tools whose data is set in the TOOL DATA setting mode and can be used.

TOOL EDGE DATA
(FORM CODE NO., TOOL ANGLE, STICKING OUT)

Note 3: *If no data has been set for the following input items in the TOOL DATA setting mode, the one-touch IGF will automatically set the appropriate data for them.*

TOOL DATA NO.
OFFSET NO.
TURRET NO.

There may be cases in which a tool number larger than the turret position number is set. Check the data set for TOOL DATA NO. and OFFSET NO. on the process check screen and MACH. METHOD AMEND screen and correct the data when needed.

Note 4: *It is not possible to display the shape of the special-shaped tool. Set "0" for FORM CODE NO. FC when a machining process in which a special-shaped tool is used is created. In this case, only the tool path will be displayed on the process test screen.*

Note 5: *For details of the tool data, refer to 3. "Items to be Set as Tool Data" in Section 3.*

Cutting Conditions

a) Function

Cutting conditions in each process (CUTTING SPEED, FEEDRATE, CUT. DEPTH) are changed.

b) Data to be Designated

Refer to Table 11-1.

c) How to Designate

Move the cursor (reverse display) to the data column of the input item using the cursor control keys ↑, ↓, ← and →, key in the value and press the WRITE key.

Table 11-1

PROCESS KIND	Data to be Designated	
R.TURN COPY	CUTTING SPEED	VR
	FEEDRATE	FR
	CUT. DEPTH	D
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	CCS (ON/OFF)	CS
	FIN. STOCK	LX*, LZ
	*LX: diametrical value	
F.TURN	CUTTING SPEED	VF
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	CCS (ON/OFF)	CS
	FIN. STOCK	LX, LZ
	∇ (ROUGH) FEEDRATE	F1
	∇∇ (SEMI FINISH) FEEDRATE	F2
	∇∇∇ (FINISH) FEEDRATE	F3
	∇∇∇∇ (FINE FINISH) FEEDRATE	F4
THRD	CUTTING SPEED	V
	CUT. DEPTH	D
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	PITCH	F
	TPI	J
	LEAD NO.	NT
	FIN. STOCK	L
TAPPING	CUTTING SPEED	V
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	PITCH	F

PROCESS KIND	Data to be Designated	
GROOVE R. GRV	CUTTING SPEED	VR
	FEEDRATE	FR
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	CCS (ON/OFF)	CS
	STEP (ON/OFF)	SP
	CUT. DEPTH (ONE STEP)	D1
	CUT. DEPTH (RETRACT)	D2
	POSSIBLE DEPTH	DL*
	TOOL SHIFT	W*
	FIN. STOCK	LX, LZ*
	*: Designation is necessary only for rough groove cutting.	
F.GRV	CUTTING SPEED	VF
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	CCS (ON/OFF)	CS
	∇ (ROUGH) FEEDRATE	F1
	∇∇ (SEMI FINISH) FEEDRATE	F2
	∇∇∇ (FINISH) FEEDRATE	F3
	∇∇∇∇ (FINE FINISH) FEEDRATE	F4
	FEEDRATE SELECTION	SR
DRILL	CUTTING SPEED	V
	FEEDRATE	F†
	SPINDLE (CW/CCW)	MS†
	COOLANT (ON/OFF)	MC†
	STEP (ON/OFF)	SP
	CUT. DEPTH (ONE STEP)	D1
	†: Designate these data for centering.	

PROCESS KIND	Data to be Designated	
DRILL	CUT. DEPTH (RETRACT)	D2
	SPEED (CENTERING)	S†
	FEEDRATE (CENTERING)	F
	CUT. DEPTH (CENTERING)	D
	CUT.DEPTH (THRU)	DE*
	†: Designate these data for centering.	
	*: The value of the dimension parameter No. 43 is set.	
R. ENDMIL	CUTTING FEEDRATE	VR
	FEEDRATE	FR
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	STEP (ON/OFF)	SP
	CUT. DEPTH (ONE STEP)	D1
	CUT. DEPTH (RETRACT)	D2

PROCESS KIND	Data to be Designated	
F.ENDMIL	CUTTING FEEDRATE	VF
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	STEP (ON/OFF)	SP
	CUT. DEPTH (ONE STEP)	D1
	CUT. DEPTH (RETRACT)	D2
	∇ (ROUGH) FEEDRATE	F1
	$\nabla\nabla$ (SEMI FINISH) FEEDRATE	F2
	$\nabla\nabla\nabla$ (FINISH) FEEDRATE	F3
	$\nabla\nabla\nabla\nabla$ (FINE FINISH) FEEDRATE	F4
	FEEDRATE SELECTION	SR
RECESS	CUTTING FEEDRATE	V
	FEEDRATE	F
	SPINDLE (CW/CCW)	MS
	COOLANT (ON/OFF)	MC
	CCS (ON/OFF)	CS

Precautions When Setting Cutting Conditions

i) GROOVE/R. GRV

CUT. DEPTH (ONE STEP) D1

When cutting is carried out in the step mode, D1 represents the one step depth.

CUT. DEPTH (RETRACT) D2

The depth of at which cutting tool is retracted.

POSSIBLE DEPTH DL

Allowable depth of cut for the cutting tool to be used. The value obtained through (cutting tool width) × (percent parameter No. 26) is preset.

TOOL SHIFT W

Cutting tool shift amount for wide-groove cutting cycle.

The value obtained through (cutting tool width) × (percent parameter No. 7) is preset.

FIN. STOCK LX, LZ (diametrical value)

In the roughing process, the finishing stock designated by LX and LZ is left uncut.

ii) DRILL

CUT. DEPTH (ONE STEP) D1

When cutting is carried out in the step mode, D1 represents the one step depth.

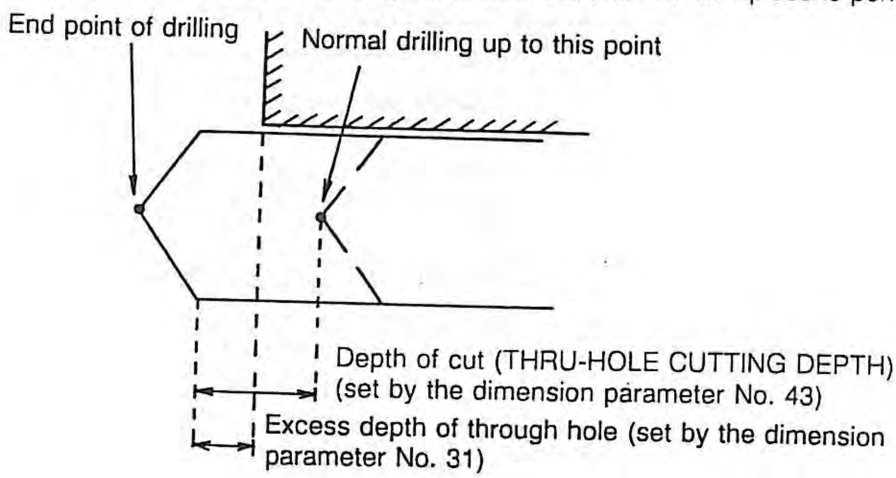
CUT. DEPTH (RETRACT) D2

The depth of at which cutting tool is retracted.

CUT. DEPTH (THRU) DE

The through-hole drilling cycle is executed in the following sequence.

- Normal drilling till just before the workpiece is penetrated
- The drill tool is retracted.
- The drill tool is again infed to the point near the end of first drilled depth at a rapid feedrate.
- The drill is fed at the through-hole feedrate until the workpiece is penetrated.



Feedrate used to drill a through hole can be set by the percent parameter No. 12.

Usually, drilling is carried out after centering. If centering is not needed, enter "0" for CUT. DEPTH.

Whether or not centering is executed and whether or not the center drilling process is generated can be set by the integer parameter No. 33.

Integer parameter No. 33 (CENTERING TOOL SELECTION)

0 ..... The center drilling process is generated and centering is not executed in the drilling process.

1 ..... The center drilling process is not generated and centering is executed in the drilling process.

(3) **Balance Cut** (two-saddle specification)

With the two-saddle specification, it is possible to cut machining time using the balance cut function. Processes to be machined in the balance cut mode can be easily created by the one-touch IGF. The following processes can be machined in the balance cut mode.

Rough turning
Copying
Finish turning
Rough grooving
Finish grooving

a) To Create Balance Cut Process in the Process Edit Mode

Enter "1 = EXIST" for BALANCE CUT on the MACH. METHOD AMEND screen.

b) To Create Balance Cut Process Using the Automatic Process Determination Function

With the one-touch IGF, it is possible to automatically generate the balance cut process by setting the proper data to the integer parameter No. 25.

Integer parameter No. 25 BALANCED CUTTING

0 ...... Balance cut process is not generated.

1 ...... Balance cut process is generated if the tool which can be used for balance cut is registered in the tool data.

2 ...... The balance cut process for the rough turning process is generated when the tool which can be used for balance cut is registered in the tool data.

3 ...... The balance cut process is generated irrespective of the tool data.

4 ...... The balance cut process for the rough turning process is generated irrespective of the tool data.

Note 1: *If the one-touch IGF is unable to create the process for the other turret because no space is left in the process table when a balance cut process is designated in the PROCESS EDITING mode, the following message will be displayed and the creation of balance cut process will be disabled.*

The other process in BALANCED CUTTING cannot be created.

Note 2: *In case an independent process already created for the A- or B-turret is changed to a balance cut process when a balance cut process is designated in the PROCESS EDITING mode, the following message will be displayed and the MACH. METHOD AMEND screen of the other turret will appear when the function key [F7] (QUIT) is pressed.*

The other process in BALANCED CUTTING was automatically created.

In this case, finished shape, machining kind, and cutting conditions for the process on the other turret are automatically determined. However, since no data is set f or TOOL DATA NO., TOOL NO., OFFSET NO., TOOL GROUP, and OFFSET GROUP, no process can be created. To create a process, enter the data for these input items.

Note 3: *When cutting conditions or shape data in the balance cut process are changed, the process of the other turret is also modified. However, the tool data is not modified.*

Note 4: *When a balance cut process is deleted, the process of the other turret remains as an independent process of that turret.*

Note 5: *When the balance cut process is separated using the "copy" or "replace" function in process editing operation, the positions of the processes disagree. An alarm occurs if process edit completion or process test is attempted in this condition.*

Note 6: *When a balance cut process for ROUGH OD← is automatically created, there may be cases in which an interference of the tool may occur depending on the material shape.*

c) Cutting Conditions for Balance Cut Process

Cutting condition data for the balance cut processes are entered assuming the both processes to be an independent process. Therefore, the cutting conditions for process test or for the part program to be created are determined as indicated below:

Depth of Cut = CUT. DEPTH set on the MACH. METHOD AMEND screen ×
Setting of the percent parameter No. 27

Feedrate = FEEDRATE set on the MACH. METHOD AMEND screen ×
Setting of the percent parameter No. 28

Cutting conditions for the balance cut processes can be checked on the Process list screen. For details, refer to 6. "Process List Display" in this section.

(4) **Simultaneous Four-axis Machining** (Two-saddle Specification)

Cutting condition data for the simultaneous four-axis machining processes are entered assuming each of the processes to be an independent process. Therefore, the cutting conditions for process test or for the part program to be created are determined as indicated below:

CCS (constant peripheral speed) ON/OFF

"OFF" is set. Cutting is executed at a constant rpm.

Spindle speed

For each of the simultaneous four-axis machining processes, the spindle speed is calculated assuming that each of the paired processes is an independent process. And the lower spindle speed of two spindle speeds in the paired processes is selected. The spindle speed for the simultaneous four-axis machining is calculated as indicated below:

Spindle speed = Lower spindle speed of the paired processes ×
Setting of the percent parameter No. 20

Depth of cut

Depth of cut = CUT. DEPTH set on the MACH. METHOD AMEND screen ×
Setting of the percent parameter No. 21

Feedrate in the drilling, center-drilling, or endmilling process

Feedrate = FEEDRATE set on the MACH. METHOD AMEND screen ×
Setting of the percent parameter No. 22

Feedrate in the grooving process

Feedrate = FEEDRATE set on the MACH. METHOD AMEND screen ×
Setting of the percent parameter No. 23

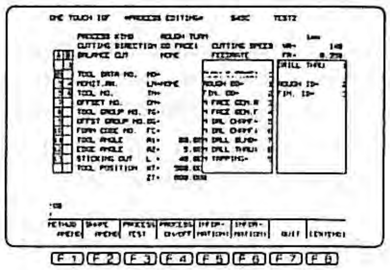
Cutting conditions for the simultaneous four-axis machining (two-saddle specification) can be checked on the process list screen.

For details, refer to 6. "Process List Display" in this section.

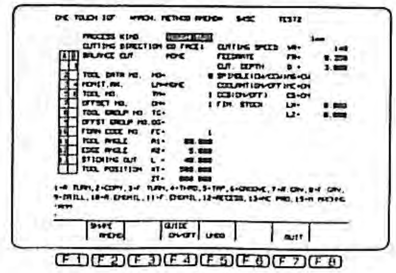
(5) **Deleting the Edited Data**

It is possible to delete the data corrected on the MACH. METHOD AMEND screen, and to return to the PROCESS EDITING screen.

Press the function key [F5] (UNDO).



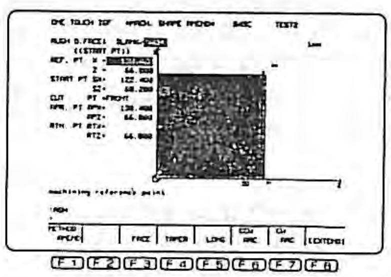
[F5]
(UNDO)



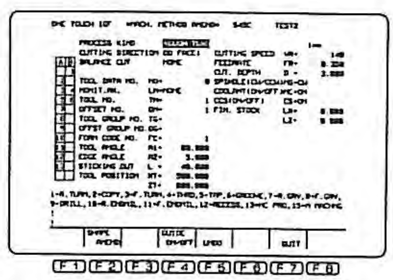
(6) **Switching to the MACH. SHAPE AMEND Screen**

It is possible to switch the CRT from the MACH. METHOD AMEND screen to the MACH. SHAPE AMEND screen.

Press the function key [F2] (SHAPE AMEND). The MACH. SHAPE AMEND screen will be displayed. Correct the data as needed.



[F2]
(SHAPE AMEND)



Note: *There are cases in which switching to the MACH. SHAPE AMEND screen is disabled due to faulty or incorrect data on the MACH. METHOD AMEND screen.*

(7) **Terminating the Machining Method Editing**

Press the function key [F7] (QUIT) after the completion of machining method editing. The CRT will be switched from the MACH. SHAPE AMEND screen to the PROCESS EDITING screen.

Note: *There are cases in which switching to the PROCESS EDITING screen is disabled due to faulty or incorrect data on the MACH. METHOD AMEND or MACH. SHAPE AMEND screen.*

5. Editing the Machining Method (optional: multi-machining specification, contour generating machining specification)

The procedure to edit the machining method for each process is explained. Machining method includes machining kind, tool data of cutting tools to be used, and cutting conditions. Refer to 4. "Editing the Machining Method (turning, special machining)" as editing is executed in the similar manner.

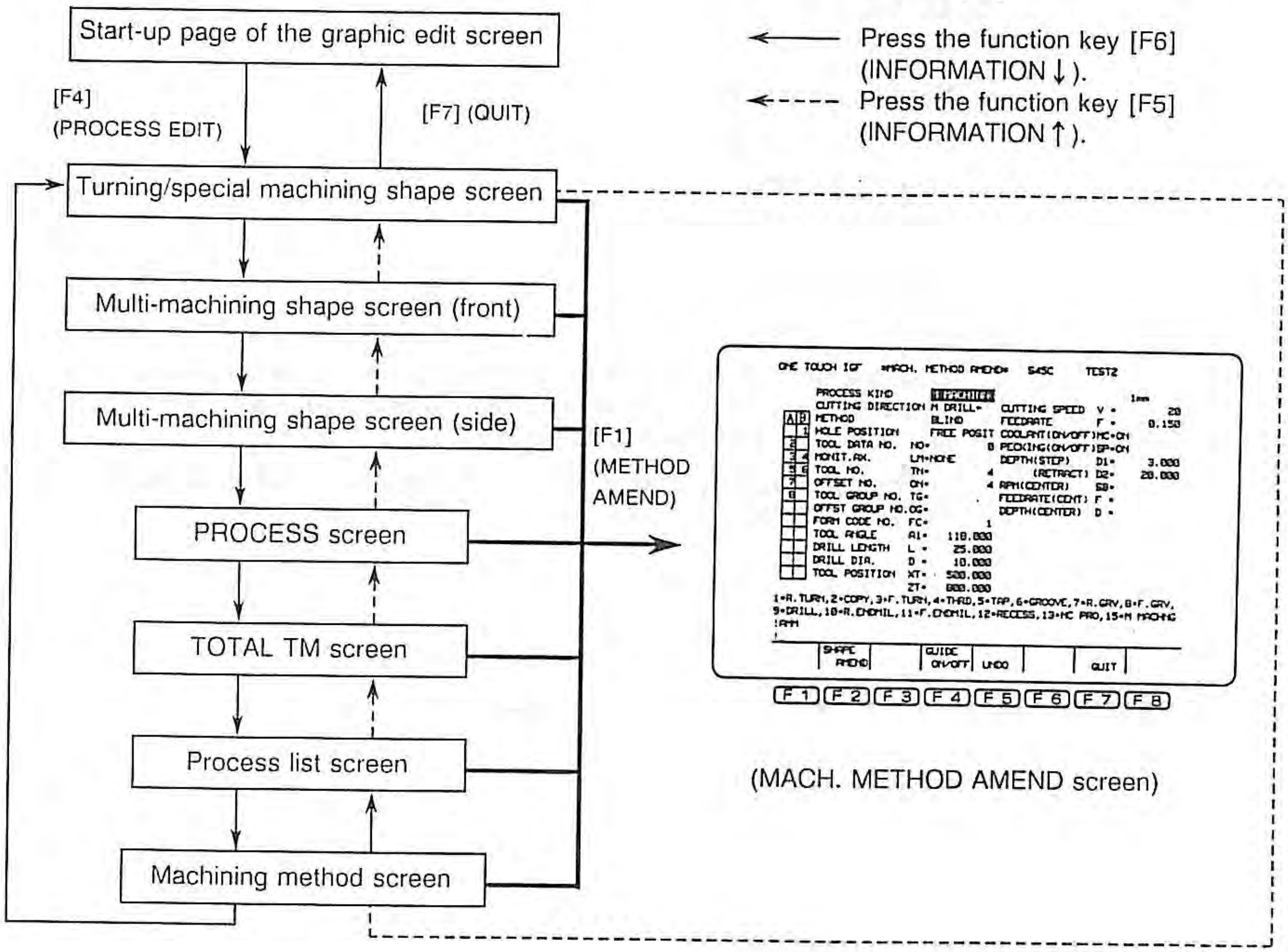
(1) **Selection of Process to be Edited**

To select the process to be edited, move the cursor to the desired process on any of the following screens.

- Machining method screen
- PROCESS screen
- TOTAL TM screen

a) How to Designate

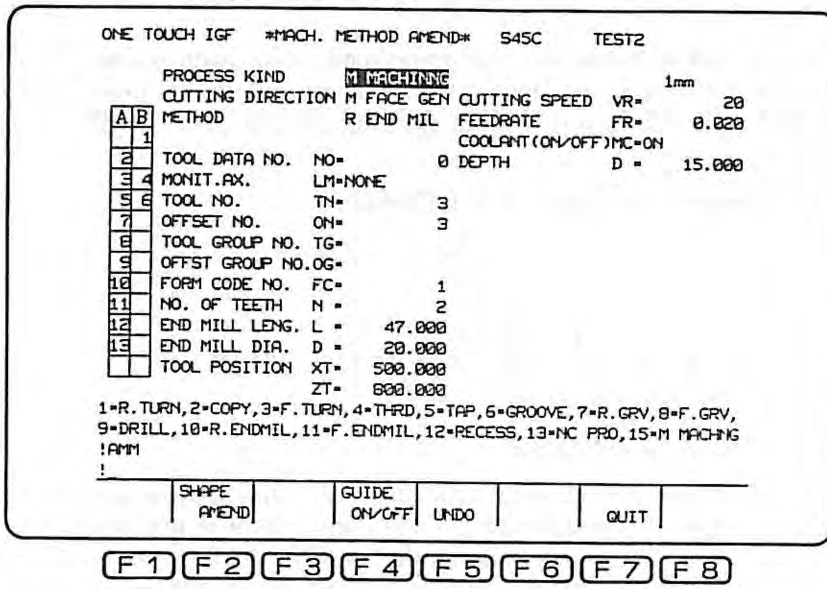
Select the display page on which the desired process is present using the page keys **PAGE ↑** and **PAGE ↓**. Move the cursor to the process number of the process to be edited using the cursor control keys ↑, ↓, ←, and →. Press the function key [F1] (METHOD AMEND). The MACH. METHOD AMEND screen will be displayed.



(MACH. METHOD AMEND screen)

(2) **Data Setting**

On the MACH. METHOD AMEND screen, PROCESS KIND (machining kind), TOOL DATA NO. (tool data), and cutting conditions are changed.



PROCESS KIND

a) Function

Machining kind is changed.

b) Data to be Designated:

PROCESS KIND
CUTTING DIRECTION
METHOD
HOLE POSITION

c) How to Designate

Move the cursor to the data column of the input item to be changed using the cursor control keys ↑, ↓, ←, and →. As the prompt is displayed at the bottom of the screen in yellow characters, enter the data accordingly.

For the combination between PROCESS KIND and CUTTING DIRECTION, METHOD, HOLE POSITION, refer to Section 11.

Tool Data of Cutting Tools

a) Function

Tool data for each machining process (tool number, offset number, shape data, etc.) is changed.

b) Data to be Designated

TOOL DATA NO. NO

COMMAND DATA
TOOL NO. TN
OFFSET NO. ON (or O1, O2)

TOOL EDGE DATA
FORM CODE NO. FC
TAP LENGTH L
BORING LENG TH

TOOL INDEX POSIT. XT, ZT

c) How to Designate

Move the cursor to the data column of input items using the cursor control keys ↑, ↓, ←, →, and key in the numerical value.

With the one-touch IGF, machining process and the tools to be used are automatically determined. Register the tool data of the tools in hand on the TOOL DATA screens. The one-touch IGF system will automatically look for and find out the necessary tool. The tool data of the cutting tool used in each process is displayed on the screen.

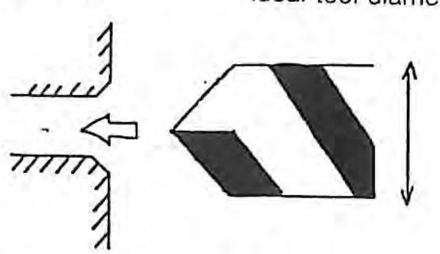
When the necessary tool is not found in the tool data, the one-touch IGF will automatically set "0" for TOOL DATA NO., indicating that the tool is a non-registered tool.

If no data has been set for the following input items in the TOOL DATA setting mode, the one-touch IGF will automatically set the optimal data for them.

TOOL NO. TN
OFFSET NO. ON
DRILL LENGTH
BORING LENGTH

Note: *The M-drill chamfering tool is determined by adding the setting of the dimension parameter No. 61 M MACHINING CHAMFERING ALLOWA. DIA to the data entered for HOLE DIA. and CHF SIZE. The ideal tool diameter for the chamfering process which one-touch IGF automatically sets is calculated as follows:*

Ideal tool diameter = Hole diameter + (chamfer size × 2) + (setting of the dimension parameter No. 61)



Thus, the drill whose diameter D is within the following range is selected for the M-drill chamfering process from the tool data:

(Ideal drill diameter for M-drill chamfering) ≦ D ≦
(Ideal drill diameter for M-drill chamfering) × (setting of the percent parameter No. 52)

This rule also applies to turning or special machining operation.

Cutting Conditions

a) Function

Cutting conditions in each process are changed.

b) Data to be Designated

Refer to Table 11-2.

c) How to Designate

Move the cursor (reverse display) to the data column of the input item using the cursor control keys ↑, ↓, ←, →, key in the value and press the **WRITE** key.

Table 11-2

PROCESS KIND	Data to be Designated	
M DRILL	CUTTING SPEED	V#
	FEEDRATE	F#
	COOLANT (ON/OFF)	MC#†
	STEP (ON/OFF)	SP
	CUT. DEPTH (ONE STEP)	D1
	CUT. DEPTH (RETRACT)	D2
	SPEED (CENTERING)	SB†
	FEEDRATE (CENTERING)	F†
	CUT. DEPTH (CENTERING)	D
	CUT. DEPTH (THRU)	DE*
	#: Designate these data for chamfering.	
	†: Designate these data for centering.	
	*: The value of the dimension parameter No. 60 M MACHINING DRILLING THRU-HOLE CUT. DEPTH is set.	
M BORING	CUTTING SPEED	V
M TAP	PITCH	P
M REAMING	COOLANT (ON/OFF)	NC
M KEYWAY	CUTTING SPEED	V
	FEEDRATE	F
	COOLANT (ON/OFF)	MC
	CUT. DEPTH	D
	FIN. STOCK	L*
	*: Finishing stock of keyway cutting is the value of depth of cut in the final cutting cycle.	

PROCESS KIND	Data to be Designated	
M ENDMILL	CUTTING SPEED	V
	FEEDRATE	F
	COOLANT (ON/OFF)	MC
	STEP (ON/OFF)	SP
	CUT. DEPTH (ONE STEP)	D1
	CUT. DEPTH (RETRACT)	D2
CONTOUR ROUGH	CUTTING SPEED	VR
	FEEDRATE	FR
	COOLANT (ON/OFF)	MC
	CUT. DEPTH	D
CONTOUR FINISH	CUTTING SPEED	VF
	COOLANT (ON/OFF)	MC
	▽ (ROUGH) FEEDRATE	F1
	▽▽ (SEMI FINISH) FEEDRATE	F2
	▽▽▽ (FINISH) FEEDRATE	F3
	▽▽▽▽ (FINE FINISH) FEEDRATE	F4

Precautions When Setting Cutting Conditions

i) M DRILL

CUT. DEPTH (ONE STEP) D1

When cutting is carried out in the step mode, D1 represents the one step depth.

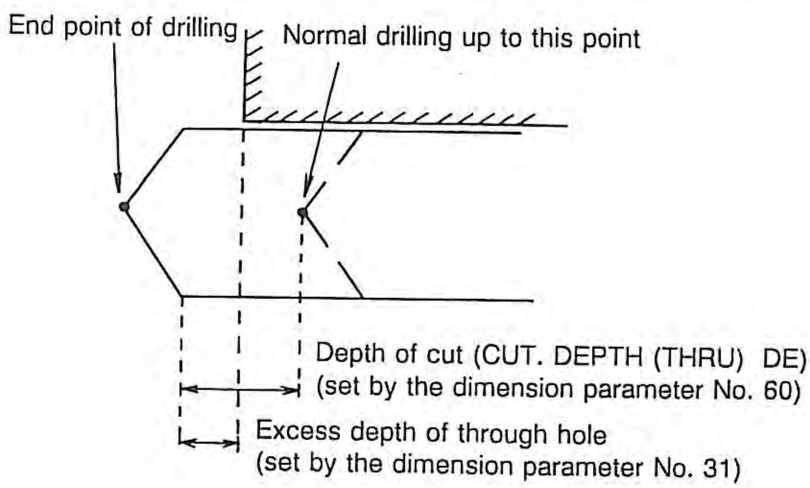
CUT. DEPTH (RETRACT) D2

The depth of at which cutting tool is retracted.

CUT. DEPTH (THRU) DE

The through-hole drilling cycle is executed in the following sequence.

- Normal drilling till just before the workpiece is penetrated
- The drill tool is retracted.
- The drill tool is again infed to the point near the end of first drilled depth at a rapid feedrate.
- The drill is fed at the through-hole feedrate until the workpiece is penetrated.



Feedrate used to drill a through hole can be set by the percent parameter No. 12.

(3) **Contour Generating Machining Process**

The finished shape of the contour generating machining can not be edited in the process editing mode. Edit it in the contour machining shape definition mode.

(4) **Milling Process**

When the rough and finish milling processes are created, change the cutting tool from an endmill to a milling tool on the MACH. METHOD AMEND screen. Refer to the sequence indicated below.

1) Designation of finished shape created in the milling process
2) Automatic determination of the milling process (Cutting tool is changed to "endmill".)
3) Change the cutting tool to a milling tool on the MACH. METHOD AMEND screen for M MACHINING.

(5) **Deleting the Edited Data**

It is possible to delete the data corrected on the MACH. METHOD AMEND screen, and to return to the PROCESS EDITING screen.

Press the function key [F5] (UNDO).

(6) **Switching to the MACH. SHAPE AMEND Screen**

Press the function key [F2] (SHAPE AMEND). The MACH. SHAPE AMEND screen will be displayed. Correct the data as needed.

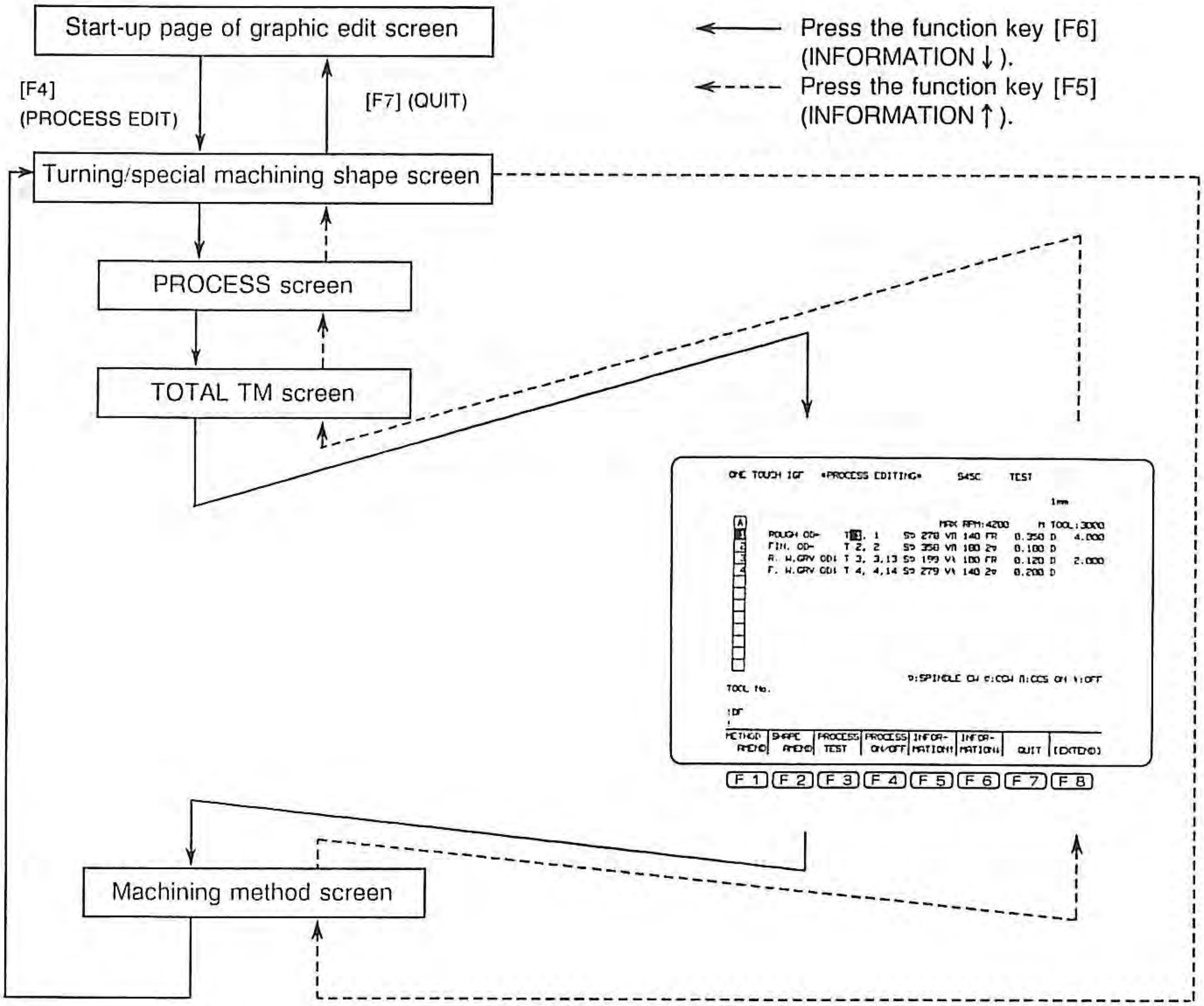
(7) **Terminating the Machining Method Editing**

Press the function key [F7] (QUIT). This completes the editing of machining method.

6. Process List Display

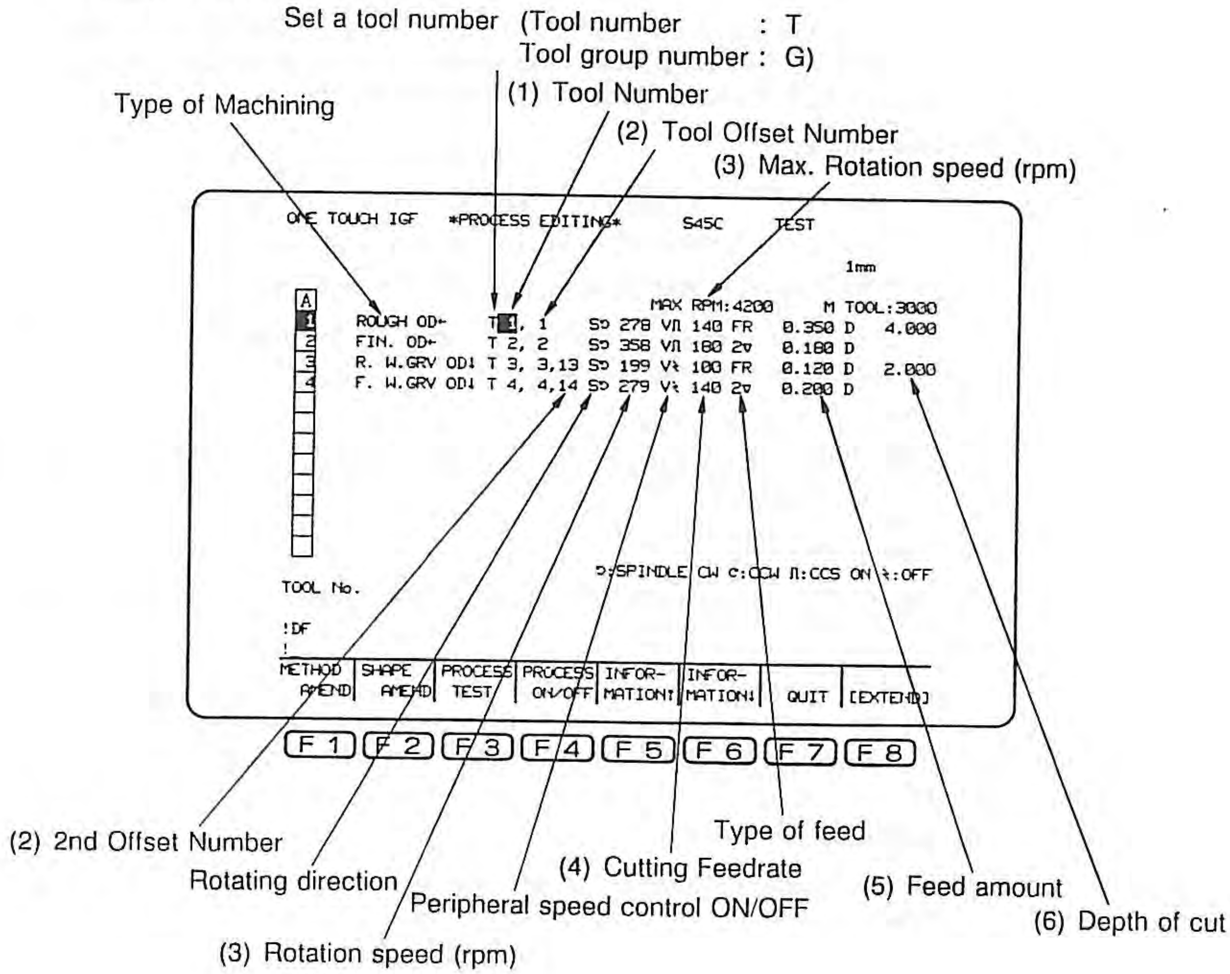
On the process list screen, tool data and cutting conditions for all machining processes are displayed. The following data can be changed on this screen, without calling the MACH. METHOD AMEND screen for each individual machining process.

Tool number, tool offset number, rotation speed, cutting feedrate, feed per revolution, depth of cut



* Process test is performed and a part program is created on the basis of the data set in this process list.

Display Example



* Type of feed

- Rough turning, copying, single grooving, rough grooving, rough endmilling

  → FR

- Finishing, finish grooving, finish endmilling

  → Varies depending on the feed per revolution

  ∇ → 1∇, ∇∇ → 2∇

  ∇∇∇ → 3∇, ∇∇∇∇ → 4∇

- Threading, tapping, drilling, recessing

  → F

Data to be set

The following data can be set on this screen. Call the desired display page using the page keys **PAGE ↑** and **PAGE ↓**, move the cursor to the desired data column using the cursor control keys ↑, ↓, ←, and →, and set the necessary data.

(1) Tool Number

Set a tool number when the type of tool number is "T".

Set a tool group number when the type of tool number is "G".

(2) Tool Offset Number (1st offset number, 2nd offset number)

Set the offset number of the tool in use. In the case of rough grooving and finish grooving, input the 1st and 2nd offset numbers.

(3) Rotation speed (rpm)

Data can be set for the process in which constant peripheral speed control is not used. (" " is displayed.)

(4) Cutting Feedrate

Data can be set for the process in which constant peripheral speed control is used. (" " is displayed.)

(5) Feed per revolution

Set the feed per revolution for each individual machining process. For finishing processes, the feed for the area in which the finest surface roughness is required can be set.

(6) Depth of cut

Set the depth of cut for each individual machining process. For the process with step feed ON, the depth of cut for one step is set.

Supplement:

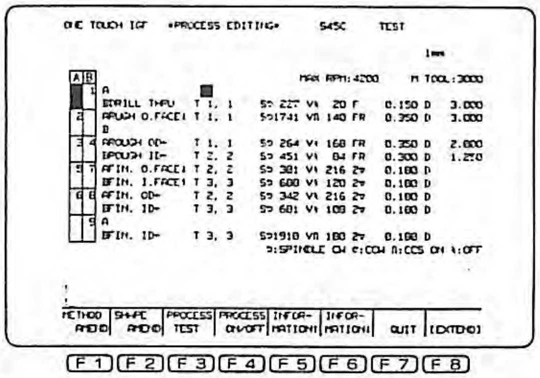
- The following data can be changed using the MACH. METHOD AMEND screen for each machining process.

  Spindle rotating direction
  Peripheral speed control ON/OFF
  Step cutting ON/OFF

- For the data column that is blank and for the data displayed in green, data can not be set or changed. And the set data can not be deleted by keying-in "N" and pressing the **WRITE** key.

- The rotation speed (rpm) data which exceeds the maximum rpm is displayed in yellow.

Display for two-saddle models



With two-saddle models, the machining process for the A-turret and the machining process for the B-turret are displayed alternately on the process list screen. (See the screen above.) The items whose data can be changed and the way of their display are the same as those for one-saddle models, however, for the balance cut process and the simultaneous two-saddle machining process, this is not the case.

- Balance cut process:

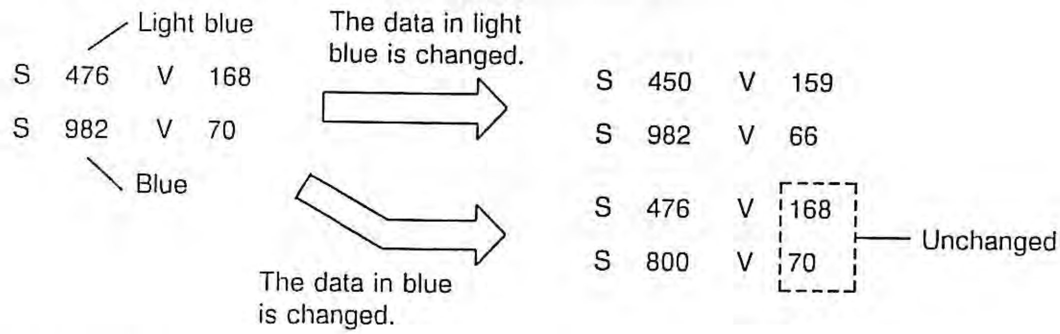
A	B	
1	2	A ROUGH OD ← T11 S 300 V 140 FR0.7 D2.0
		B ROUGH OD ← T22 S 300 V 140 FR0.7 D2.0
		A
		B

Whether the balance cut process is present or not can be designated on the MACH. METHOD AMEND screen for each machining process. When the cutting conditions of one side have been changed, the cutting conditions of the other side are also changed. Note however the cutting conditions set on the MACH. METHOD AMEND screen do not change when the cutting conditions set on the process list have been changed.

- Simultaneous two-saddle machining process:

A	B	
1	2	A ROUGH OD ← T11 S 476 V 168 FR0.35 D2.00
		B ROUGH ID ← T33 S 982 V 70 FR0.30 D1.25
		A
		B

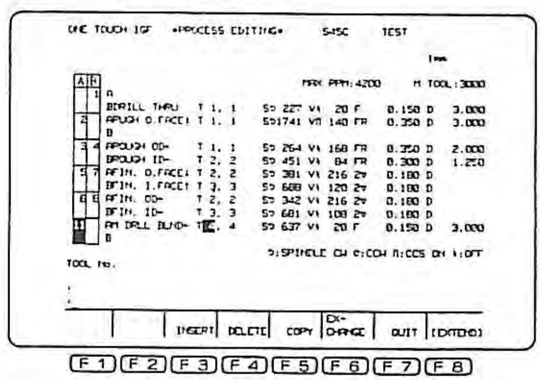
Simultaneous two-saddle machining is carried out with the constant peripheral speed control OFF. Two different rotation speeds are displayed, one in light blue and the other in blue, and the rotation speed displayed in light blue is output to the NC program. Therefore, to change the rotation speed for the simultaneous two-saddle machining process, change the rotation speed displayed in light blue. When the rotation speed is changed, the cutting feedrate will be also changed accordingly. Note however the cutting conditions set on the MACH. METHOD AMEND screen do not change when the cutting conditions set on the process list have been changed.



Reference:

Section 11, 4. Editing the Machining Method (turning/special machining)

Display with the multi-machining specification or contour generating machining specification



- The feed for drilling in the multi-machining process, for example M-drilling, must be set in mm/rev.
- The feed in the contour generating machining process must be set in mm/tooth.

7. Editing the Finished Shape (rough turning, copying, finish turning)

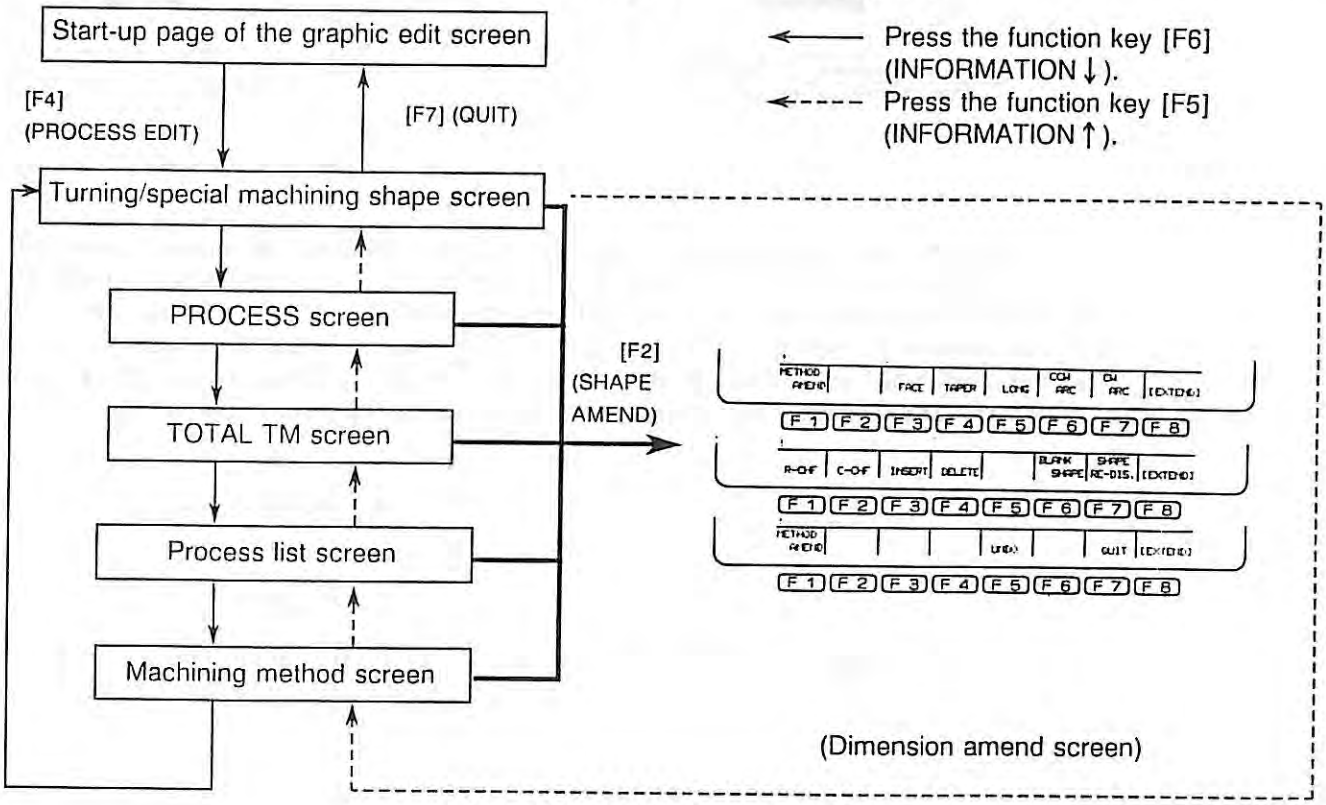
The procedure to edit the finished shape in the rough turning, copying, and finish turning processes is explained.

7-1. Selection of Process to be Edited

To select the finished shape to be edited, move the cursor to the desired process on the turning/special machining shape screen.

a) How to Designate

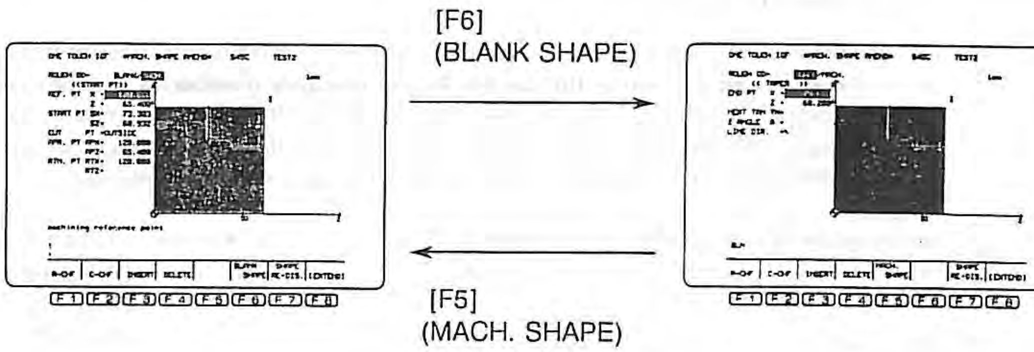
Select the display page on which the desired process is present using the page keys **PAGE ↑** and **PAGE ↓**. Move the cursor to the process number of the process to be edited using the cursor control keys ↑, ↓, ←, and →. When the cursor is moved to the process number, the process name of that process number will flicker. Press the function key [F2] (SHAPE AMEND). The MACH. SHAPE AMEND screen will be displayed.



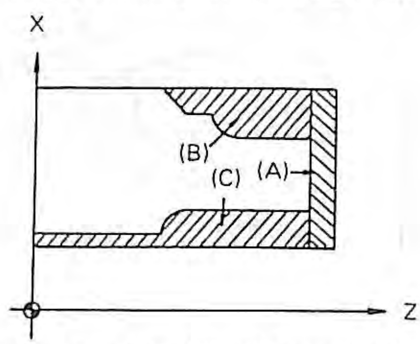
On the MACH. SHAPE AMEND screen for rough turning and copying processes, both the finished shape and the blank material shape are edited. (Only the finished shape is edited for the finish turning process.)

To switch the CRT between the blank shape amend screen and the finished shape amend screen, press the function key [F8] (EXTEND) on the MACH. SHAPE AMEND screen until the function BLANK SHAPE is assigned to the function key [F6]. Then, press the function key [F6] (BLANK SHAPE).

The BLANK SHAPE screen will be displayed. Pressing the function key [F5] (MACH. SHAPE) on the BLANK SHAPE screen will switch the CRT to the MACH. SHAPE AMEND screen.



Tool paths for turning are determined so that the portion between the finished shape and the blank material shape is cut. This means unnecessary tool motion is generated if the defined blank material is larger than the actual blank material. On the other hand, the tool could collide against the blank material if the defined blank material is smaller than the actual blank material. Therefore, the size of blank material must be defined accurately. The blank material shape editing function is provided for the rough turning and copying processes to accurately define the blank material shape.



A: RUGH O. FACE ↓

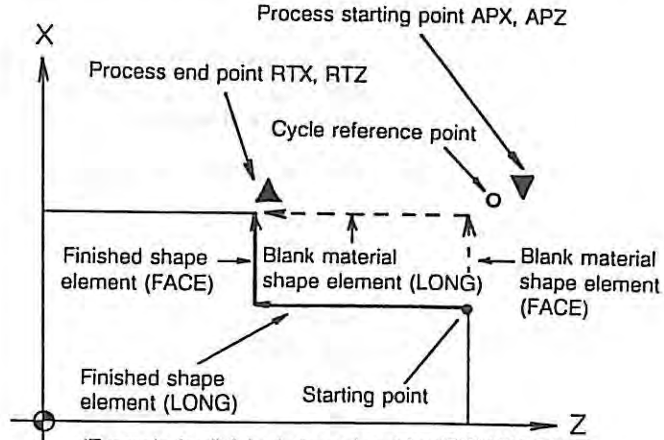
B: ROUGH OD←

C: ROUGH ID←

The portion to be cut is defined separated for each process. Assume that processes are determined as shown above and the ID of the blank material must be reduced. Since the change of ID has influence on portions A and C, the blank material shape data for RUGH O. FACE ↓ and ROUGH OD← need to be changed.

The following functions are provided to edit the finished shape and the blank material shape.

(1) START PT
(2) FACE
(3) TAPER
(4) LONG
(5) CCW ARC
(6) CW ARC
(7) R-CHF
(8) CHAMFER
(9) BLANK/MACH.
(10) Insertion of shape element
(11) Deletion of shape element
(12) Re-display of shape



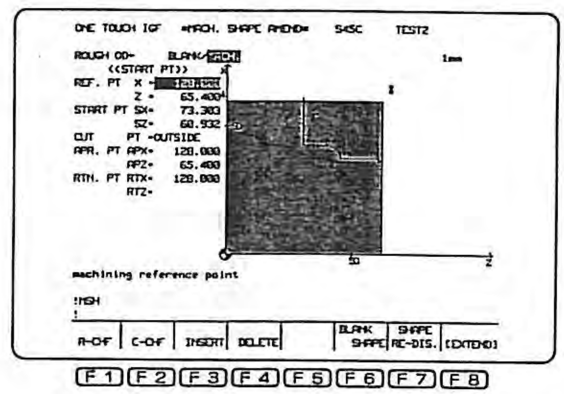
(Example for finished shape input for (ROUGH OD←))

For procedures and notes for editing the shape element, refer to Section 6, 3. "TURNING MACH SHAPE Definition" or Section 7, 3. "BLANK SHAPE Definition".

(1) **START PT**

a) Function

The reference point and starting point when the finished shape and blank material shape are defined are designated.



b) Data to be Designated

REF. PT	X
	Z
START PT	SX
	SZ
CUT PT	CP
APR. PT	APX
	APZ
RTN. PT	RTX
	RTZ

c) How to Designate

REF. PT X, Z

This is the cycle reference point. After the cutting tool has approached this point at the rapid feedrate, machining is started.

START PT SX, SZ

The coordinate values of the end most point for the series of shape elements of finished and blank material shape. This is the cutting starting point.

When the cutting starting point is moved, it will influence on other shape elements.

CUT PT CP

APR. PT APX, APZ

RTN. PT RTX, RTZ

Refer to Section 11, 9. "Tool Path Control".

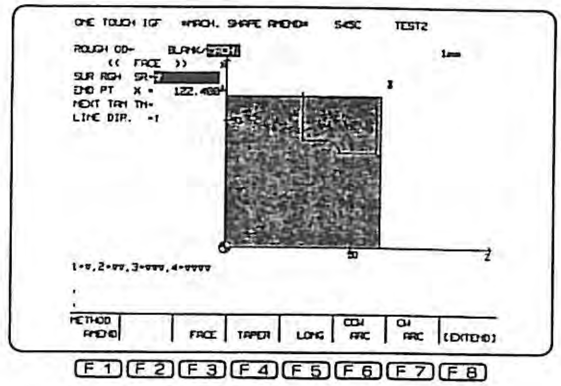
(2) **FACE**

a) Function

The shape of the end face in either the blank material or the finished product is defined.

b) Data to be Designated

SUR RGH SR
END PT X
Z
NEXT TAN TN
LINE DIR.



c) How to Designate

Designate in the same manner as turning shape definition or arbitrary shape definition. Refer to Section 6, 3. "TURNING MACH SHAPE Definition" or Section 7, 3. "BLANK SHAPE Definition".

SR RGH SR

The feedrate for finishing the end face in the finish turning process is designated. The actual feedrate can be designated in the MACH. METHOD AMEND screen.

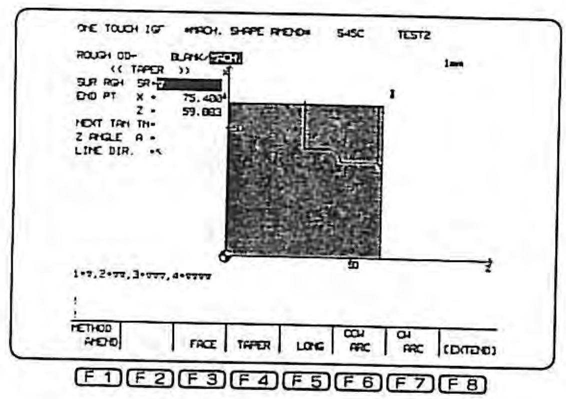
(3) **TAPER**

a) Function

The shape of the taper in either the blank material or the finished product is defined.

b) Data to be Designated

SUR RGH SR
END PT X
Z
NEXT TAN TN
Z ANGLE A
LINE DIR.



c) How to Designate

Designate in the same manner as turning shape definition or arbitrary shape definition.

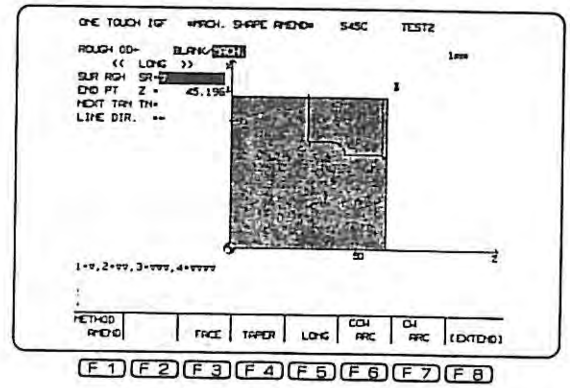
(4) LONG

a) Function

The shape of the longitudinal portion in either the blank material or the finished product is defined.

b) Data to be Designated

SUR RGH	SR
END PT	X
	Z
NEXT TAN	TN
LINE DIR.	



c) How to Designate

Designate in the same manner as turning shape definition or arbitrary shape definition.

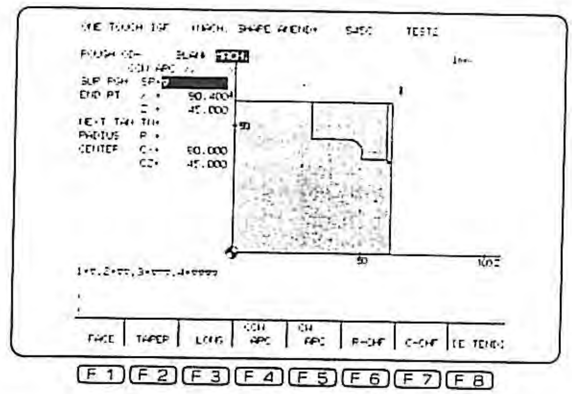
(5) CCW ARC

a) Function

The shape of the CCW arc in either the blank material or the finished product is defined.

b) Data to be Designated

SUR RUGH	SR
END PT	X
	Z
NEXT TAN	TN
RADIUS	R
CENTER	CX
	CZ



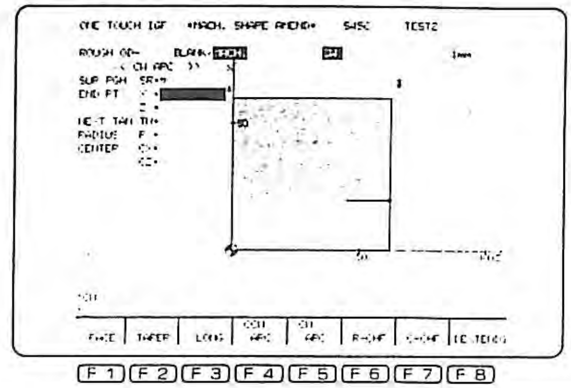
c) How to Designate

Designate in the same manner as turning shape definition or arbitrary shape definition.

(6) CW ARC

a) Function

The shape of the CW arc in either the blank material or the finished product is defined.

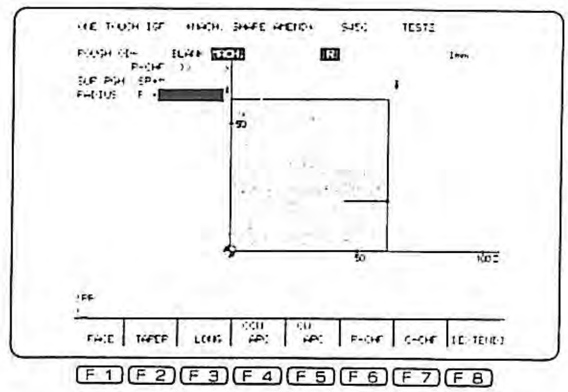


b) Data to be Designated

SUR RUGH SR
END PT X
Z
NEXT TAN TN
RADIUS R
CENTER CX
CZ

c) How to Designate

Designate in the same manner as turning shape definition or arbitrary shape definition.



(7) R-CHF

a) Function

The shape of the rounding in either the blank material or the finished product is defined.

b) Data to be Designated

SUR RUGH SR
RADIUS R

c) How to Designate

Designate in the same manner as turning shape definition or arbitrary shape definition.

(8) **CHAMFER**

a) Function

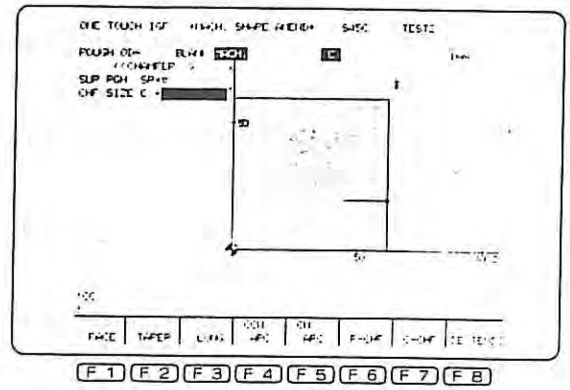
The shape of the chamfer in either the blank material or the finished product is defined.

b) Data to be Designated

SUR RUGH SR
RADIUS R

c) How to Designate

Designate in the same manner as turning shape definition or arbitrary shape definition.



(9) **Switching Between Blank Shape and Finished Shape**

a) Function

Switching between the finished shape amend screen and the blank material shape amend screen is executed.

b) How to Designate

Pressing the function key [F6] (BLANK SHAPE) displays the BLANK SHAPE screen, and pressing the function key [F5] (MACH. SHAPE) displays the MACH. SHAPE AMEND screen.

(10) **Insertion of Shape Element**

a) Function

Shape element is inserted.

b) How to Designate

Move the marker using the cursor control keys ← and →, and press the function key [F3] (INSERT). The shape element is inserted before the marker-located shape element.

Designate in the same manner as turning shape definition or arbitrary shape definition.

Note: *The thread, groove, tap, or recess process cannot be inserted in the machining shape amend mode. Newly create these processes.*

(11) **Deletion of Shape Element**

a) Function

The designated shape element is deleted.

b) How to Designate

Place the marker at any position of the shape element to be deleted using the cursor control keys ← and →.

Press the function key [F4] (DELETE). The marker-located shape element will be deleted.

Designate in the same manner as turning shape definition or arbitrary shape definition.

(12) **Re-display of Shape**

a) Function

The defined shape elements are all displayed on the screen.

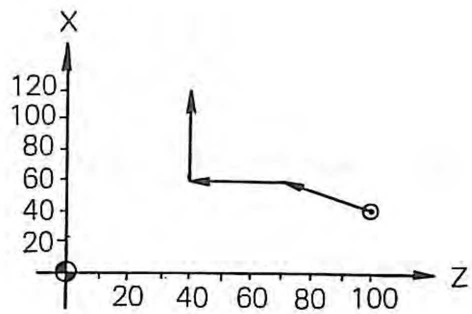
b) How to Designate

Press the function key [F7] (SHAPE RE-DIS.). The defined shape elements are all displayed on the screen.

Designate in the same manner as turning shape definition or arbitrary shape definition.

Example 1:

Change the taper on the figure below to the arc of R50 which is tangent to the longitudinal portion.

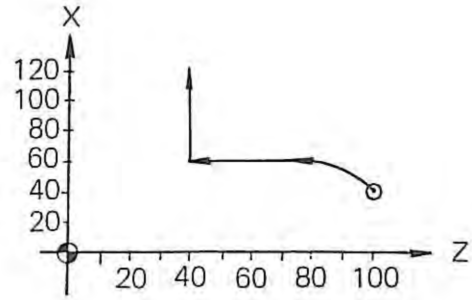


Shape elements

START PT
↓
TAPER
↓
LONG
↓
FACE

Fig. 11-1

1) Locate the marker (magenta) to any position of taper using the cursor control keys ← and →.
2) Press the function key [F8] (EXTEND) until the function "DELETE" is assigned to the function key [F4].
3) Press the function key [F4] (DELETE).
4) The marker is advanced to the shape element "LONG".
5) Press the function keys [F3] (INSERT), [F8] (EXTEND), and [F6] (CCW) in this order.
6) Key in the value for NEXT TAN TN and RADIUS R.
7) Press the function key [F8] (EXTEND) until the function "SHAPE RE-DIS." is assigned to the function key [F7].
8) Press the function key [F7] (SHAPE RE-DIS.). The shape in Fig. 11-2 will be displayed.



Shape elements

START PT
↓
CCW ARC
↓
LONG
↓
FACE

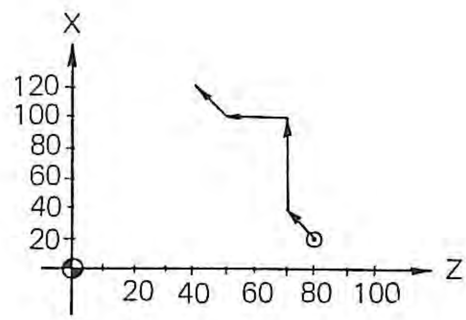
Fig. 11-2

Example 2:

Insert a taper between the longitudinal portion and the end face on the figure below.

Details of the taper are follows:

Starting point (Z, X) = (70, 80)
Angle made between the Z-axis 135°



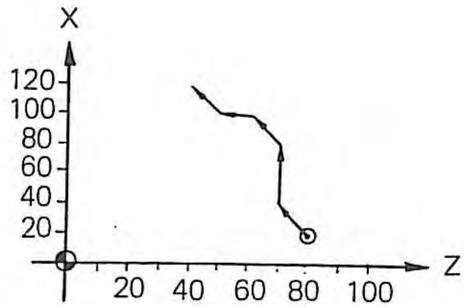
Shape elements

START PT
↓
TAPER
↓
LONG
↓
FACE
↓
TAPER

Fig. 11-3

1) Move the marker (magenta) to any position of longitudinal portion using the cursor control keys ← and →.
2) Press the function key [F8] (EXTEND) until the function "INSERT" is assigned to the function key [F3].
3) Press the function key [F3] (INSERT).
4) Press the function key [F8] (EXTEND) twice. The function "TAPER" is assigned to the function key [F3].
6) Press the function key [F3] (TAPER).
7) Enter "135" for Z ANGLE A.
8) Move the marker (magenta) to any position on the end face using the cursor control keys ← and →.
9) Change the value for END PT X to "80".
10) Press the function key [F8] (EXTEND) until the function "SHAPE RE-DIS." is assigned to the function key [F7].

The shape in Fig. 11-4 will be displayed.



Shape elements

START PT
↓
TAPER
↓
FACE
↓
TAPER
↓
LONG
↓
TAPER

Fig. 11-4

7-2. Motion from Cycle Reference Point to Infeed Point

Rough turning, copying, and finish turning are executed in the LAP mode.

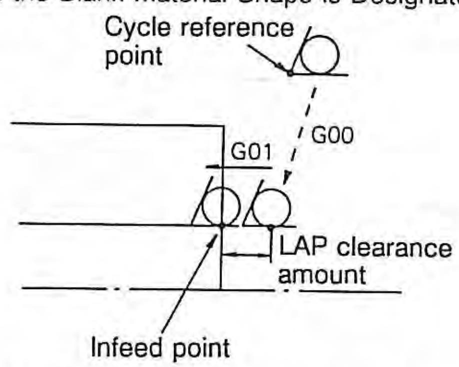
Rough turning .... High-speed bar turning cycle

Copying .... Copy turning cycle

Finish turning .... Finish turning cycle

Tool motion from the cycle reference point to the (first) infeed point is as indicated below.

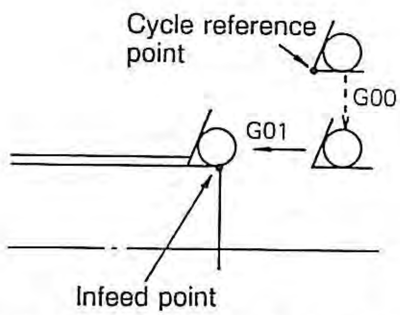
(1) When the Blank Material Shape is Designated



The tool nose radius compensation function is activated when the X and Z axes are positioned in the G00 mode.

(2) When the Blank Material Shape is not Designated

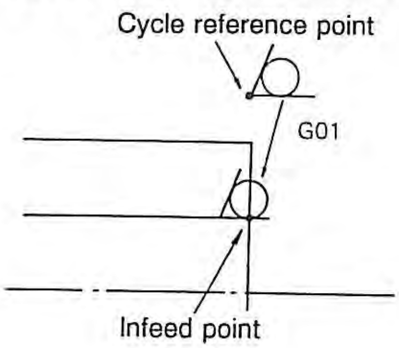
- The X- and Z-coordinate values of the cycle reference point (REF. PT) and these of the cutting starting point (START PT) differ each other.



After the X-axis has been positioned in the G00 mode, the tool nose radius compensation function is activated when the Z-axis is positioned to the infeed point in the G01 mode.

This motion is carried out in the finish turning cycle.

- The X-coordinate value of the cycle reference point (REF. PT) is identical to the X-coordinate value of the cutting starting point (START PT), or the Z-coordinate value of the cycle reference point is identical to the Z-coordinate value of the cutting starting point.



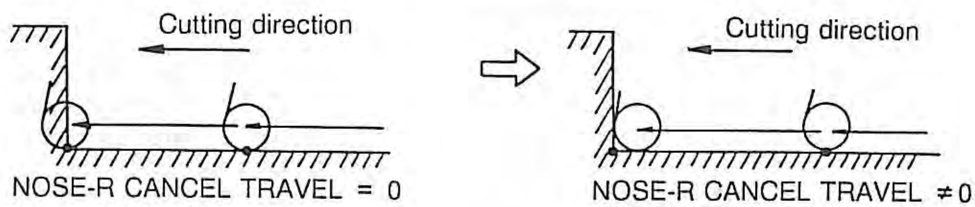
The axis is directly positioned to the infeed point in the G01 mode with the tool nose radius compensation function activated.

7-3. Motion at the End Point

When cutting is ended halfway in the finish turning process created in the process edit mode, excessive infeed might occur due to the tool nose radius compensation function.

With the one-touch IGF, I and K commands can be output in the program block where tool nose radius compensation cancel G40 is designated, by setting the value for the dimension parameter No. 8 NOSE-R CANCEL TRAVEL.

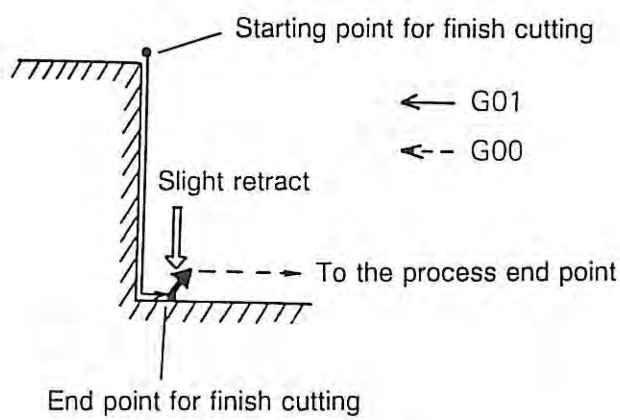
Example:



Suitable I and K commands are output taking the cutting process and the relationship between the cycle reference point and the cutting starting point into consideration.

In finish turning, the tool tip is either in contact with the workpiece or very close to the workpiece when the finish turning cycle is completed. Therefore, the cutting tool is finally retracted to the process end point after it has been slightly retracted from the workpiece at the cutting feedrate.

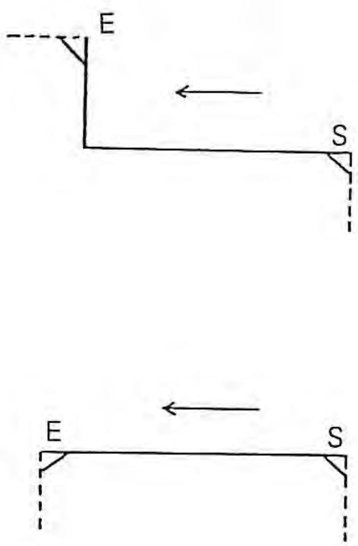
Example: FIN. OD ↓



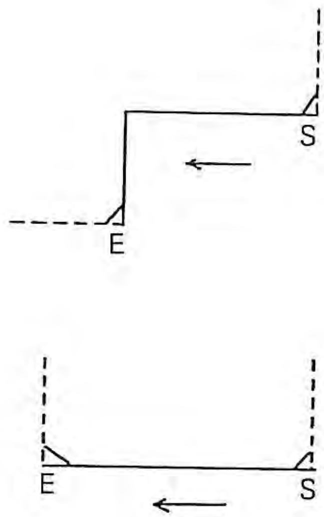
7-4. Starting and End Points for Rounding (R-CHF) and Chamfering

When rounding or chamfering is designated as the shape element following the starting point or as the final shape element, its shape is determined from the cutting direction and the machining shape in that process.

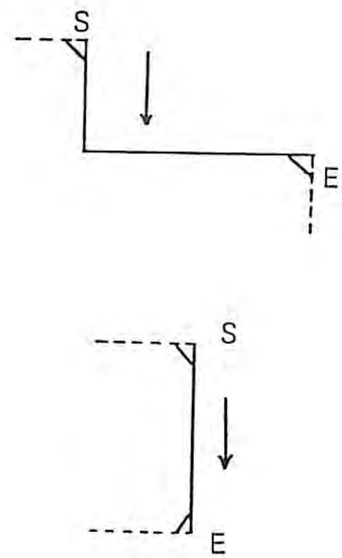
1. OD ←



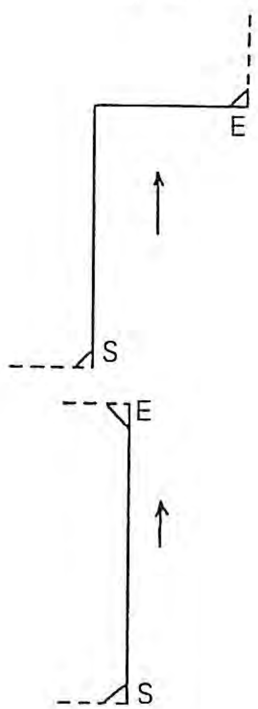
2. ID ←



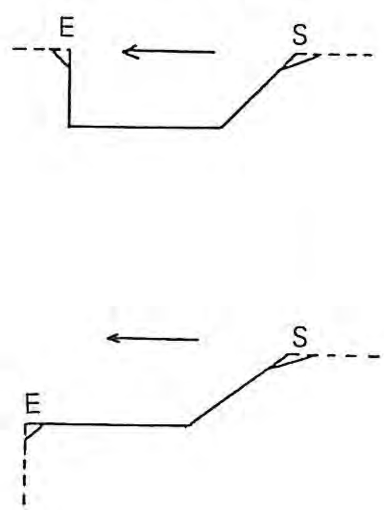
3. OD FACE ↓



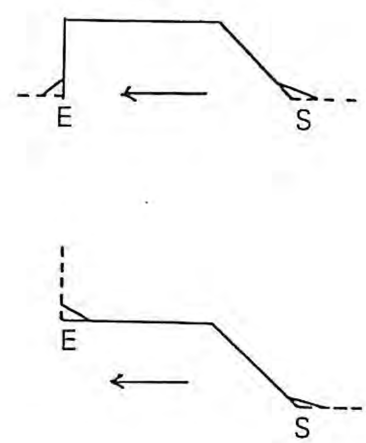
4. ID FACE ↑



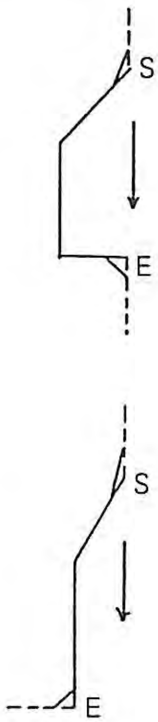
5. MID OD ←



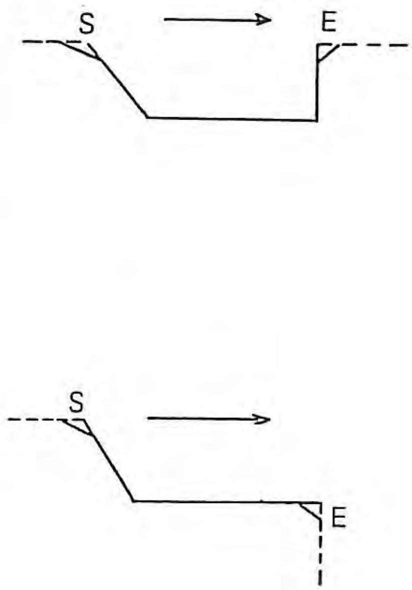
6. MID ID ←



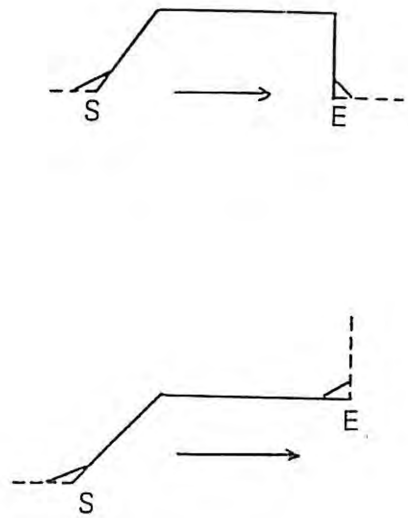
7. MID FACE ↓



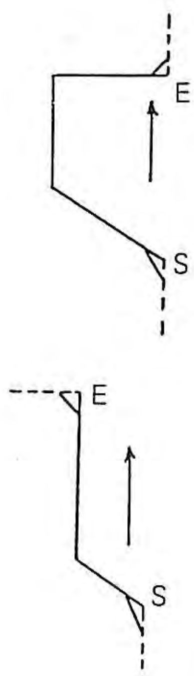
8. MID OD →



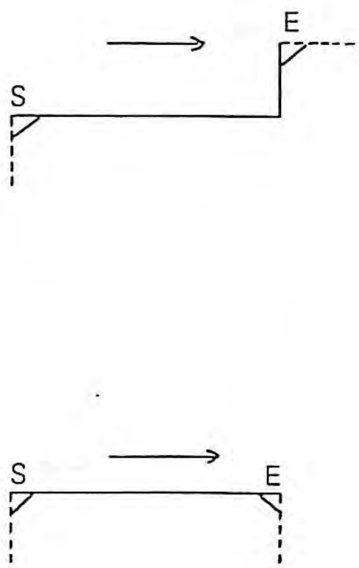
9. MID ID →



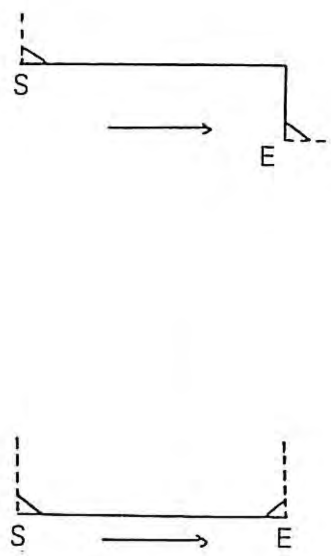
10. MID FACE ↑



11. OD →



12. ID →



This applies to rounding as well.

7-5. Deleting the Edited Data

It is possible to delete the data corrected on the MACH. SHAPE AMEND or MACH. METHOD AMEND screen, and to return to the process check screen for turning/special machining shape.

Press the function key [F8] (EXTEND) until the function "UNDO" is assigned to the function key [F5].

Press the function key [F5] (UNDO).

The process check screen for turning/special machining shape will be displayed.

7-6. Switching to the MACH. METHOD AMEND Screen

It is possible to switch the CRT from the MACH. SHAPE AMEND screen to the MACH. METHOD AMEND screen.

Press the function key [F8] (EXTEND) until the function "METHOD AMEND" is assigned to the function key [F1].

Press the function key [F1] (METHOD AMEND). The MACH. METHOD AMEND screen will be displayed.

Note: *There are cases in which switching to the MACH. METHOD AMEND screen is disabled due to faulty or incorrect data on the MACH. SHAPE AMEND screen.*

7-7. Terminating the Machining Shape Editing

Press the function key [F7] (QUIT) after the completion of machining shape editing. The CRT will be switched from the MACH. SHAPE AMEND screen to the process check screen for turning/special machining shape.

Note: *There are cases in which switching to the process check screen is disabled due to faulty or incorrect data on the MACH. SHAPE AMEND screen.*

8. Editing the Finished Shape (special turning)

The procedure to edit the finished shape in the thread cut, tapping, grooving, drilling, endmilling, and recessing processes is explained.

8-1. Selection of Process to be Edited

To select the finished shape to be edited, move the cursor to the desired process (thread, tap, groove, drill, endmill, recess) on the turning/special machining shape screen.

Designate in the same manner as finished shape definition for the rough turning, copying, and finish turning processes. Refer to Section 11, 6. "Editing the Finished Shape (rough turning, copying, finish turning)".

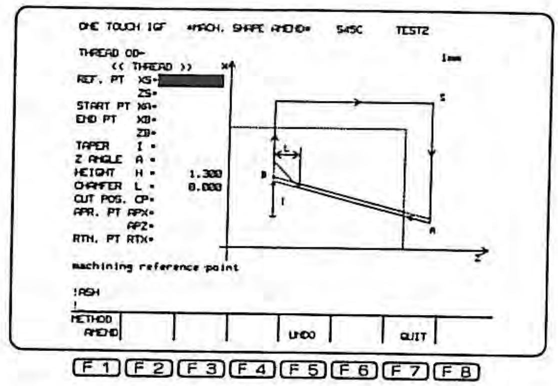
(1) **THREAD**

a) Function

The shape of thread is changed.

b) Data to be Designated

REF. PT	XS
	ZS
START PT	XA (or ZA)
END PT	XB
	ZB
TAPER	I (or K)
Z ANGLE	A
HEIGHT	H
CHAMFER	L
CUT PT	CP
APR. PT	APX, APZ
RTN. PT	RTX, RTZ



(S: reference point, A: starting point, B: end point)

c) How to Designate

Change the shape of thread while watching the guide drawing on the screen.

For detailed explanation of the data to be designated, refer to Section 6, 3. (10) "THREAD" on page 98. For CUT PT CP, APR. PT APX and APZ, RTN. PT RTX and RTZ, refer to Section 11, 9. "Tool Path Control".

Note 1: *Data to be designated and the way to designate are the same for all THREAD OD←, THREAD ID←, THREAD FACE↓, THREAD OD→, THREAD ID→, and THREAD FACE↑.*

Note 2: *Correction of the data for surface roughness, pitch, number of threads, and number of leads is accomplished on the MACH. METHOD AMEND screen.*

Press the function key [F1] (METHOD AMEND) to display the MACH. METHOD AMEND screen.

Note 3: *When the thread shape is defined in the MACH. METHOD AMEND screen, designate the coordinate values for points A and B taking the incomplete thread portion into consideration.*

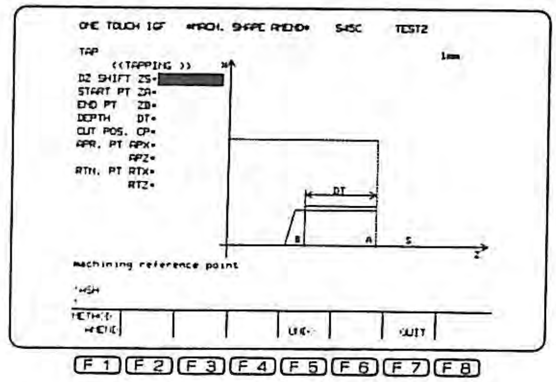
(2) TAPPING

a) Function

The shape of tapping is changed.

b) Data to be Designated

REF. PT	ZS
START PT	ZA
END PT	ZB
DEPTH	DT
CUT PT	CP
APR. PT	APX
	APZ
RTN. PT	RTX
	RTZ



c) How to Designate

Change the shape of tapping while watching the guide drawing on the screen.

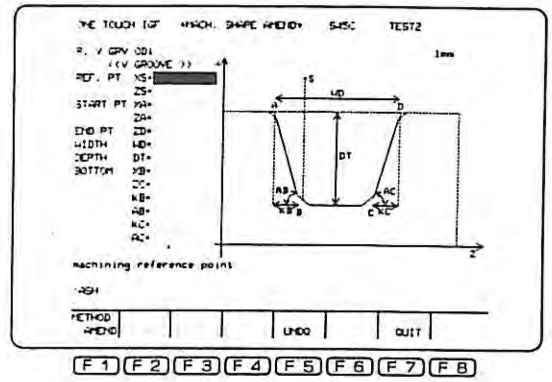
Note 1: *When an attempt is made to define the shape of tapping in the machining shape definition mode, the one-touch IGF will automatically determine a series of processes consisting of drilling, rough endmilling, and tapping. However, if an attempt is made to change the shape data of tapping in the process edit mode, shape data for drilling, rough endmilling, and tapping must be all changed. Therefore it is recommended to re-define the machining shape.*

Note 2: *Designate the coordinate values for point B taking the incomplete thread portion of tap into consideration.*

(3) GROOVE

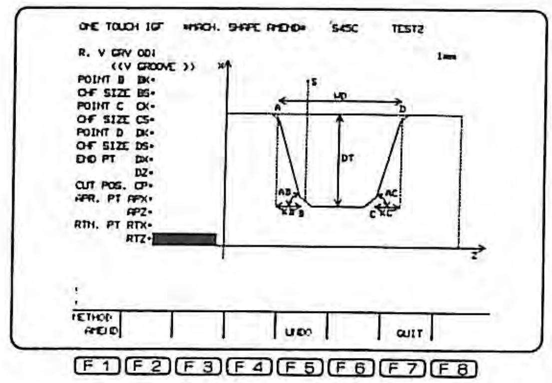
a) Function

The shape of single groove, rough wide groove, rough V-groove, or finish groove is changed.



b) Data to be Designated

REF. PT	XS, ZS
START PT	XA (or ZA)*
END PT	ZD (or XD)#
WIDTH	WD
DEPTH	DT
BOTTOM	XB (or ZB)
	ZC (or XC)*
	KB (or IB)*
	AB*
	KC (or IC)*
	AC*
GROOVES	NG
SHIFT AMOUNT	K (I)
SHIFT POINT A	AK
CHF SIZE	AS
START PT	AX, AZ
POINT B	BK#
CHF SIZE	BS#
POINT C	CK#
CHF SIZE	CS#
POINT D	DK
CHF SIZE	DS
END PT	DX, DZ
CUT PT	CP
APR. PT	APX, APZ
RTN. PT	RTX, RTZ



\# Designated for rough wide groove, rough V-groove, and finish groove

* Designated for rough V-groove and finish groove

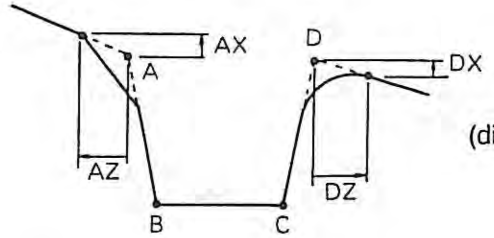
Designate both XA and ZA.

c) How to Designate

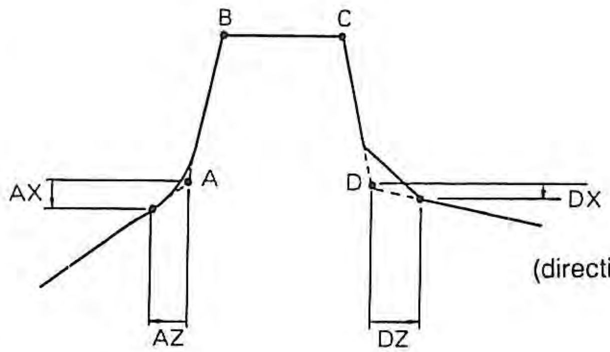
Change the shape of groove while watching the guide drawing on the screen.

Assume START PT AX and AZ, and END PT DX and DZ as diagramed below. These data must be entered when a groove is cut on the taper.

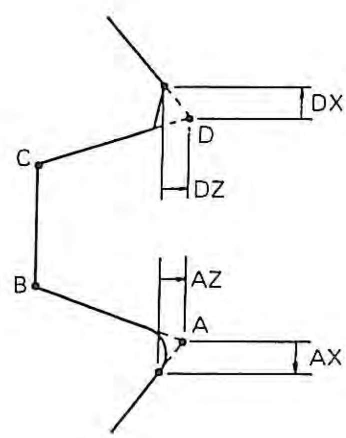
When a groove is cut in the transverse portion or longitudinal portion, no data entry is required.



(direction of groove ↓)



(direction of groove ↑)



(direction of groove←)

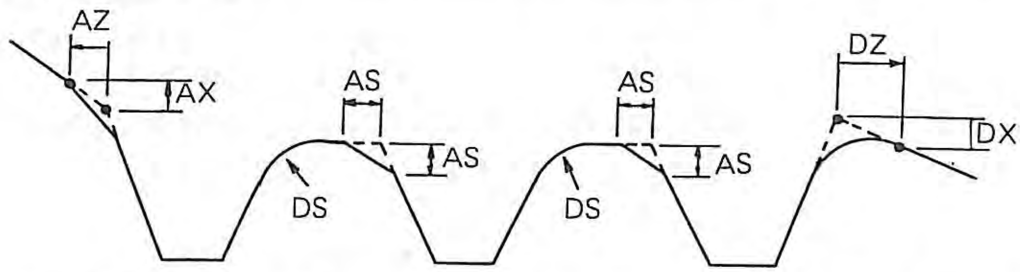
* AX, AZ .... Distance between the chamfer starting point and point A in incremental value

  DX, DZ .... Distance between the chamfer starting point and point D in incremental value

The arrow-marked direction is positive (+).

Enter the values for AX and DX in radius.

In the case of multi-groove cutting, assume START PT AX and AZ, and END PT DX and DZ as diagramed below.



For detailed explanation of the data to be designated, refer to Section 6, 3. (9) "GROOVE" on page 94.

Note 1: *Correction of the data for surface roughness is accomplished on the MACH. METHOD AMEND screen.*

Press the function key [F1] (METHOD AMEND) to display the MACH. METHOD AMEND screen.

Note 2: *For CUT PT CP, APR. PT APX and APZ, RTN. PT RTX and RTZ, refer to Section 11, 9. "Tool Path Control".*

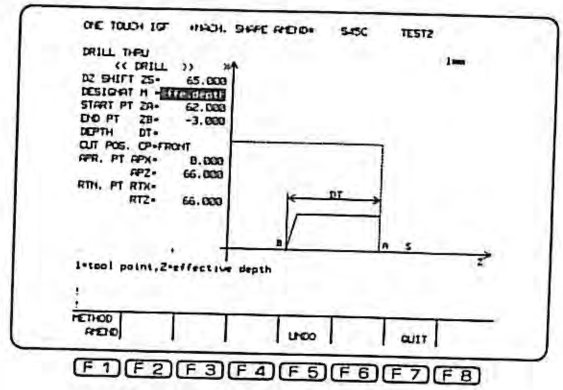
(4) DRILL

a) Function

The shape of drilling is changed.

b) Data to be Designated

REF. PT	ZS
DESIGNAT	M
START PT	ZA
END PT	ZB
DEPTH	DT
CUT PT	CP
APR. PT	APX
	APZ
RTN. PT	RTX
	RTZ



(S: reference point, A: starting point, B: end point)

c) How to Designate

Change the shape of drilling while watching the guide drawing on the screen.

Note 1: *The value for END PT ZB can be set either by the coordinate of the tool tip or by the effective length, according to the setting at DESIGNAT M.*

Note 2: *Data to be designated and how the data is designated are the same for blind hole, through hole, and centering.*

Note 3: *For CUT PT CP, APR. PT APX and APZ, RTN. PT RTX and RTZ, refer to Section 11, 9. "Tool Path Control".*

Note 4: *The following drills are not selected even if they have been registered to the tool data.*

Drill whose diameter is larger than the setting of the dimension parameter No. 37

Carbide drill whose diameter is smaller than the setting of the dimension parameter No. 36

(5) **END MILL**

a) Function

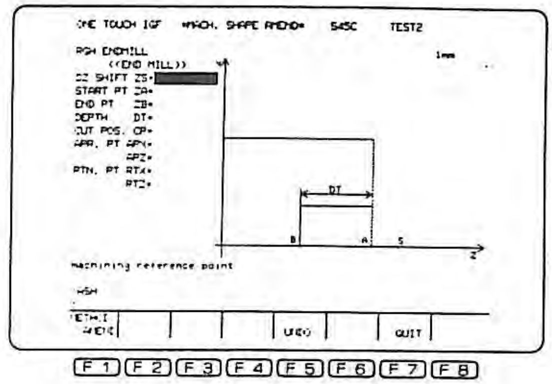
The shape of endmilling is changed.

b) Data to be Designated

REF. PT	ZS
START PT	ZA
END PT	ZB
DEPTH	DT
CUT PT	CP
APR. PT	APX, APZ
RTN. PT	RTX, RTZ

c) How to Designate

Change the shape of endmilling while watching the guide drawing on the screen.



(S: reference point, A: starting point, B: end point)

Note 1: *Data to be designated and the way to designate are the same for both rough endmilling and finish endmilling.*

Note 2: *For CUT PT CP, APR. PT APX and APZ, RTN. PT RTX and RTZ, refer to Section 11, 9. "Tool Path Control".*

(6) **RECESS**

a) Function

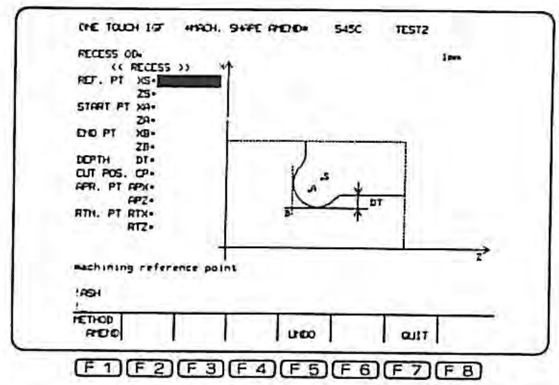
The shape of recessing is changed.

b) Data to be Designated

REF. PT	XS
	ZS
START PT	XA
	ZA
END PT	XB
	ZB
DEPTH	DT
CUT PT	CP
APR. PT	APX, APZ
RTN. PT	RTX, RTZ

c) How to Designate

Change the shape of recessing while watching the guide drawing on the screen.



(S: reference point, A: starting point, B: end point)

Note 1: *Data to be designated and the way to designate are the same for all RECESS OD↙, RECESS OD↘, RECESS ID↖, and RECESS ID↗.*

Note 2: *For CUT PT CP, APR. PT APX and APZ, RTN. PT RTX and RTZ, refer to Section 11, 9. "Tool Path Control".*

8-2. Deleting the Edited Data

It is possible to delete the data corrected on the MACH. METHOD AMEND or MACH. SHAPE AMEND screen, and to return to the process check screen for turning/special machining shape.

Press the function key [F5] (UNDO).

The process check screen for turning/special machining shape will be displayed.

8-3. Switching to the MACH. METHOD AMEND Screen

It is possible to switch the CRT from the MACH. SHAPE AMEND screen to the MACH. METHOD AMEND screen.

Press the function key [F1] (METHOD AMEND). The MACH. METHOD AMEND screen will be displayed.

Note: *There are cases in which switching to the MACH. METHOD AMEND screen is disabled due to faulty or incorrect data on the MACH. SHAPE AMEND screen.*

8-4. Terminating the Machining Shape Editing

Press the function key [F7] (QUIT) after the completion of machining shape editing. The CRT will be switched from the MACH. SHAPE AMEND screen to the process check screen for turning/special machining shape.

Note: *There are cases in which switching to the process check screen is disabled due to faulty or incorrect data on the MACH. SHAPE AMEND screen.*

9. Editing the Finished Shape (multi-machining) (optional: multi-machining specification)

The procedure to edit the finished shape in the M-drilling, M-boring, M-tapping, M-reaming, M-keyway cutting, and M-endmilling processes is explained.

9-1. Selection of Process to be Edited

To select the shape to be edited, first move the cursor to the desired process (M-drilling, M-boring, M-tapping, M-reaming, M-keyway cutting, M-endmilling) on the multi-machining face shape screen or the multi-machining side shape screen in the process edit mode. Then, press the function key [F2] (SHAPE AMEND). The corresponding MACH. SHAPE AMEND screen will be displayed.

(1) **M DRLL (BLND, THRU, CENT, CHAMF)**

(2) **M ENDMILLING**

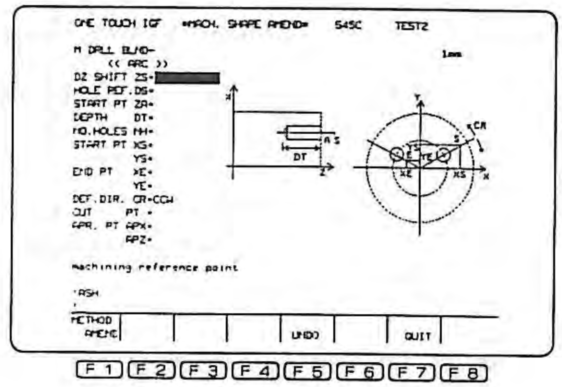
a) Function

The shape for M-drilling or M-endmilling is changed.

b) Data to be Designated

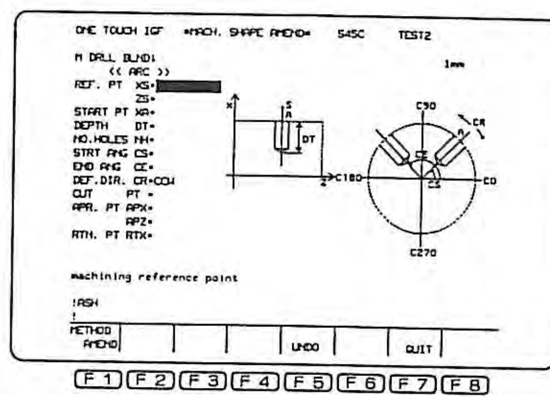
(Front Cutting)

REF. PT	ZS
HOLE REF.	DS (*1)
START PT	XA (*2)
	YA (*2)
	ZA
DEPTH	DT
NO. HOLES	NH (*1)
START PT	XS (*1)
	YS (*1)
END PT	XE (*1)
	YE (*1)
DEF. DIR.	CR (*1)
CUT PT	CP
APR. PT	APX
	APZ
RTN. PT	RTZ



(Side Cutting)

REF. PT	XS
	ZS (*1)
START PT	XA
	CA (*2)
	ZA (*2)
DEPTH	DT
NO.HOLES	NH (*1)
STRT ANG	CS (*1)
END ANG	CE (*1)
DEF.DIR.	CR (*1)
CUT PT	CP
APR. PT	APX
	APZ
RTN. PT	RTZ



*1 For ARC only
*2 For FREE POSIT only

c) How to Designate

Change the shape for M-drilling or M-endmilling while watching the guide drawing on the screen.

For detailed explanation of the data to be designated, refer to Section 6, 4. "Finished Shape Definition for Multiple Machining".

For CUT PT CP, APR. PT APX and APZ, RTN. PT RTX and RTZ, refer to Section 11, 9. "Tool Path Control".

Note 1: *The shape for the M-drill chamfering process is indicated by the drill diameter.*

Note 2: *When an attempt is made to define the shape of M-drilling in the machining shape definition mode, the one-touch IGF will automatically determine the preparatory process. However, if an attempt is made to change the shape data of M-drilling in the process edit mode, shape data for the preparatory process must be changed. Therefore it is recommended to re-define the machining shape.*

(3) **M BORING**

(4) **M TAPPING**

(5) **M REAMING**

a) Function

The shape for M-boring, M-tapping, and M-reaming cutting is changed.

b) Data to be Designated

Same as M DRLL.

c) How to Designate

Change the shape while watching the guide drawing on the screen.

Note: *In multi-machining shape definition, when the shape to be created is designated, the one-touch IGF will automatically generate the preparatory, roughing, and finishing processes for that shape. However, since the automatic generation of preparatory, roughing, and finishing processes is not accomplished in the process edit mode, if an attempt is made to edit the process, shape data in the preparatory, roughing, and finishing processes must be changed as needed. Therefore it is recommended to re-define the machining shape.*

(6) M KEYWAY

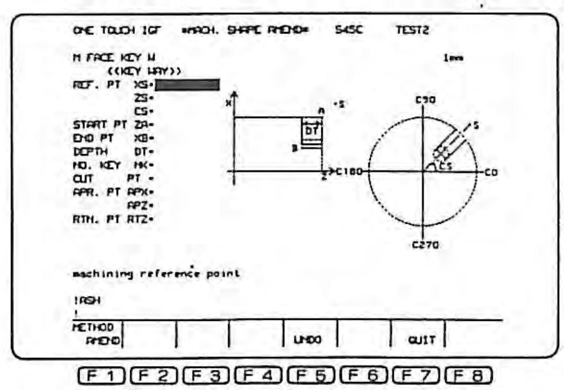
a) Function

The shape for M-keyway cutting is changed.

b) Data to be Designated

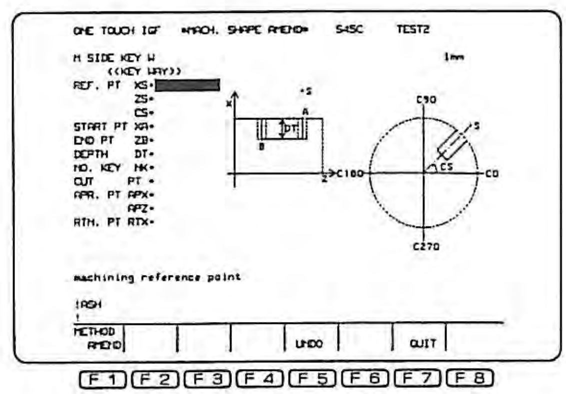
(Face cutting)

REF. PT	XS
	ZS
	CS
START PT	XA
	ZA
END PT	XB
DEPTH	DT
NO. KEY	NK
CUT	CP
APR. PT	APX
APR. PT	APZ
RTN. PT	RTZ



(Side cutting)

REF. PT	XS
	ZS
	CS
START PT	XA
END PT	ZB
DEPTH	DT
NO. KEY	NK
CUT	CP
APR. PT	APX
APR. PT	APZ
RTN. PT	RTX



c) How to Designate

Change the shape while watching the guide drawing on the screen.

Note: *In multi-machining shape definition, when the shape of M-keyway is designated, the one-touch IGF will automatically generate a series of preparatory, roughing, and finishing processes for cutting the M-keyway. However, since the automatic generation of preparatory, roughing, and finishing processes is not accomplished in the process edit mode, if an attempt is made to edit the process, shape data in the preparatory, roughing, and finishing processes must be changed as needed. Therefore, it is recommended to re-define the machining shape.*

9-2. Deleting the Edited Data

It is possible to delete the data corrected on the MACH. SHAPE AMEND or MACH. METHOD AMEND screen, and to return to the process check screen for multi-machining shape.

Press the function key [F5] (UNDO).

The process check screen for multi-machining shape will be displayed.

9-3. Switching to the MACH. METHOD AMEND Screen

It is possible to switch the CRT from the MACH. SHAPE AMEND screen to the METHOD AMEND screen.

Press the function key [F1] (METHOD AMEND). The MACH. METHOD AMEND screen will be displayed.

Note: *There are cases in which switching to the MACH. METHOD AMEND screen is disabled due to faulty or incorrect data on the MACH. SHAPE AMEND screen.*

9-4. Terminating the Machining Shape Editing

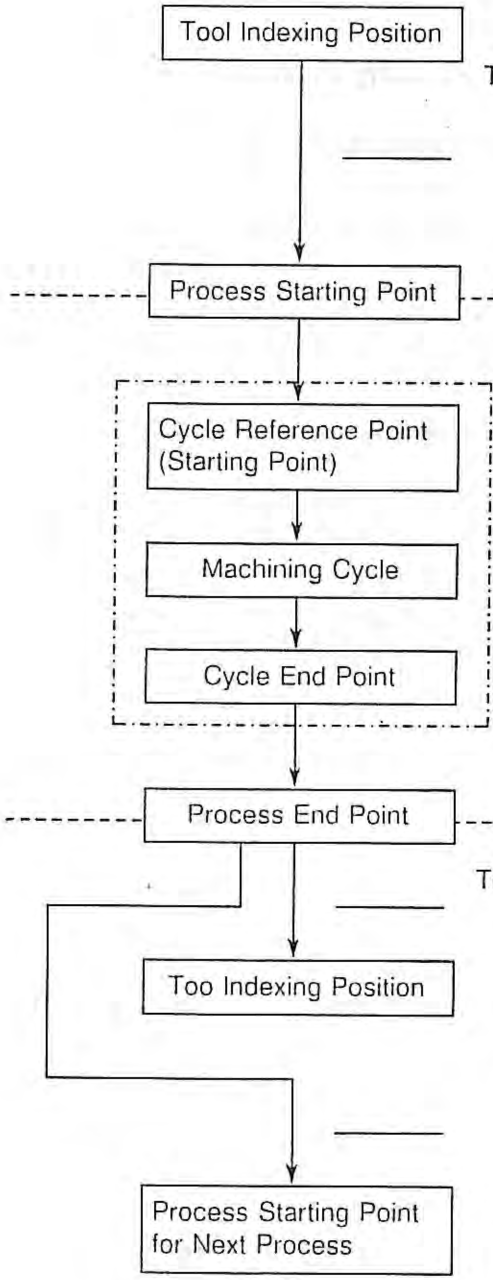
Press the function key [F7] (QUIT) after the completion of machining shape editing. The CRT will be switched from the MACH. SHAPE AMEND screen to the process check screen for multi-machining shape.

Note: *There are cases in which switching to the process check screen is disabled due to faulty or incorrect data on the MACH. SHAPE AMEND screen.*

10. Tool Path Control

10-1. Approach and Retract of Cutting Tool

In the part program created by the one-touch IGF, tools paths are defined as indicated below.



Tool approaching path in other than machining process

Cutting tool moves to the process starting process at the rapid feedrate. Designation of simultaneous two-axis movement or simultaneous one-axis movement is possible by the parameter. The tool approaching path is determined on the basis of portion to be cut in the process.

Tool paths in machining process

Cutting tool moves from the process starting point to the cycle reference point, and from the cycle end point to the process end point at the rapid feedrate in the simultaneous two-axis control mode. In finish turning (finishing cycle) or multiple machining, cutting tool moves from the cycle end point to the process end point at the rapid feedrate.

Tool retracting path in other than machining process

Cutting tool retracts from the process end point at the rapid feedrate. Designation of simultaneous two-axis movement or simultaneous one-axis movement is possible by the parameter. The tool retracting path is determined on the basis of portion to be cut in the process.

Cutting tool moves to the starting point of the next process at the rapid feedrate. The axis control mode, simultaneous two-axis or simultaneous one-axis, is determined on the basis of portions to be cut in the active and next processes.

For detailed explanation on the approach and retract of cutting tool, refer to Section 12, 8. "Tool Change Control".

10-2. Tool Path Control

In the process edit mode of the one-touch IGF, it is possible to designate the following data for each of the machining processes on the MACH. SHAPE AMEND screen.

a) Data to be Designated

CUT PT	CP
APR. PT	APX
	APZ
RTN. PT	RTX*
	RTZ*

For the multi-machining process, designate either RTX or RTZ.

b) How to Designate

CUT PT CP

Five positions are available as cutting position. For the processes which have been created through the automatic process determination function, since the one-touch IGF automatically sets the data for CUT PT CP, no data input is required. However, for the processes which have been edited or newly created in the process edit mode, enter the data.

1 = OUTSIDE .... OD longitudinal cutting
2 = OUTSIDE (BACK) ..... OD back end face cutting
3 = FRONT .... End face cutting
4 = INSIDE .... ID longitudinal cutting
5 = INSIDE (BACK) .... ID back end face cutting

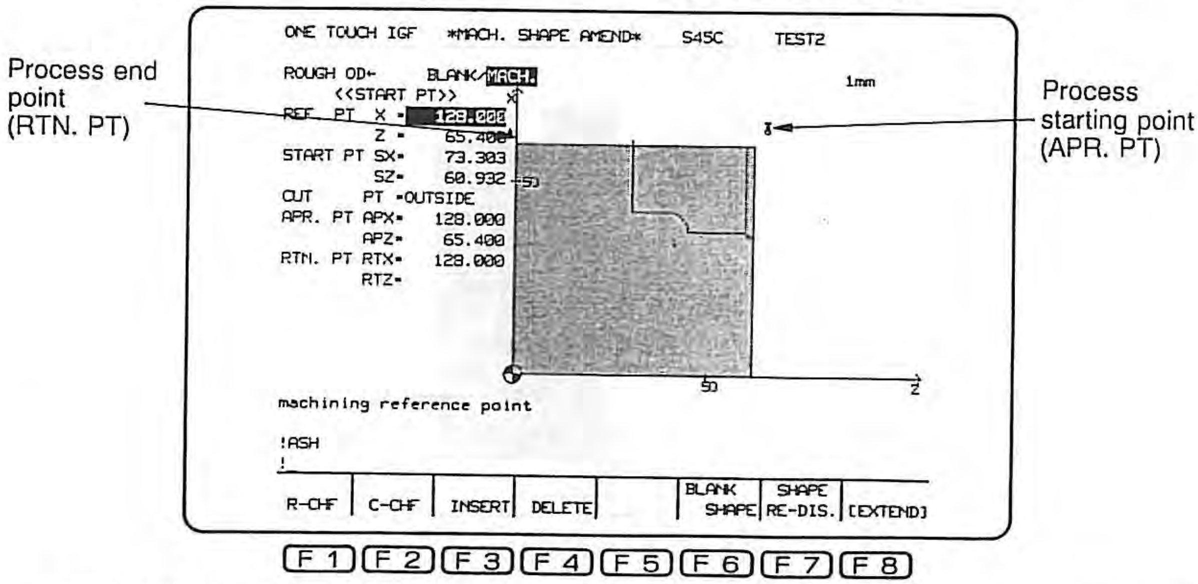
This determines the tool approaching and retracting paths.

APR. PT APX, APZ

RTN. PT RTX, RTZ

Process starting point and process end point for each process are set. For the processes which have been created through the automatic process determination function, although the one-touch IGF automatically sets the data for these input items, check the tool path on the TEST screen and correct it if needed.

Process starting point and process end point of the rough turning, copying, and finish turning processes are displayed on the MACH. SHAPE AMEND screen.



For the relationship between APR. PT, RTN. PT, CUT PT, and the actual tool approaching and retracting paths, refer to Section 12, 8. "Tool Path Control".

11. Process Test

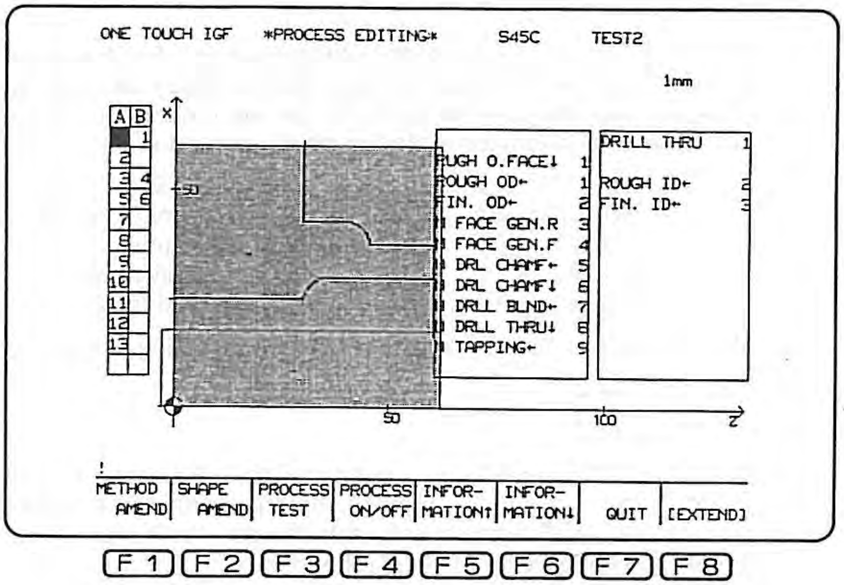
This is the function to simulate the part program being edited through the one-touch IGF on the CRT using animation.

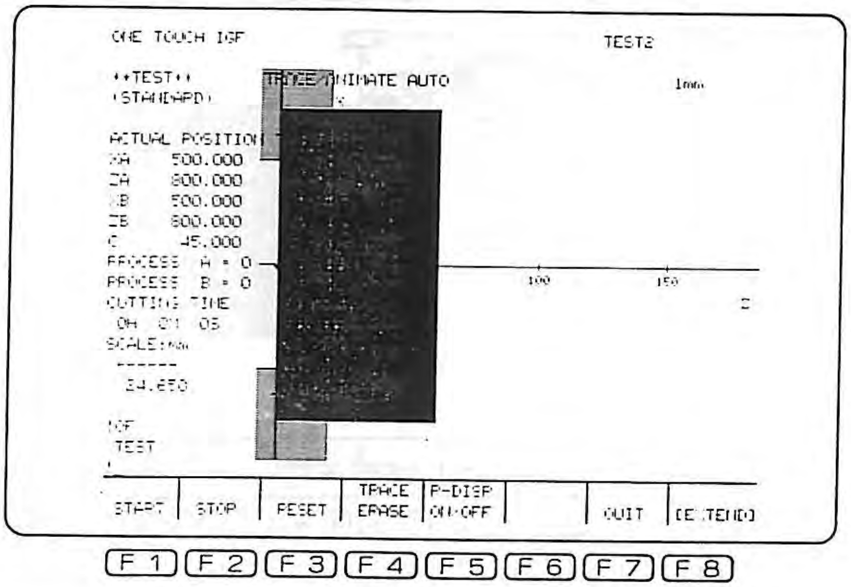
a) How to Designate

Press the function key [F3] (PROCESS TEST) on the process check screen. The TEST screen will be displayed.

After the completion of process test, press the function key [F7] (QUIT). The CRT will be returned to the process check screen.

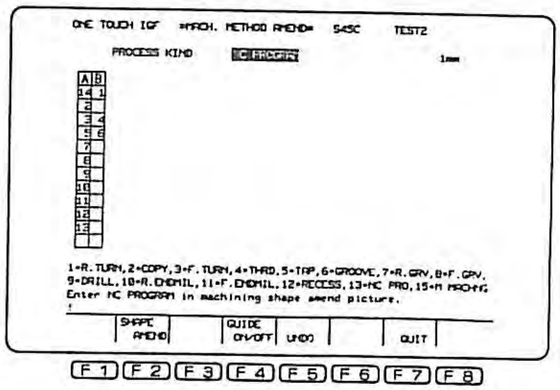
For details, refer to Section 10, "Process Test".

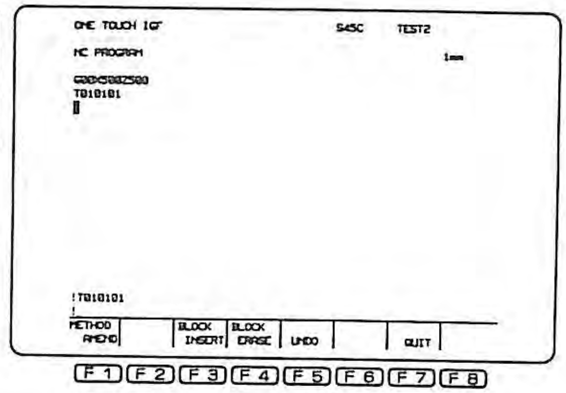




12. Editing the NC Program

By designating "13 = NC PRO" for PROCESS KIND on the MACH. METHOD AMEND screen, arbitrary G code, M code, or CALL command can be input between the processes in multiple blocks. The explanation that follows is the procedure to edit the process which is directly input as an NC program.





(1) Following the explanation in Section 11, 4. "Editing the Machining Method (turning, special machining)", move the cursor to the blank process where an NC program process is newly created, and press the function key [F1] (METHOD AMEND). The MACH. METHOD AMEND screen will be displayed.

(2) Designate "13 = NC PRO" for PROCESS KIND.

(3) Press the function key [F2] (SHAPE AMEND). The MACH. SHAPE AMEND screen will be displayed. Input an NC program on this screen.

- The NC program to be input is first displayed on the console line at the bottom of the screen. When the **WRITE** key is pressed after one block of the program has been input, the entered block of the NC program will be displayed on the cursor-located position on the screen.
- To input multiple blocks, first move the cursor downward and then enter the next block.
- When the screen has become full, advance the screen page using the page key and input the program. (The screen does not advance automatically.)
- One NC program can contains up to 1,024 characters.

  A space or return (**WRITE**) is also recognized as one character.

  Sequence number is automatically assigned.
- Before the input of NC program is started, coolant is turned off and the turret is retracted to the turret indexing position.
- Press the function key [F3] (BLOCK INSERT). A blank line is inserted before the cursor-located line.
- Press the function key [F4] (BLOCK ERASE). The cursor-located line is deleted.
- Press the function key [F5] (UNDO). The entered NC program data is all cancelled.

  Press the function key [F7] (QUIT). The CRT is returned to the start-up page of the PROCESS EDITING screen. The system checks the entered NC program contains one or more block.

13. Editing the Gauging Process (Optional: in-process gauging specification)

Using the one-touch IGF with the in-process gauging specification, a gauging cycle program can be created only by designating the gauging point for the gauging process.

For the procedure to edit the gauging process, refer to Section 21, "In-process Gauging Specification".

14. Machining Kind (Process Kind) List

PROCESS KIND	CUTTING DIRECTION	METHOD	HOLE POSITION	Remarks
R. TURN COPY F. TURN	OD← ID← OD FACE↓ ID FACE↑ MID OD← MID ID← MID FACE↓ MID OD→ MID ID→ MID FACE↑ OD→ ID→			
THRD	OD← ID← FACE↓ OD→ ID→ FACE↑			
TAP				
GROOVE	OD↓ ID↑ FACE←			
R. GRV	WD OD↓ WD ID↑ WD FACE← V OD↓ V ID↑ V FACE←			
DRILL	BLIND (HSS, CRBD) THRU (HSS, CRBD) CENTRNG			
R. ENDMIL F. ENDMIL				

PROCESS KIND	CUTTING DIRECTION	METHOD	HOLE POSITION	Remarks
RECESS	OD↙ ID↖ OD↘ ID↗			
NC PRO				
GAUGING (optional: in-process gauging)	O RAD I RAD O DIA I DIA O FACE I FACE			
M MACHNG	M DRILL ↓	BLIND THRU CENTRNG CHAMFER	FREE POSIT ARC	(optional: multi-machining specification)
	M BORING ↓		FREE POSIT ARC	(Same as above)
	M TAP		FREE POSIT ARC	(Same as above)
	M DRILL←	BLIND THRU CENTRNG CHAMFER	FREE POSIT ARC	(Same as above)
	M BORING←		FREE POSIT ARC	(Same as above)
	M TAP		FREE POSIT ARC	(Same as above)
	M REAMING←		FREE POSIT ARC	(Same as above)
	M REAMING ↓		FREE POSIT ARC	(Same as above)
	M FACE KEY M SIDE KEY			(Same as above)
	M ENDMILL ↓		FREE POSIT ARC	(Same as above)

PROCESS KIND	CUTTING DIRECTION	METHOD	HOLE POSITION	Remarks
M MACHNG	M ENDMILL← ID←		FREE POSIT ARC	(optional: multi-machining specification)
	M FACE GEN* M SIDE GEN*	R. ENDMIL F. ENDMIL R. MILL F. MILL		(Same as above)

* Enter the shape in the shape definition mode. The process is automatically determined.

SECTION 12 PART PROGRAM CREATION

1. What is the Function of "Part Program Creation"?

The one-touch IGF system creates a part program from the IGF data after the completion of the following operations.

Shape definition

Process determination

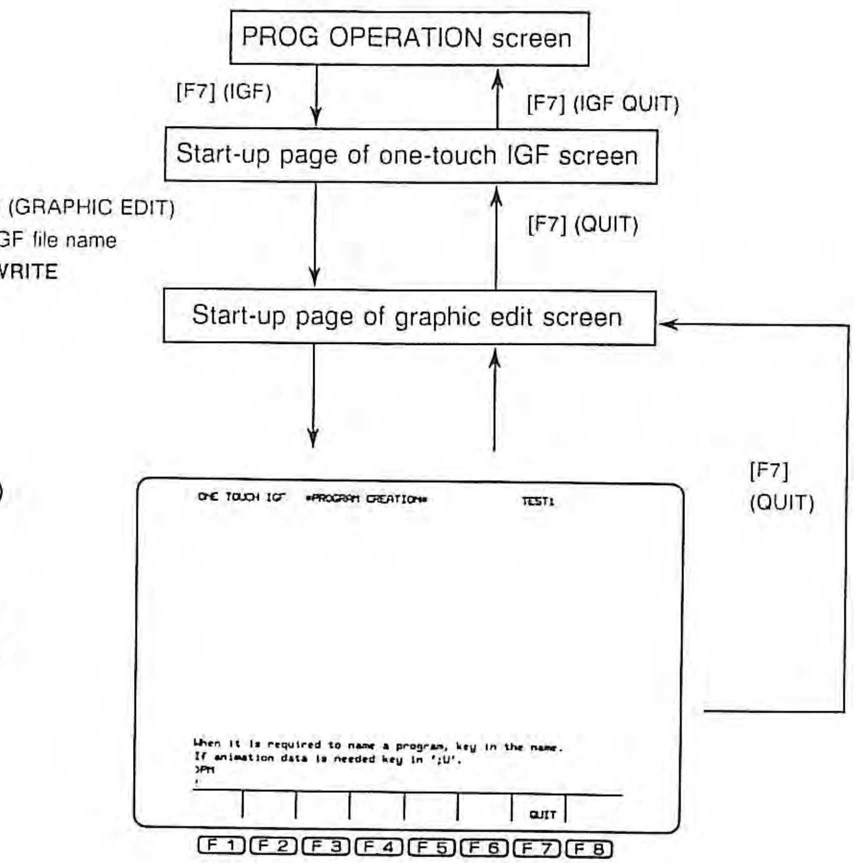
Process editing

Process test

A part program is assigned a file name and stored in the bubble memory.

Screen Transfer:

The PROGRAM CREATION screen is called from the PROG OPERATION screen as diagramed below.



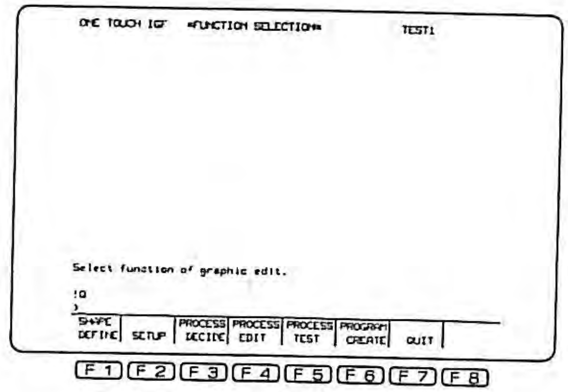
Press the function key [F7] (QUIT) when program creation is not required.
After the completion of a part program, the CRT will be returned to the start-up page of the graphic edit screen.

2. Program Creating Procedure

The program creating procedure is explained below.

Operating Procedure:

1) Press the function key [F6] (PROGRAM CREATE) on the start-up page of the graphic edit screen. The CRT will be changed to the PROGRAM CREATION screen.



2) Enter the file name and the program name.

For example, when the file name is "SHAFT.MIN", key in [S], [H], [A], [F], [F], [T], and press the **WRITE** key.

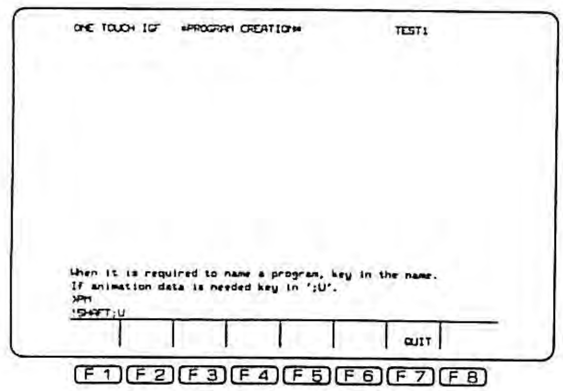
Press the function key [F7] (QUIT) when program creation is not required.

To assign a program name, enter "SHAFT.MIN,O1000" and press the **WRITE** key. When the program is stored as a sub program (extension name: .SUB) or a system sub program (extension name: .SSB), the program name (program number) must be entered.

Example: SHAFT.SUB,O1000 **WRITE**

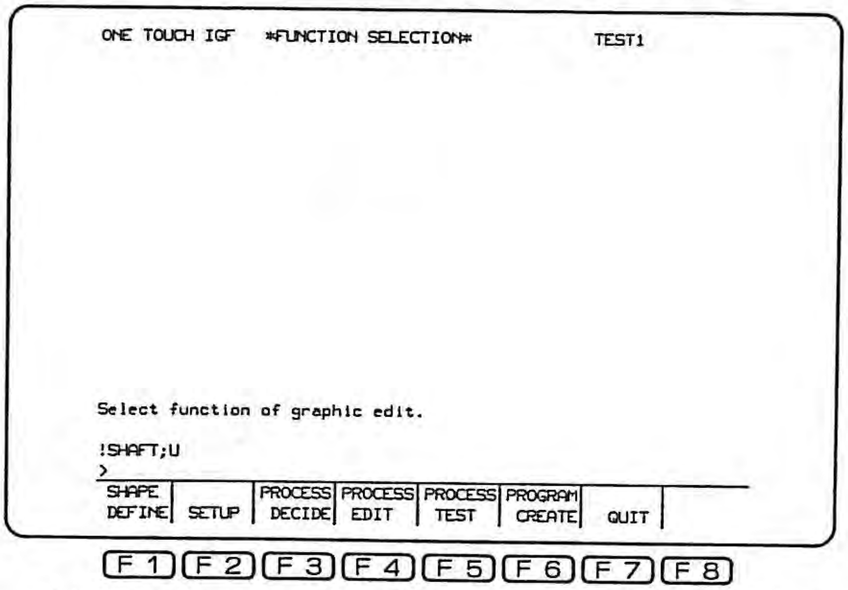
When ";" and "U" are entered following the file name and the program name and the **WRITE** key is pressed after that, graphic data is entered to the part program to be created.

When the **WRITE** key is pressed without keying-in ";" and "u", graphic data is not entered to the part program to be created.



3) A part program is created from the IGF data.

The CRT will be returned to the start-up page of the graphic edit screen.



Note 1: *If the remaining capacity of the bubble memory is not sufficient, a part program cannot be created even when the file name is entered. In this case, suspend program creating operation and proceed to either a) or b) indicated below.*

a) Reduce the number of machining processes in the process edit mode.

b) Terminate the one-touch IGF and select the EDIT AUX. (program operation) mode.

Delete the unnecessary files in the bubble memory, restart the one-touch IGF, and create a part program.

Note 2: *To create a part program from the existing IGF file, call the start-up page of the graphic edit screen following the procedure indicated below, and then proceed with the steps 1) to 3) above.*

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

b) Enter the IGF file name and press the ***WRITE*** *key.*

Note 3: *To create a part program with graphic data, never fail to designate the shape of chuck and tailstock center on the SETUP screen before starting the operation.*

3. Sequence Numbers and Sequence Names

(1) Sequence Numbers

The sequence numbers are assigned according to the setting for the one-touch IGF integer parameter No. 1 SEQUENCE NO. INCREMENT. Refer to Section 5, "One-touch IGF Parameter Setting".

(2) Sequence Names

The sequence names are output in a block in the format indicated below at the beginning of the individual processes to identify the tools used in the corresponding process.

Example:

NA T06

Tool number

A-turret/B-turret designation
Fixed at A for one-saddle models

When a tool is designated with the tool group number with thee tool life management specification model.

Example:

NA T04

Tool number

A-turret/B-turret designation
Fixed at A for one-saddle models

The sequence names are output at the beginning of a new process for one-saddle models. For two-saddle or two-turret models, they are output at the block succeeding the G13 or G14 block when the turret is changed and at the beginning of a new process.

Note: *When the same tool is used in more than one process, the same sequence name is output for multiple processes.*

4. Output of Modal M Codes

Generally, the modal M codes are not output for individual processes when they are to be designated in more than one process. For example, when the spindle rotates in the forward direction in both the first and the second processes, M03 is not output in the second process.

However, if the program is edited in units of processes, this output method is inconvenient because the process to be moved might not contain the necessary M codes.

To eliminate such inconvenience, the one-touch IGF integer parameter No. 27 MODAL M CODE FORCED OUTPUT (EACH PROCESS) is provided. When "1" is set, the NC program is output in units of processes so that each unit contains all necessary information.

M codes influenced by the setting of one-touch IGF parameter No. 27 MODAL M CODE FORCED OUTPUT (EACH PROCESS) are as follows.

(1) Spindle Rotation (M03/M04/M05)

When the C-axis is not connected, a process begins with M03 or M04 and ends with M05.

(2) Coolant (M08/M09)

When coolant supply is designated, a process begins with M08 and ends with M09.

M08 and M09 are not output when coolant supply is not designated.

(3) Tailstock Barrier Check (M20/M21)*

A process begins with M21 and ends with M20.

(4) Chuck Barrier Check (M24/M25)*

A process begins with M25 and ends with M24.

(5) Tool Interference Check (M28/M29)*

A process begins with M29 and ends with M28.

(6) Ignoring Rapid Feed for the Load Monitor Specification (M215/M216)

A process begins with M216 (ignore ON) and ends with M215 (ignore OFF).

(7) C-axis Connect (M109/M110)

For multi-machining processes, a process begins with M110 (C-axis connect) and ends with M109 (cancel of C-axis connect).

(8) Rotation of M-tool Spindle (M12/M13/M14)

When M-tool is used, a process begins with M13 or M14 and ends with M12.

(9) Spindle Speed Change Gear (M41/M42/M43/M44)

M41, M42, M43, or M44 is output in turning processes. No M code is output unless the data is set for the one-touch IGF integer parameter.

(10) M-tool Spindle Speed Change Gear (M241/M242)

When M-tool is used, M241 or M242 is output. No M code is output unless the data is set for the one-touch IGF integer parameter.

(11) Turret Designation (G13/G14)

For two-saddle or two-turret models, G13 or G14 is output. When more than one process is continuously executed on the same turret, the output of G13 or G14 is not omitted.

(12) Feed per Minute (G94/G95)

A process begins with G94 and ends with G95

* These M codes are not output unless the output is designated by the one-touch IGF integer parameter.

5. Automatic Generation of Optional Stop (M01)

For machining only one piece of a workpiece, or production of small lot of workpieces, it is necessary to halt the machining processes at the end of each process to measure the machined dimensions, etc.

In such cases, set "1" for the one-touch IGF integer parameter No. 28 M01 OUTPUT. OR NOT (G77) M01 is automatically output at the end of individual processes. Output is made just after the turret has been retracted to the turret indexing position after the completion of cutting. During the simultaneous 4-axis control mode in two-saddle models, process execution is synchronized by P codes.

6. Spindle Control

(1) Spindle Rotating Direction Control

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

b) If the spindle rotating direction change command is designated, the spindle is stopped once before the rotating direction is changed.

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

Note: *For the control equipped with the mirror image function, if the actual spindle rotating direction is changed, the spindle is once stopped. Then, the spindle rotating direction is changed.*

(2) Spindle Speed Control

Constant Peripheral Speed Mode ON:

a) The spindle is started in the constant speed mode. The spindle speed at the cycle reference point is used for rotating the spindle.

b) When the positioning to the cycle reference point has been completed, the constant peripheral speed mode is activated.

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

Constant Peripheral Speed Mode OFF:

a) The spindle is rotated at such a speed that the peripheral speed at the maximum cutting diameter is identical to the designated peripheral speed.

Note: *Whether or not the constant speed mode is selected is set by the one-touch IGF parameter. For drilling and thread cutting operations, however, the constant speed mode is not applied irrespective of the setting of this parameter.*

(3) Spindle Speed Range Control

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

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

c) If the one-touch IGF parameter setting is improper, spindle speed range selection is not executed. In other words, if the setting of maximum spindle speed is equal to or smaller than that of minimum spindle speed in a range, this spindle speed range is ignored.

7. 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.

8. Tool Change Control

The tool change control refers to the control in which the air cutting cycle and the T code are controlled.

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

The air cutting cycle control includes the following operations:

(1) Retract to the initial turret indexing position

(2) Approach to the cycle reference point

(3) Move from the process starting point to the cycle reference point (starting point)

(4) Move from the cycle end point to the process end point

(5) Retract from the process end point to the turret indexing position

(6) Move to the turret indexing position for the next process

(7) Move to the cycle reference point of the next process when turret indexing is not necessary

(8) Final retract

Three types of data are used to control the air cutting cycle as indicated below.

a) Clearance boundary

b) Classification of turret indexing position

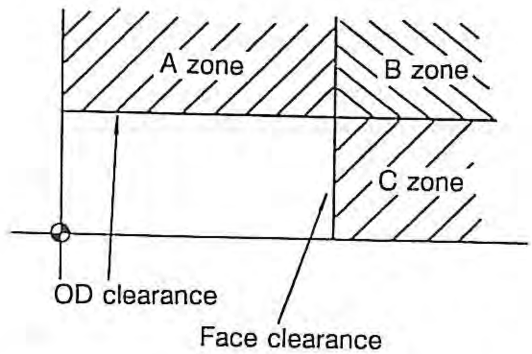
c) Cutting direction

a) Clearance boundary

OD clearance boundary	a
ID clearance boundary	b
Face clearance boundary	c
Multi-machining OD clearance	a'
Multi-machining face clearance	c'

Values for the above items can be set by the one-touch IGF dimension parameter.

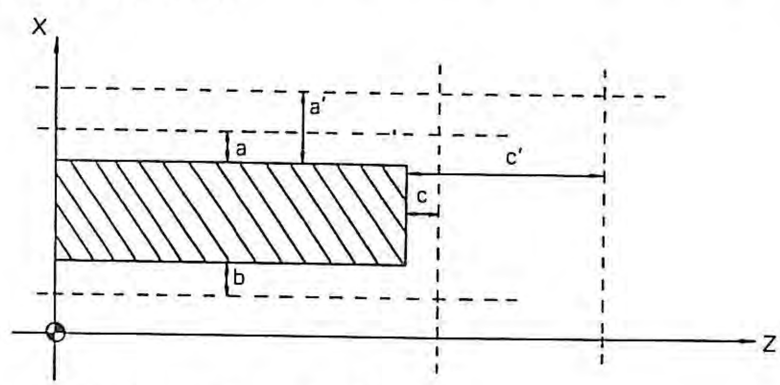
They indicate the boundary shown in the diagram below.



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

b) Classification of turret indexing position

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



Assume that the tool indexing position is (ZT, XT).

A zone:

Turret indexing position ZT ≦ Face clearance boundary value

Turret indexing position XT > OD clearance boundary value

B zone:

Turret indexing position ZT > Face clearance boundary value

Turret indexing position XT > OD clearance boundary value

C zone:

Turret indexing position ZT > Face clearance boundary value

Turret indexing position XT ≦ OD clearance boundary value

c) Cutting direction

Five cutting directions are provided depending on the portion to be cut.

1 = OUTSIDE	OD longitudinal cutting
2 = OUTSIDE (BACK)	OD back end face cutting
3 = FRONT	End face cutting
4 = INSIDE	ID longitudinal cutting
5 = INSIDE (BACK)	ID back end face cutting

(1) Retract to the Initial Turret Indexing Position

a) Before the execution of cutting process is started, the turret is retracted to the turret indexing position for the first cutting process. This is accomplished in the simultaneous two-axis control mode.

b) With two-saddle models, first A-turret then B-turret is retracted to the turret indexing position.

c) With the mirror image specification, retraction is made to the turret indexing 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 designated either in the automatic process determination operation or in the process edit mode and the tool offset number.

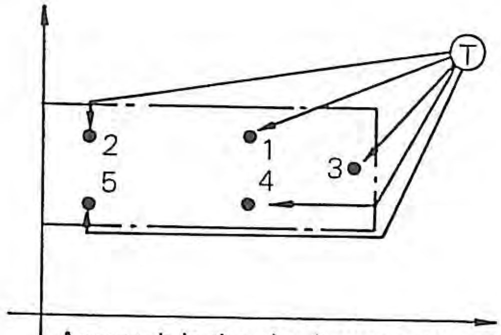
b) The approaching tool path is determined based on the classification of turret indexing position and the air cutting direction.

c) The approach can be made either in the simultaneous two-axis control mode or in the simultaneous one-axis control, depending on the setting of the one-touch IGF parameter.

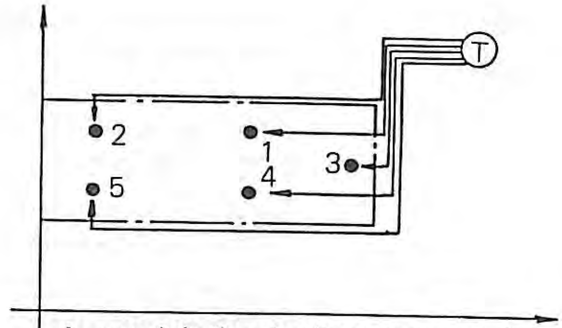
d) Note that there are two variations for tool approaching path in the simultaneous two-axis control mode, depending on the turret indexing position and the current direction.

Turret indexing position ⟶ Clearance boundary

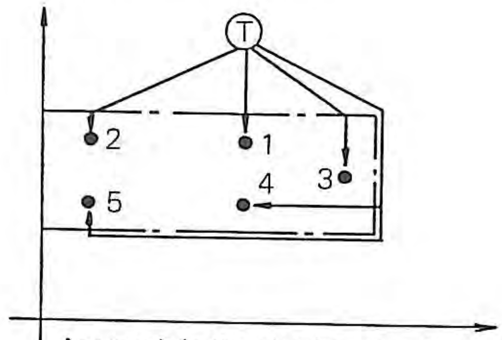
Turret indexing position ⟶ Clearance boundary ⟶ Clearance boundary



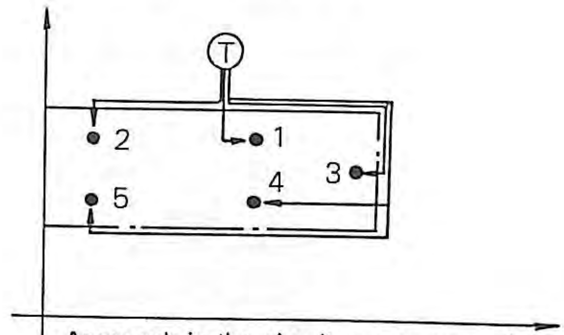
Approach in the simultaneous two-axis control mode in B zone



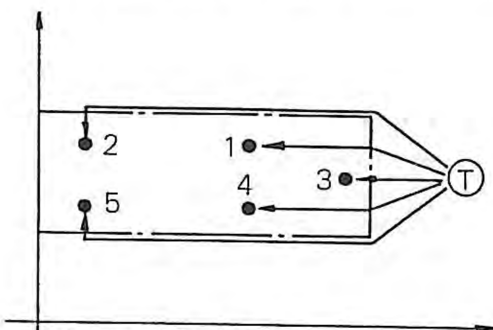
Approach in the simultaneous one-axis control mode in B zone



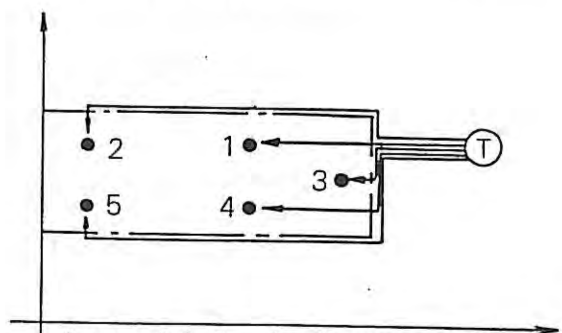
Approach in the simultaneous two-axis control mode in A zone



Approach in the simultaneous one-axis control mode in A zone



Approach in the simultaneous two-axis control mode in C zone



Approach in the simultaneous one-axis control mode in C zone

—·—·—·— Clearance boundary

(T) Turret indexing position

● Process starting point for each portion to be cut

(3) Move from the Process Starting Point to the Cycle Reference Point (starting point)

a) The movement is made at the rapid feedrate in the simultaneous two-axis control mode. Note an axis movement command is not designated in the following cases:

The X-coordinate value of the cycle reference point (REF. PT) is identical to the X-coordinate value of the cutting starting point (START PT), an X-axis movement command is not designated.

The Z-coordinate value of the cycle reference point is identical to the Z-coordinate value of the cutting starting point, an Z-axis movement command is not designated.

The axis is directly positioned to the infeed point in the G01 mode with the tool nose radius compensation function activated.

(4) Move from the Cycle End Point to the Process End Point

The same as (3).

(5) Retract from the Process End Point to the Turret Indexing Position

a) If the tool used for the next process is different from the tool used in the present process, the turret is retracted from the process end point to the turret indexing position after the completion of cutting in the present proces.

b) When retracting to the turret indexing position, tool offset amount is not taken into consideration. (T**00)

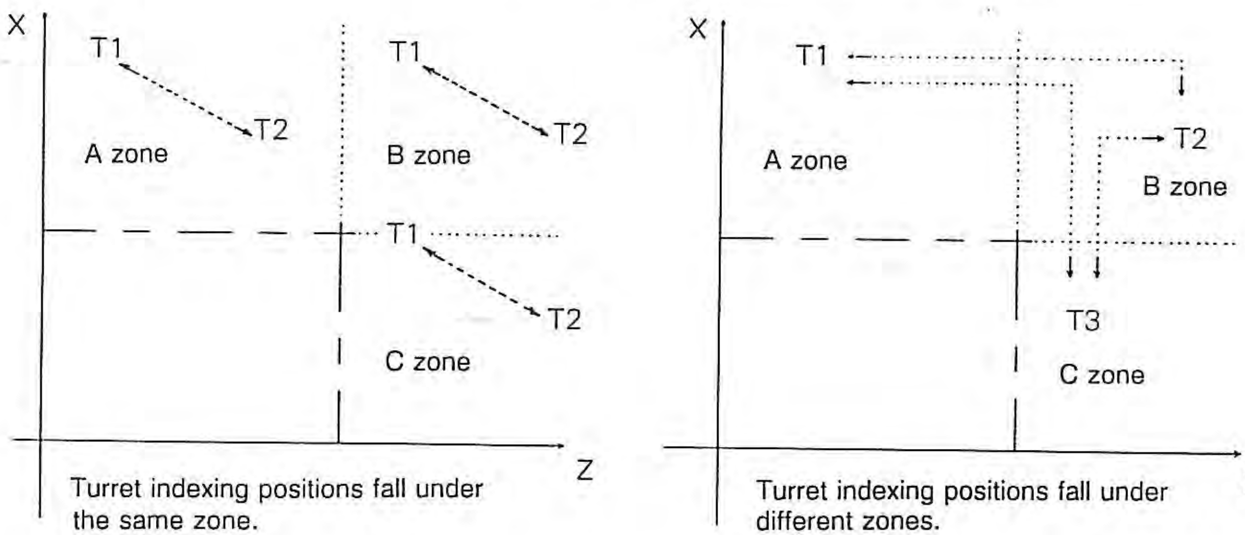
c) The retracting tool path is determined based on the classification of turret indexing position and the air cutting direction.

d) Actually, the reverse of the path in (2) is taken.

(6) Move to the Turret Indexing Position for the Next Process

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

b) The tool path is determined based on the classification of next and present turret indexing positions. That is, if the turret indexing positions fall under the same classification, the movement is made in the simultaneous two-axis control mode. If they fall under different zones, the movement is made in the simultaneous one-axis control mode.



Turret indexing positions fall under the same zone.

Turret indexing positions fall under different zones.

(7) Move to the Cycle Reference Point of the Next Process when Turret Indexing Is Not Necessary

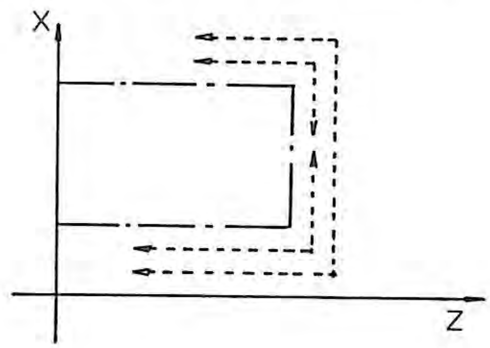
a) If the same tool as used in the present process is used in the next process, the turret moves to the new process starting point.

b) The tool path is determined based on the cutting directions of next and present processes.

c) The movements are made along the clearance boundary in the simultaneous one-axis control mode.

When the same portion is to be cut, the movements are made in the simultaneous two-axis control mode.

d) Tool offset may be switched during axis movements



(8) Final Retract

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

b) The retracting path is the reverse of the approaching path in (2).

WARNING

Tool paths explained above are those of "tool tip". Therefore, when a cutting tool with a large-diameter toolholder or a cutting tool with a wide insert is used, there are cases in which the cutting tool interferes with the workpiece. In such cases, increase the values set at the one-touch IGF dimension parameters No. 1 OD CLEARANCE, 2 ID CLEARANCE, and No. 3 FACE CLEARANCE, and execute the automatic process determination operation again.

SECTION 13 TERMINATION OF ONE-TOUCH IGF OPERATION

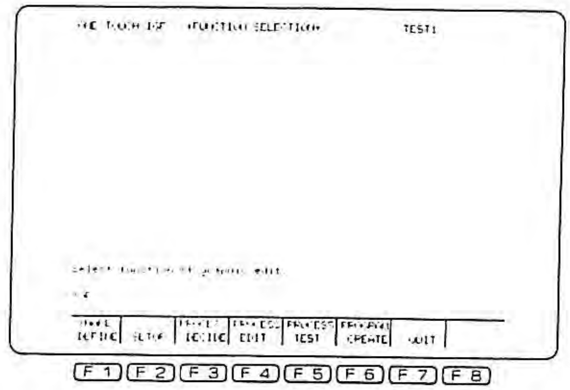
The one-touch IGF operation is terminated in the following order.

(1) Terminating the Graphic Edit Mode

The start-up page of the graphic edit screen will be displayed when the following operations have been terminated:

- Shape definition
- Setup input
- Process determination
- Process editing
- Process test
- Program creation

Press the function key [F7] (QUIT) on this screen. The start-up page of the one-touch IGF screen will be displayed.

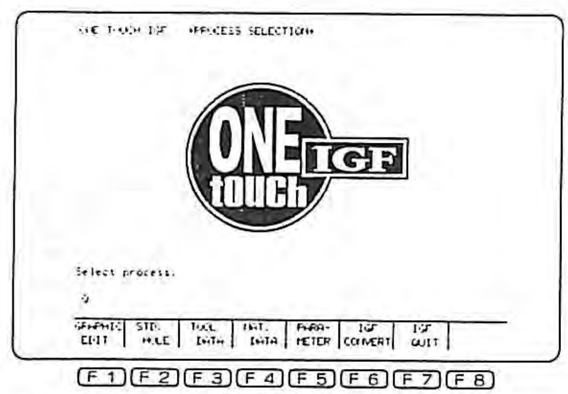


(Start-up page of the graphic edit screen)

(2) Terminating the One-touch IGF Mode

Press the function key [F7] (IGF QUIT) on the start-up page of the one-touch IGF screen.

This terminates the one-touch IGF mode and the CRT returns to the PROG OPERATION screen.



(Start-up page of the one-touch IGF screen)

Note: *When the graphic edit mode is terminated, the IGF file created is stored to the bubble memory. However, if the remaining capacity of the bubble memory is not sufficient, terminating the graphic edit mode is disabled because the IGF file cannot be stored to the bubble memory. In this case, establish the program operation (EDIT AUX.) mode following the description in Section 16, "One-touch IGF Temporary Quit and Recovery Function", and delete the unnecessary files in the bubble memory. Then, restart the one-touch IGF for the same IGF file and terminate the one-touch IGF following the procedure above.*

SECTION 14 DATA INPUT USING MATH OPERATION

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.

1. Arithmetic and Function Operation Symbol

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

Operation Symbol List

Symbol	Contents	Example	Remarks
+ −	Plus sign Minus sign	+ 12.34 [− SIN 90]	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 (×). / indicates the division symbol (÷).
R S Q R T	Square root	R **SP** 30 S Q R T [30 − 20]	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.
S S I N	Sine	S [45 * 2] S I N **SP** 60	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.
C C O S	Cosine	C **SP** 30 C O S [15 + 45]	
T T A N	Tangent	T [45 − 15] T A N **SP** [15 * 3 / 2]	

Symbol	Contents	Example	Remarks
A S A S I N	Arc sine	**A S SP** 0.5 **A S I 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.
A C A C O S	Arc cosine	**A C** [0.8 * 0.6] **A C O S SP** 0.45	
A T A T A N	Arc tangent (−90° - 90°)	**A T SP** 45 **A T A N** [45 * 2]	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.
A T 2 A T A N 2	Arc tangent (−180° - 180°)	**A T 2** [0.5, 1.5] **A T A N 2** [0.45, 1]	Designation should always be AT2 [numerator, denominator]. Symbols [ ] cannot be omitted.

Note 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: S30 → Calculation error (No SP between "S" and "30")

Note 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 [180 + [S * [30 − 5] − 20 * [6 − 2]]]*

Note 3: *Calculation order follows usual arithmetic operation rules:*

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

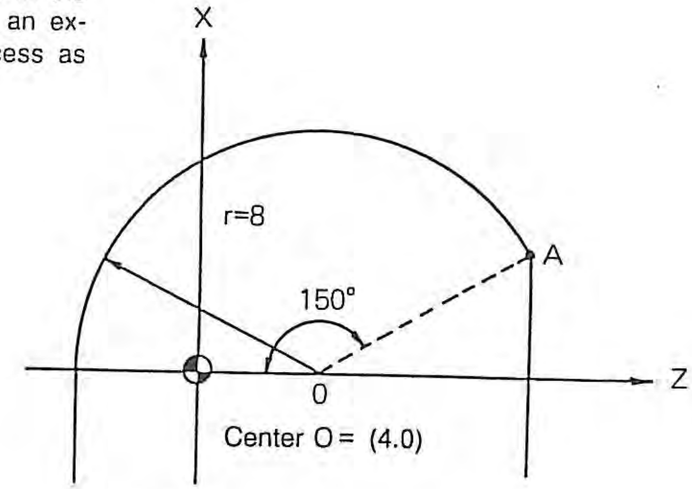
If operations of the same calculation priority are used, calculation is made from the leftmost operations in order.

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

1) *3 / 4 = 0.75*
2) *SIN0.75 = 0.013*
3) *COS5 = 0.996*
4) *0.013 + 0.996 = 1.009*
5) *2 × 1.009 = 2.018*
6) *2018 ÷ 6 = 0.336*
7) *1 + 0.336 = 1.336*

2. Input Examples

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



Point A

Z = 4 + 8 × cos (180 − 150)
X = 8 × sin (180 − 150)

(1) Move the cursor to 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 0 − 1 5 0] **WRITE**

The input is displayed at the console of the CRT as 4 + 8 * C [180 − 150]. The result of calculation is displayed at the data column.

(3) Move the cursor to 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 0 − 1 5 0] **WRITE**

The input is displayed at the console of the CRT as 8 * S [180 − 150]. The result of the calculation is displayed at the data column.

Note: *Input of symbol "["*

Press T[while holding down the ***UPPER CASE*** *key.*

Input of symbol "]"

Press U] while holding down the ***UPPER CASE*** *key.*

SECTION 15 DATA INPUT USING ADDITION FUNCTION

To set dimensions, feedrates, positions, etc. in the graphic edit operation, parameter setting operation, material data setting operation and tool data setting operation, it is possible to add or subtract the designated value to or from the cursor-located, in addition to the conventional direct numerical data input method.

Data Input Procedure for Addition:

(1) Move the cursor to the data for which the value is to be added.

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

A D SP numerical value or expression WRITE

The keyed-in value or the result of calculation is added to the cursor-located data.

Note 1: *If the keyed-in numerical value 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 numerical value or the result of calculation is set as is.*

Note 3: *For details of expression, refer to Section 14.*

Example:

A D SP 1 5 .... Numerical value "15" is added to the cursor-located data.

A D SP – 0 . 1 .... Numerical value "0.1" is subtracted from the cursor-located data.

*A D SP 0 . 1 * 3 ...... Numerical value "0.3" is added to the cursor-located data.*

A D SP – 1 / 5 .... Numerical value "0.2" is subtracted from the cursor-located data.

SECTION 16 ONE-TOUCH IGF TEMPORARY QUIT AND RECOVERY FUNCTION

There are cases in which one-touch IGF operation is required to be temporarily suspended during graphic edit operation to make it possible to carry out some other operations (switching to a different operation mode such as zero set for example) or when graphic edit operation cannot be completed up to the end due to insufficient bubble memory space remaining.

In such cases, if is of course possible to quit the one-touch IGF mode at the graphic edit operation currently executed. However, this requires several steps to be executed and is not effective when the remaining capacity of the bubble memory is not sufficient.

The one-touch IGF temporary quit and recovery function allows the graphic edit operation, interrupted by mode switching, etc. without quitting the one-touch IGF operation to be recovered from the point where the one-touch IGF mode has been suspended.

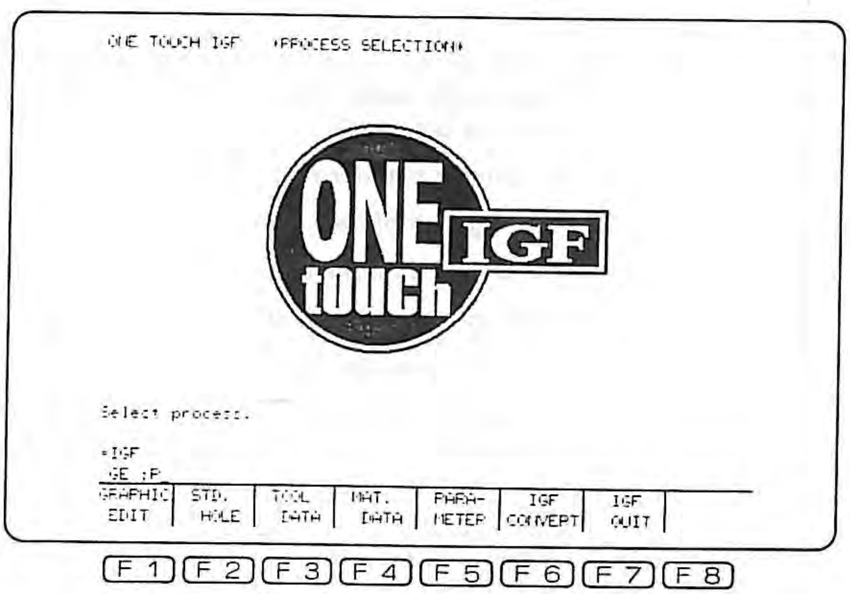
How to Quit:

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

How to Recover:

1) After the completion of the operation in other operation mode, first select the EDIT AUX mode by pressing the **EDIT AUX.** key, and then press the function key [F7] (IGF).
2) Press the function key [F1] (GRAPHIC EDIT) and key in ";", "R" instead of a file name, then press the **WRITE** key.

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



It is also possible to change the name of the IGF file for which graphic edit operation is restarted. Follow the procedure below when recovering graphic edit operation.

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

2) Key in the new file name.

3) Enter [;] and [R], and press the **WRITE** key.

This starts graphic edit operation with the IGF file name changed.

CAUTIONS

1) *If the one-touch IGF is interrupted during the execution of shape element designation, the shape element data up to the preceding shape element designation operation is saved. The shape element data being designated is lost.*

2) *If the one-touch IGF is interrupted while the data is newly registered for the process in the process edit mode, that process is lost when the graphic edit mode is recovered. Other processes remain.*

3) *Recovery of graphic edit operation is possible until control power is turned off.*

4) *When an IGF file which has the same name as the IGF file for which graphic edit operation has been restarted is present in the bubble memory, the following message will be displayed on the screen.*

   file exist overwrite? (Y/N)

   Key in "Y" and press the WRITE key to store the restarted IGF file to the bubble memory.

   Key in "N" and press the WRITE key to cancel the recovery of graphic edit operation.

5) *If any of the following operations is conducted in the one-touch IGF operation mode before recovering the graphic edit mode, recovery is disabled.*

   - *Setting of standard hole data (F2)*
   - *Setting of tool data (F3)*
   - *Setting of material data (F4)*
   - *Setting of one-touch IGF parameter (F5)*
   - *Program creation (F2)*

   Note that IGF CONVERT (F7: optional) operation does not influence the recovery of one-touch IGF operation.

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

SECTION 17 ANIMATION DATA

1. What is the Function of "Animation Data"?

Animation data is used to store the tool shape data or blank material shape data to the part program so that operations such as process test can be executed.

To provide a part program with animation data, key in ";" and "U" and press the **WRITE** key following the file name and the program name on the PROGRAM CREATION screen.

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 one-touch 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 screen.

The user graphic commands (UGC) are provided so that arbitrary blank shapes may be drawn on the graphic CRT. With the one-touch IGF system, the blank shape is automatically generated using the user graphic commands by the entry of the blank data through the interactive operations.

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 - 38
VTLFN[I]	Tool shape code	0 - 4
VTLA1[I]	Tool angle	0 - 360.000
VTLA2[I]	Edge angle	–360.000 - 360.000
VTLL[I]	Holder length/Sticking out length/Drill length	0 - 9999.999
VTLD[I]	Holder diameter/Drill diameter	0 - 9999.999
VTLW[I]	Tool width	0 - 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.

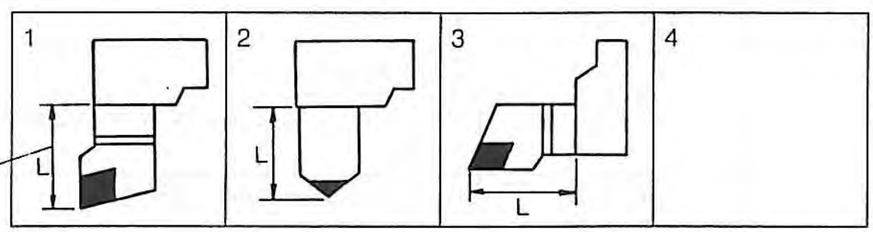
For the tool life management specification, the system variable name is assigned in the format VGR**[I]. The index [I] is the tool group number.

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↖
27 = M DRILL ↓
28 = M DRILL←
29 = BORING ↓
30 = BORING←
31 = TAP ↓
32 = TAP←
33 = REAMER ↓
34 = REAMER←
35 = END MILL ↓
36 = END MILL←
37 = FACE MILL ↓
38 = FACE MILL←

Tool Shape Code: Tool shape number which corresponds to the tool shape usable for the tool selected by the tool classification code.

(No. 1 through No. 3 for the example below.)



(ROUGH OD←, FINISH OD←)

Example:

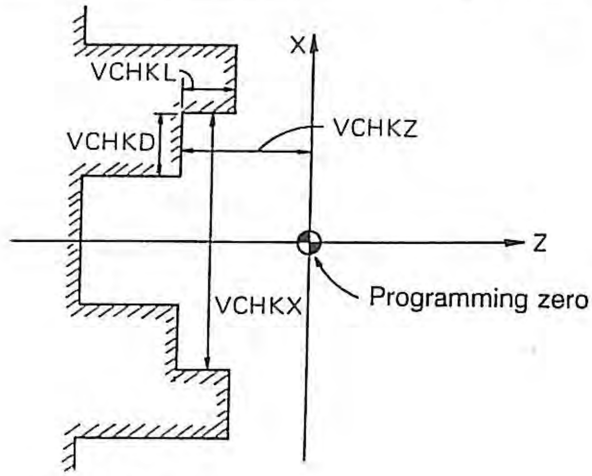
ROUGH OD←
TOOL NO. TN = 2
TOOL ANGLE AI = 80°
EDGE ANGLE A2 = 3°
STICKING OUT L = 40 mm

VTLIN[2] = 1
VTLFN[2] = 1
VTLA1[2] = 80.000
VTLA2[2] = 3.000
VTLL[2] = 40.000

1-2. Chuck Barrier Definition

The four system variables indicated below are used for defining the chuck barrier.

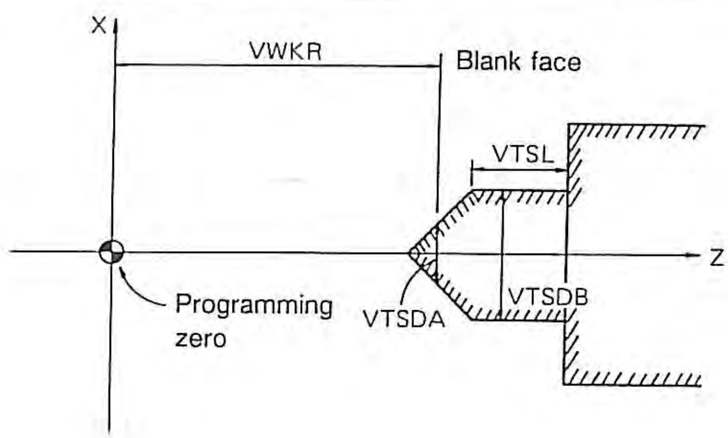
Variable Name	Contents	Data Size
VCHKL	Chuck jaw length	0 - 9999.999
VCHKD	Step on chuck jaw	0 - 9999.999
VCHKX	Gripping diameter	−9999.999 - 9999.999
VCHKZ	Distance from programming zero	−9999.999 - 9999.999



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	0 - 9999.999
VTSDA	Center hole diameter	0 - 9999.999
VTSDB	Tailstock spindle diameter	0 - 9999.999



2. Drawing Blank Shape

Blank material shape is defined as explained below using user graphic commands (UGC).

2-1. Definition Format

The format used for blank shape (side view) 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.

The format used for blank shape (front view) drawing is indicated below.
(Effective for multi-machining specification only)

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.

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.

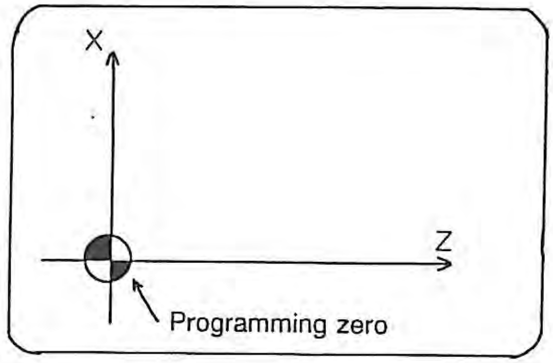
The explanation below provides the format of the representative commands.

(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 zero point.

The unit system used is:

Metric system . . . 1 mm
Inch system . . . . 0.1 inches



(2) Setting Drawing Point

POINT [Z_0, X_0]

Note: The underlined command 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 (Z_0, X_0) for the drawing and no actual drawing operation occurs.

This sets the last reference point (LP) for the drawing.

(3) Drawing Straight Line

LINE [Z_1, X_1] <,line-code>

A straight line is drawn from the last reference point (LP) of the drawing up to the commanded end point (Z_1, X_1). 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.

a) Line Code: 0 = 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 "0 (solid line)".

(4) Circle Drawing

CIRCLE [Z_1, X_1], [Z_c, X_c] <,rotation-direction> <,line-code>

An arc is drawn in the following conditions:

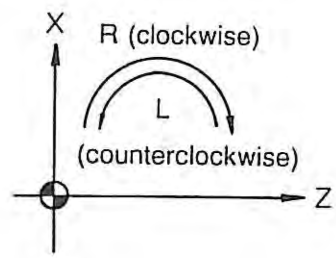
Starting point	:	LP
End point	:	(Z_1, X_1)
Center	:	(Z_c, X_c)

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.

a) Rotation Direction:

R: Clockwise
L: Counterclockwise

Default is "R (clockwise)".



b) Line Code: Eight codes, 0 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.

a) Number of Vertexes:

0: Circles or sectors
3: Triangles
4: Rectangles

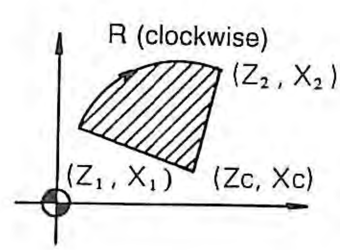
b) Vertex Coordinates:

The designation of coordinates depends on the number of vertexes specified.

Number of vertexes = 0

[Z_c, X_c], [Z_1, X_1]. [Z_2, X_2] <,rotation-direction>

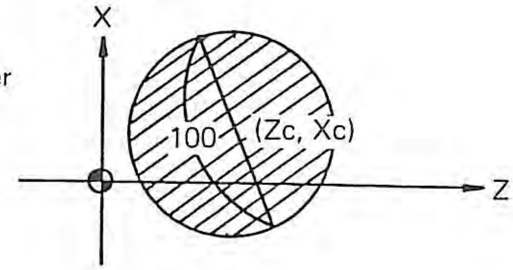
Z_c, X_c :	Center
Z_1, X_1 :	Start point
Z_2, X_2 :	End point



When the number of vertexes is zero (0), designation of diameter is also possible.

[Zc, Xc], 100, D

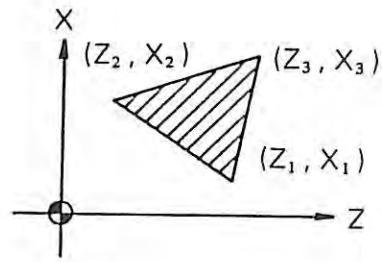
Zc, Xc	:	Center
100	:	Diameter
D	:	Code to indicate diameter designation



Number of vertexes = 3

[Z_1, X_1], [Z_2, X_2], [Z_3, X_3]

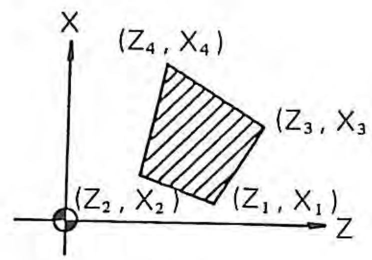
($X_1 \leqq X_2 \leqq X_3$)



Number of vertexes = 4

[Z_1, X_1], [Z_2, X_2], [Z_3, X_3], [Z_4, X_4]

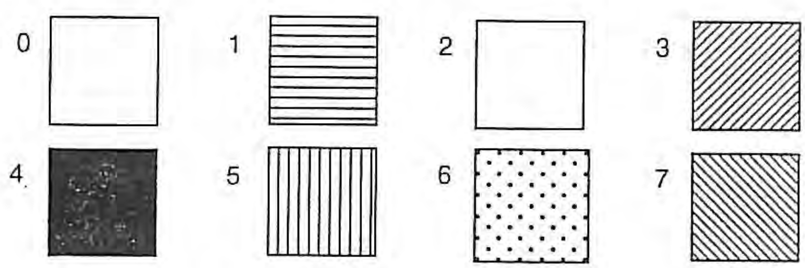
($X_1 \leqq X_2 \leqq X_3 \leqq X_4$)



Note that the coordinates of vertexes must be given in absolute values.

c) Tile Pattern:

0: 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-coordinates> <, 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.

a) Number of Vertexes:

0: Circles or sectors
3: Triangles
4: Rectangles

b) Reference Point Coordinates:

The coordinates of the reference point for defining a shape; [Z_0, X_0]

The values must be given in absolute values.

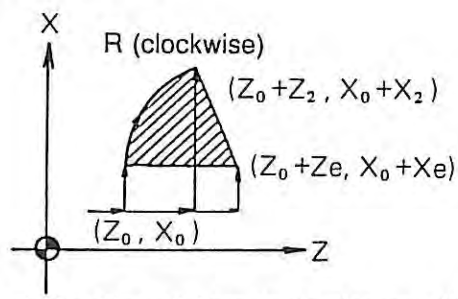
c) Vertex Coordinates:

The designation of coordinates depends on the number of vertexes specified.

Number of vertexes = 0

[Z_c, X_c], [Z_1, X_1], [Z_2, X_2] <, rotation-direction>

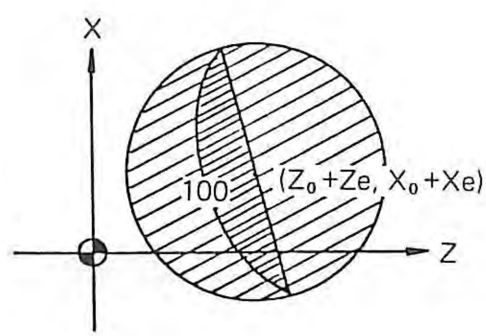
Z_c, X_c : Center
Z_1, X_1 : Start point
Z_2, X_2 : End point



When the number of vertexes is zero (0), designation of diameter is also possible.

(Z_c, X_c), 100, D

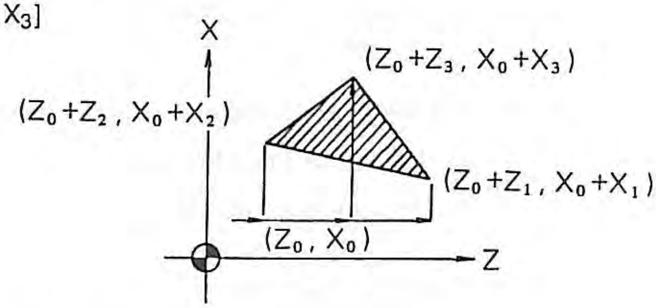
Z_c, X_c : Center
100 : Diameter
D : Code to indicate diameter designation



Number of vertexes = 3

[Z_1, X_1], [Z_2, X_2], [Z_3, X_3]

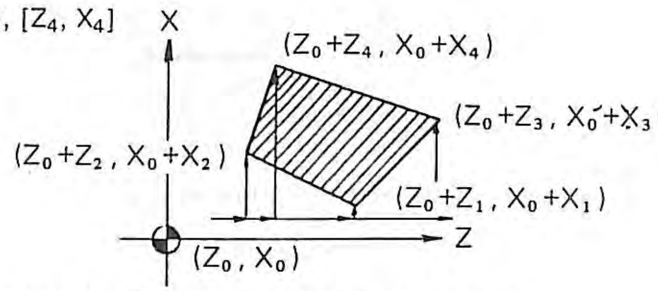
($X_1 \leqq X_2 \leqq X_3$)



Number of vertexes = 4

[Z_1, X_1], [Z_2, X_2], [Z_3, X_3], [Z_4, X_4]

($X_1 \leqq X_2 \leqq X_3 \leqq X_4$)



Note that the coordinates of vertexes must be given in incremental values.

d) Tile Pattern: Eight patterns, 0 through 7

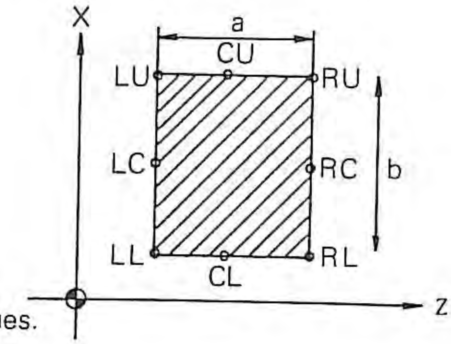
PAINTS <reference-point-position>, [Z_0, X_0], [a, b] <, tile-pattern>

The PAINTS command is used for painting a rectangle having lengths of a and b.

a) 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



b) Reference Position [Z_0, X_0]:

The coordinates must be specified in absolute values.

c) Tile Pattern: Eight patterns, 0 through 7

PAINTP <reference-point -position>, [Z_0, X_0], [a, b] <,angle-code> <,tile-pattern>

The PAINTP command is used for painting rectangles and triangles.

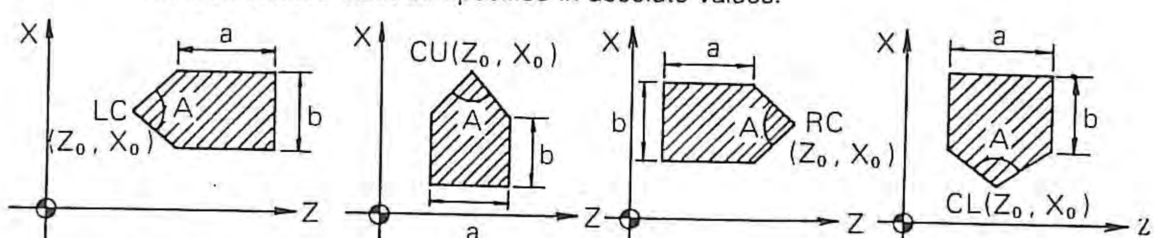
a) Reference Point Position:

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 [Z_0, X_0]:

The coordinates must be specified in absolute values.



b) Angle Code:

0: Angle A of triangle is 30°
1: Angle A of triangle is 60°
2: Angle A of triangle is 120°

Default is "0=30°".

c) Tile Pattern: Eight patterns, 0 through 7

PAINTT <shape-kind>, <reference-point-position>, [Z_0, X_0], [a, b], [H, R], <,tile-pattern>

The PAINTT command is used for painting a shape similar to an isosceles trapezoid whose legs are straight or segmental.

a) Shape Kind:

SS : Legs are convex segments equal to or smaller than a semi-circle.
LS : Legs are convex segments equal to or smaller than a semi-circle.
SD : Legs are concaved segments equal to or smaller than a semi-circle.
LF : Legs are straight lines

b) Reference Point Position:

This specifies the position of the reference point on the shape similar to an isosceles trapezoid whose legs are straight or segmental being painted.

LC : Left center
CU : Center upper
RC : Right center
CL : Center lower

Reference Position [Z_0, X_0]:

The coordinates must be specified in absolute values.

c) a, b : Length of upper and lower bases of isosceles trapezoid

d) H : Height of isosceles trapezoid

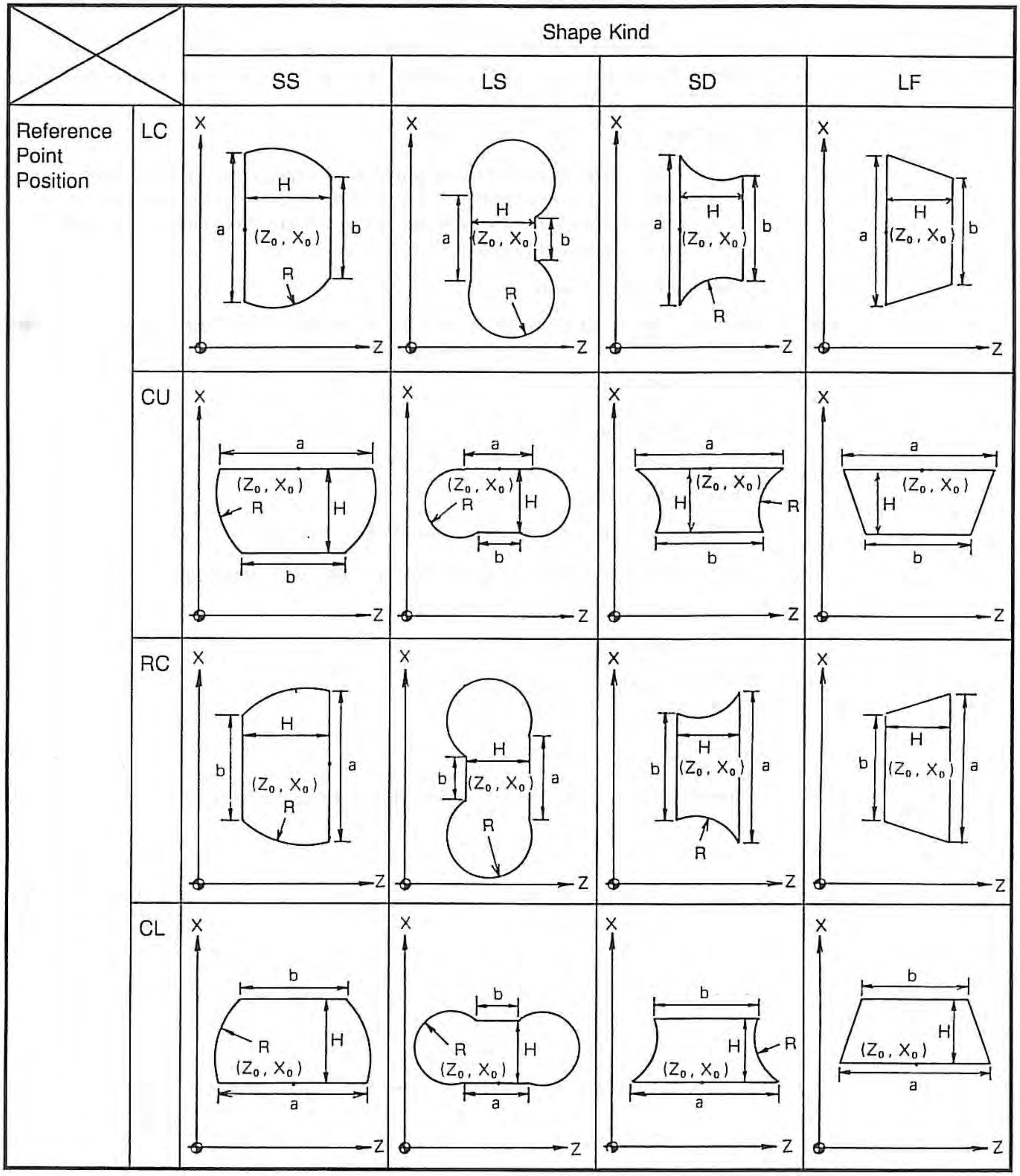
e) R : Radius of segment

"0" when the shape kind is LF.

f) Tile Pattern:

Eight patterns, 0 through 7

For reference point position, [a, b], and [H, R], refer to the list on the following page.


Shape Kind
SS
LS
SD
LF
Reference Point Position
LC
CU
RC
CL
X
Z
a
b
H
R
(Z₀, X₀)

(6) Example of Painting

This item explains the procedure to paint the blank shape indicated below.

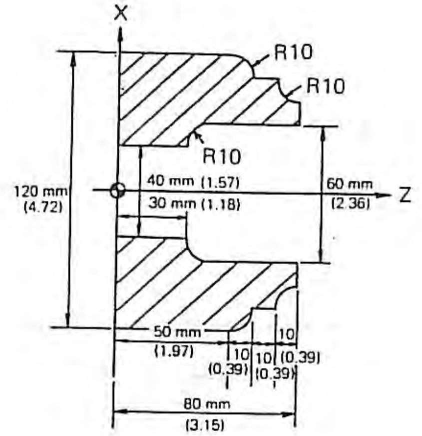


Fig. 17-1

Divide the portion to be painted into four sections, (1), (2), (3), and (4).

Divide the blank portion into three sections, A, B, and C. (Fig. 17-1)

Define each of the sections to be painted.

(1) PAINTT LF, LC, [0, 0], [120, 120], [50, 0]
(2) PAINTT SS, LC, [50, 0], [120, 100], [10, 10]
(3) PAINTT LF, LC, [60, 0], [100, 100], [10, 0]
(4) PAINTT SD, LC, [70, 0], [100, 80], [10, 10]

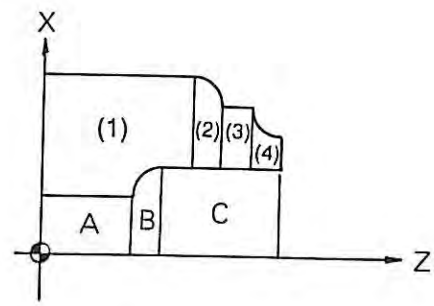


Fig. 17-2

Define each of the blank sections (Tile pattern: 0).

A. PAINTT LF, RC, [30, 0], [40, 40], [30, 0], 0
B. PAINTT SS, RC, [40, 0], [60, 40], [10, 10], 0
C. PAINTT LF, RC, [80, 0], [60, 60], [40, 0], 0

Therefore, the painting is programmed as indicated below:

```
DEF WORK
PAINTT LF, LC, [0, 0], [120, 120], [50, 0]
PAINTT SS, LC, [50, 0], [120, 100], [10, 10]
PAINTT LF, LC, [60, 0], [100, 100], [10, 0]
PAINTT SD, LC, [70, 0], [100, 80], [10, 10]
PAINTT LF, RC, [30, 0], [40, 40], [30, 0], 0
PAINTT SS, LC, [30, 0], [40, 60], [10, 10], 0
PAINTT LF, RC, [80, 0], [60, 60], [40, 0], 0
END
DRAW
```

SECTION 18 ONE-TOUCH IGF FOR PRECISION LATHES (OPTIONAL)

Differences between the one-touch IGF for conventional NC lathes and precision an NC lathes are indicated below.

1. Unit System

Unit systems of 0.1 μm and 0.00001 inches are added. See the setting of parameter (bit) data.

Setting of data at bit 0, 1 and 2 of parameter (bit) No. 3	Unit System
000	1 μm
001	0.0001 inch
010	1 mm
011	1 inch
100	10 μm
101	0.00001 inch
110	0.1 μm

Unit amount for each unit system setting is indicated below (excluding material data).

Dimension Parameter	0.1 μm	1 μm	10 μm	1 mm	0.00001 inch	0.0001 inch	1 inch
Length	0.0001 (mm)	0.001 (mm)	0.01 (mm)	1 (mm)	0.00001 (inch)	0.0001 (inch)	1 (inch)
Feed per Rev.	0.0001 (mm/rev)	0.001 (mm/rev)	0.01 (mm/rev)	1 (mm/rev)	0.00001 (inch/rev)	0.0001 (inch/rev)	1 (inch/rev)
Angle (°)	0.0001	0.001	0.01	1	0.00001	0.0001	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 one-touch IGF for the conventional NC lathes.

2. Upper and Lower Limits of Data Setting for Process 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)	−3937.0078 (inch)	+3937.0078 (inch)	−9999.9999 (mm)	+9999.9999 (mm)	−393.70078 (inch)	+393.70078 (inch)
Feed per Rev.	0.001 (mm/rev)	+99999.999 (mm/rev)	0.0001 (inch/rev)	+3937.0078 (inch/rev)	0.0001 (mm/rev)	+9999.9999 (mm/rev)	0.00001 (inch/rev)	+393.70078 (inch/rev)
Angle (°) *2	−360.000	+360.000	−360.0000	+360.0000	−360.0000	+360.0000	−360.00000	+360.00000
Speed (rpm)	1	9999	1	9999	1	0000	1	9999
Surface Speed (Cutting Speed)	1 (m/min)	3000 (m/min)	1 (feet/min)	9999 (feet/min)	1 (m/min)	3000 (m/min)	1 (feet/min)	9999 (feet/min)

*1 For DEPTH and ALLOWANCE, minimum value is 20.

*2 For EDGE ANGLE, minimum value is 20.

*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 CLEAR-ANCE	0	30.000	0	1.1811	0	30.0000	0	1.18110
ID CLEAR-ANCE	0	30.000	0	1.1811	0	30.0000	0	1.18110
FACE CLEAR-ANCE	0	30.000	0	1.1811	0	30.0000	0	1.18110
BLANK INSIDE LENGTH IL	0	+9999.999	0	+393.7007	0	+9999.9999	0	+393.70078
DIMEN-SION ZERO SHIFT	−9999.999	+9999.999	−393.7007	+393.7007	−9999.9999	+9999.9999	−393.70078	+393.70078
PRO-GRAM ZERO SHIFT	−9999.999	+9999.999	−393.7007	+393.7007	−9999.9999	+9999.9999	−393.70078	+393.70078
NOSE-R CANCEL TRAVEL	0	1.000	0	0.0394	0	1.0000	0	0.03937

3. Material Data

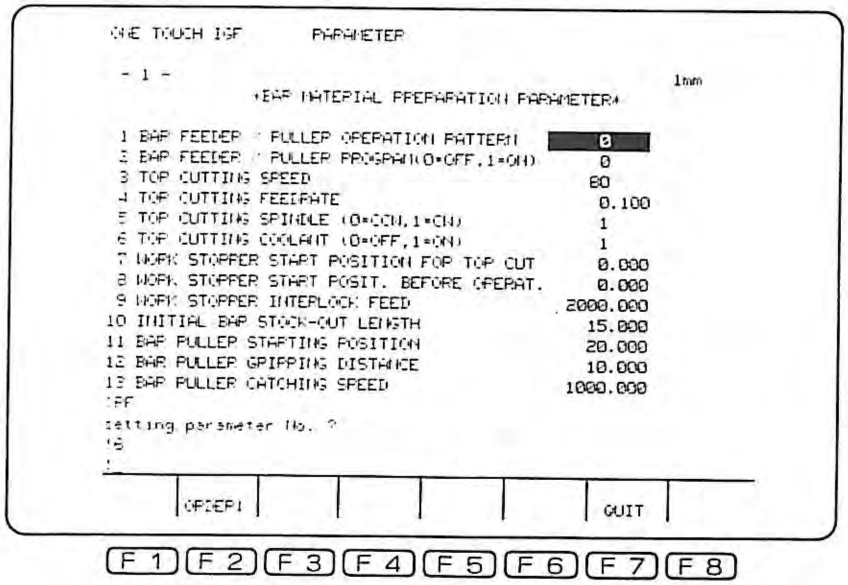
When designating data for the material data display screen, units and upper and lower value limits apply in the same way as for the non-precision lathe specifications described in Section 4, "Material Data Setting". When presetting this data through the graphic edit function, minimum command unit data (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 (0.1 μm, 0.1 μm/rev, or 0.00001 in. and 0.00001 inch/rev) at this time are normally "0", but this digit may be commanded at will when engaged in process editing.

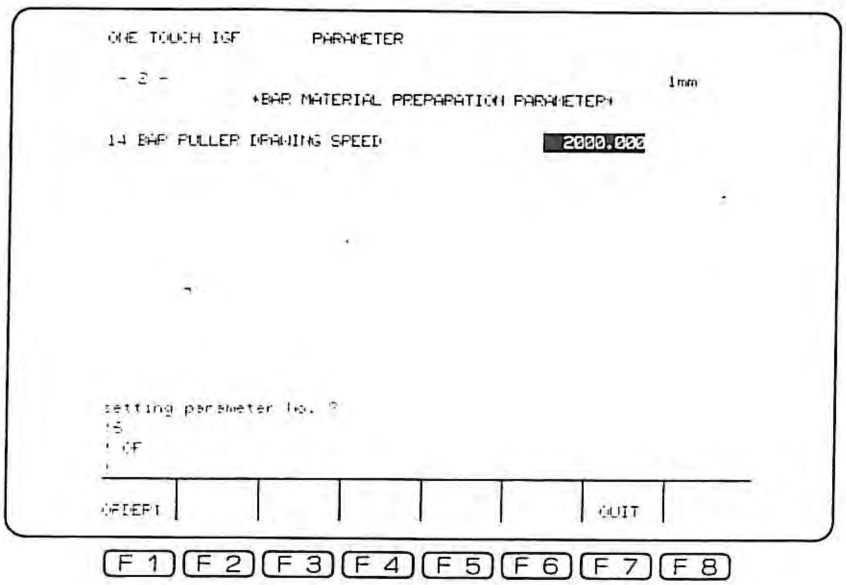
SECTION 19 BAR FEEDER AND BAR PULLER FUNCTION (OPTIONAL)

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.

1. Bar Material Preparation Parameters

The BAR MATERIAL PREPARATION PARAMETER screens as indicated below are provided. Set the data for each of these parameters following the explanation in Section 5.





Description of Parameters:

1 BAR FEEDER / PULLER OPERATION PATTERN (0 - 4)

0 ...... 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-V40, PF-V3, ANF65R, IBF-18NNC-2.0)

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 (0, 1)

0 ...... 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.

3 CUTTING SPEED (mm/rev)
4 CUTTING FEED (mm/min)
5 CUTTING SPINDLE (0 = CCW, 1 = CW)
6 CUTTING COOLANT (0 = 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.

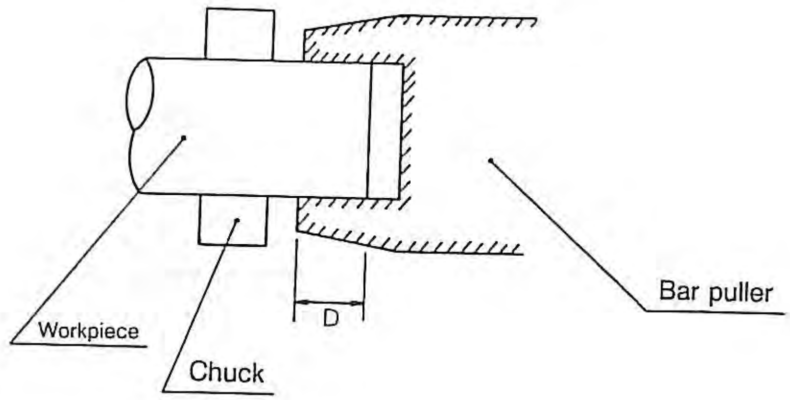
10 STOCK OUT LENGTH (FL)

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.

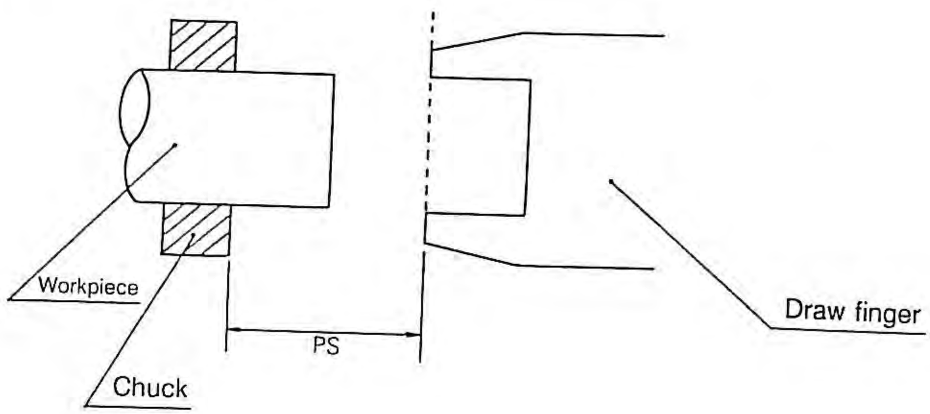
See the illustration below.



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)

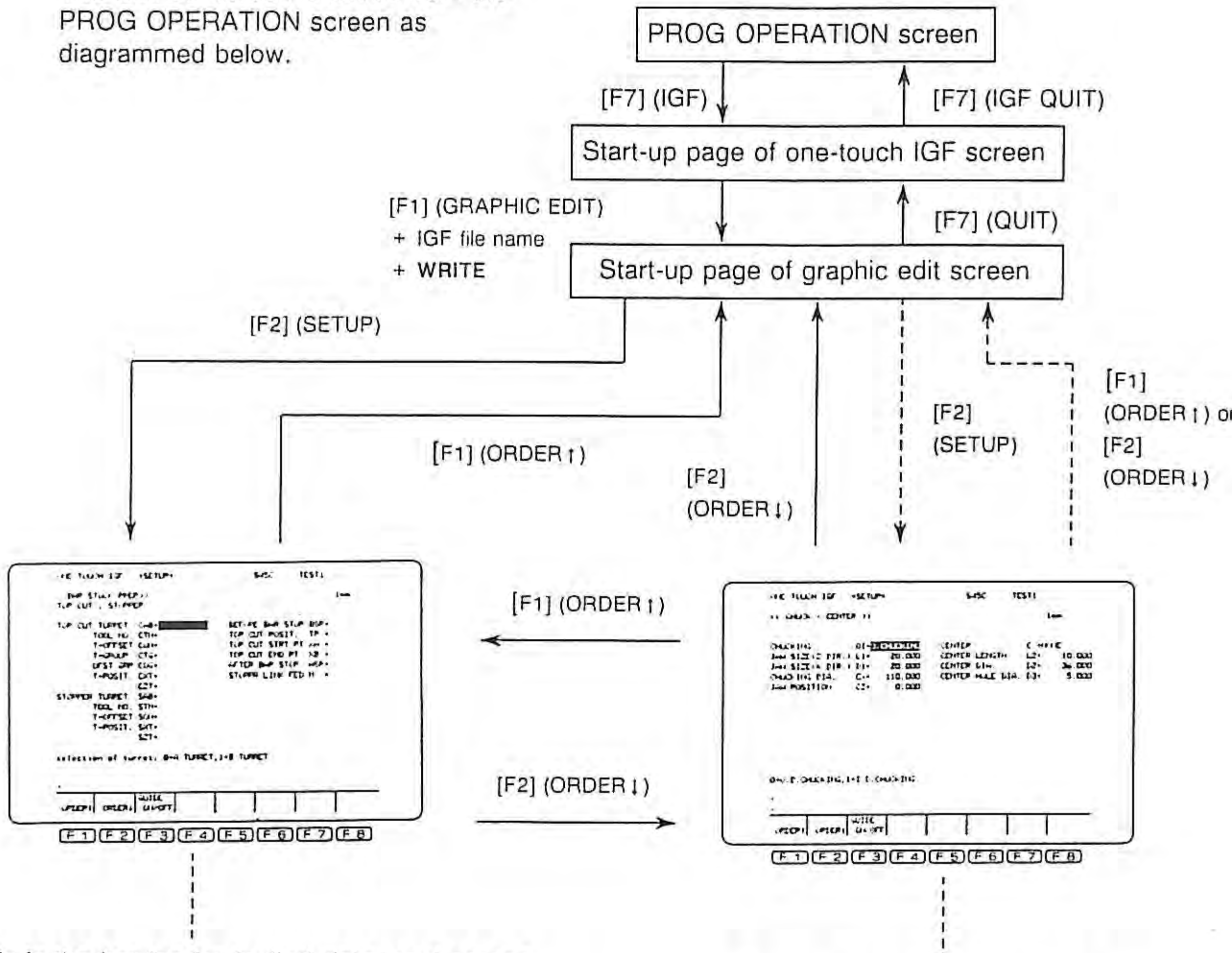
Input the feedrate of the draw finger in which it pulls out the bar.

2. Designating Bar Material Preparation Parameter

The program for controlling bar feeder (puller) can be made by setting the data for the bar material preparation parameter. To make this program, it is necessary to set the data of bar feeder (bar puller) on the SETUP screen.

Screen Transfer:

The SETUP screen is called from the PROG OPERATION screen as diagrammed below.



This is the bar feeder (puller) data input screen. (Three screens are available according to the data set at the bar material preparation parameter.)

This is the CHUCK·CENTER screen.

Note 1: *The screen transfer varies depending on the setting of parameter:*

——→ Bar material preparation parameter No. 1 BAR FEEDER/PULLER OPERATION PATTERN is set at "1", "2", "3", or "4", and No. 2 BAR FEEDER/PULLER PROGRAM (0 = OFF, 1 = ON) is set at "1"

----→ Either bar material preparation parameter No. 1 BAR FEEDER/PULLER OPERATION PATTERN is set at "0", or No. 2 BAR FEEDER/PULLER PROGRAM (0 = OFF, 1 = ON) is set at "0", or the bar feeder (puller) specification is not selected

Note 2: *When the function key [F1] (ORDER ↑) or [F2] (ORDER ↓) is pressed, the entered data is checked and the screen will be changed.*

The BAR STOCK PREP screen displayed varies as summarized below, depending on the setting for the bar material preparation parameter No. 1 and No. 2.

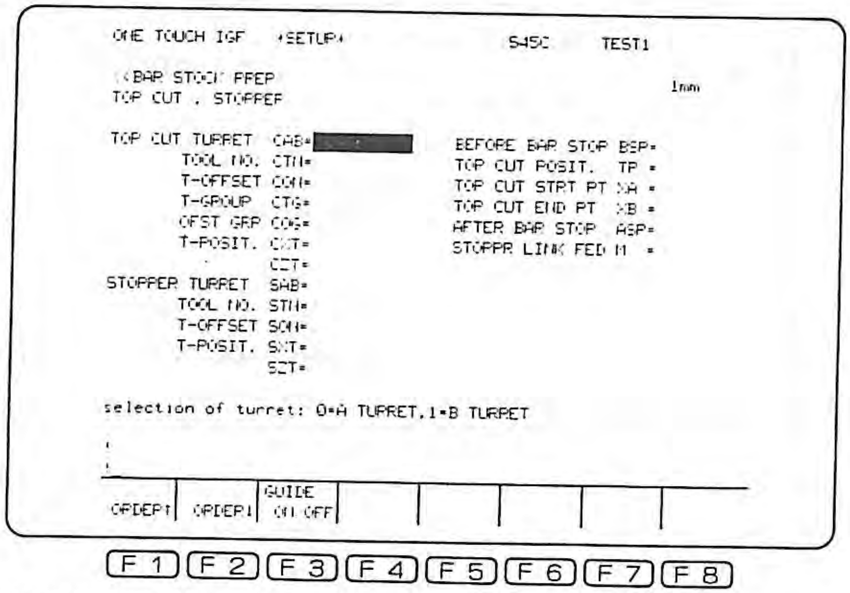
Parameter Setting		BAR STOCK PREP Screen Displayed
No. 1	No. 2	
1		TOP CUT, STOPPER screen
2		STOPPER screen
3	1	
4		BAR PULLER screen
0		The BAR STOCK PREP screen is not displayed.
0, 1, 2, 3, 4	0	The CHUCK·CENTER screen is displayed.

Note 1: *Process test is not executed for bar feeder (puller) programs.*

Note 2: *Cutoff is an independent process.* Set the data accordingly.

The BAR STOCK PREP screen displayed and the data to be set are as follows.

(1) TOP CUT, STOPPER Screen



TOP CUT TURRET CAB
(displayed for two-saddle specification models with the mirror image function activated)

The turret on which a tool for top cut is mounted is designated.

Enter "0" to select "0 = A TURRET" and "1" to select "1 = B TURRET".

* "Top cut" refers to the operation to cut the top of the bar material before starting cutting operation.

TOOL NO. CTN

T-OFFSET CON

T-GROUP CTG (displayed for tool life management specification models)

The tool group number in the tool life management specification is designated.

OFST GRP COG (displayed for tool life management specification models)

The tool offset group number in the tool life management specification is designated.

T-POSIT. CXT
CZT

STOPPER TURRET SAB
(displayed for two-saddle specification models with the mirror image function activated)

The turret on which a stop is mounted is designated.

Enter "0" to select "0 = A TURRET" and "1" to select "1 = B TURRET".

TOOL NO. STN

T-OFFSET SON

T-POSIT. SXT
SZT

BEFORE BAR STOP BSP

TOP CUT POSIT. TP

TOP CUT STRT PT XA

TOP CUT END PT XB

AFTER BARSTOP ASP

STOPPER LINK FED M

The following prompt will be displayed.

Bar stopper feed interlocked with bar movement? 0 = NONE 1 = EXIST

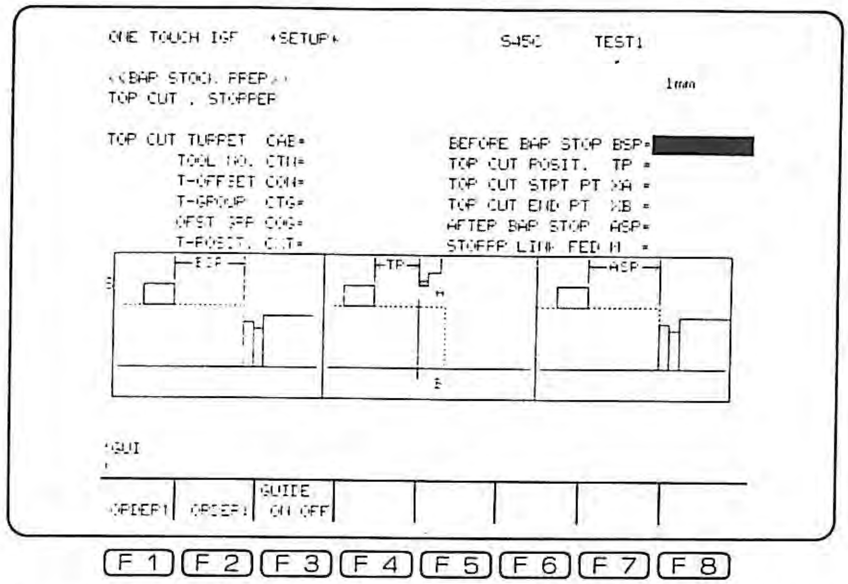
Enter "0" when bar stopper feed interlocked with bar movement is not required, and "1" when it is required.

Supplement

Guide Drawing Display for Data Input:

It is possible to display the guide drawing for data input.

Each time the function key [F3] (GUIDE ON/OFF) is pressed, the guide drawing display is turned on and off.

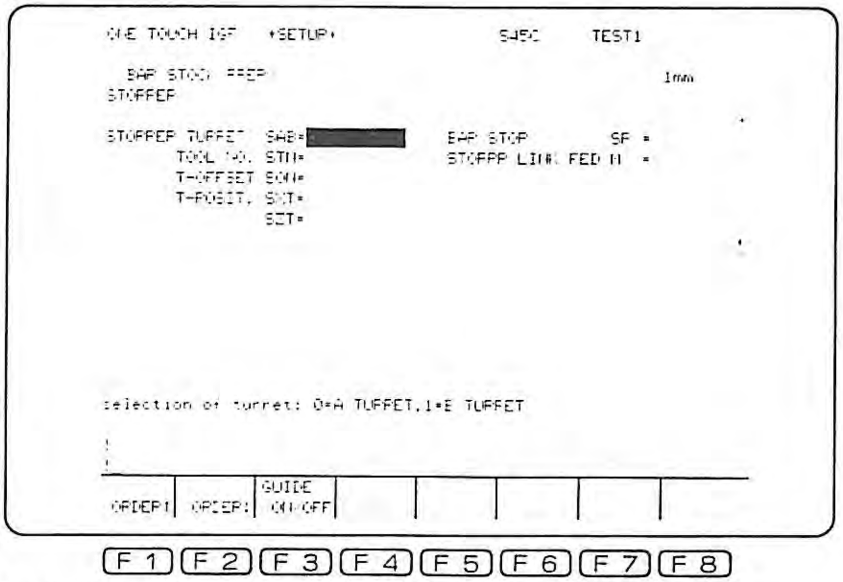


After the completion of data setting, press the function key [F2] (ORDER ↓).

The one-touch IGF system will check the entered data and the CHUCK.CENTER page of the SETUP screen will appear when the data has been correctly entered. An error will occur if there is faulty data.

Note: *Pressing the function key [F1] (ORDER ↑) will return the CRT to the start-up page of the one-touch IGF.*

(2) STOPPER Screen



STOPPER TURRET SAB TOP CUT TURRET CAB
(displayed for two-saddle specification models with the mirror image function activated)

The turret on which a stop is mounted is designated.

Enter "0" to select "0 = A TURRET" and "1" to select "1 = B TURRET".

TOOL NO. STN

T-OFSET SON

T-POSIT. SXT
SZT

BAR STOP SP

STOPPER LINK FED M

The following prompt will be displayed.

Bar stopper feed interlocked with bar movement? 0 = NONE 1 = EXIST

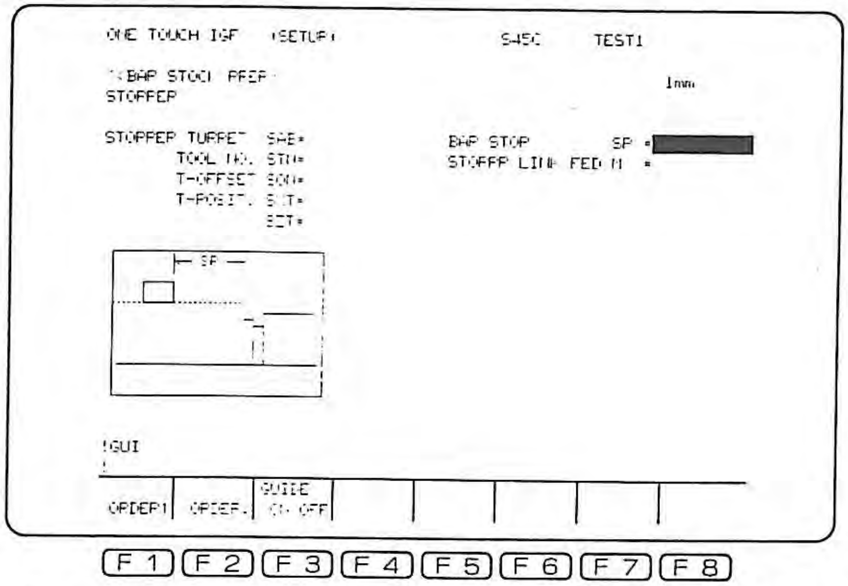
Enter "0" when bar stopper feed interlocked with bar movement is not required, and "1" when it is required.

Supplement

Guide Drawing Display for Data Input:

It is possible to display the guide drawing for data input.

Each time the function key [F3] (GUIDE ON/OFF) is pressed, the guide drawing display is turned on and off.

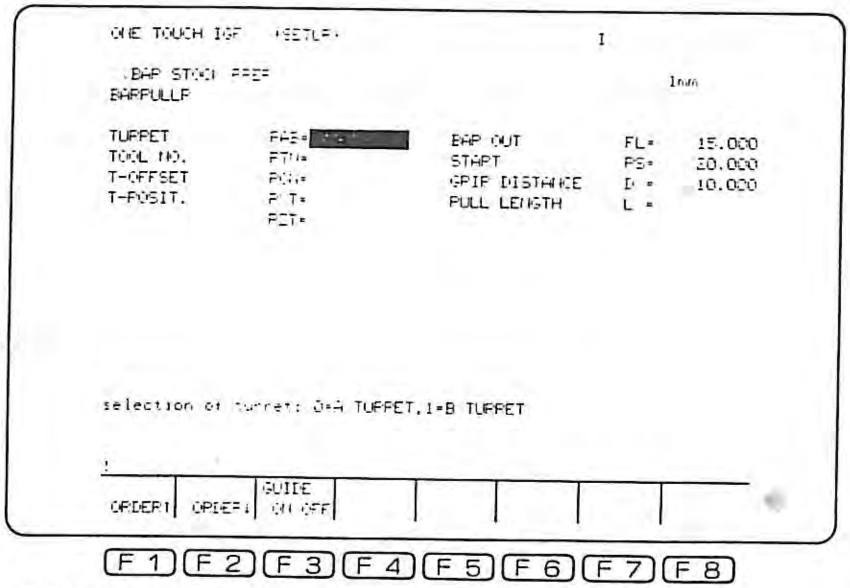


After the completion of data setting, press the function key [F2] (ORDER ↓).

The one-touch IGF system will check the entered data and the CHUCK·CENTER page of the SETUP screen will appear when the data has been correctly entered. An error will occur if there is faulty data.

Note: *Pressing the function key [F1] (ORDER ↑) will return the CRT to the start-up page of the one-touch IGF.*

(3) BAR PULLER Screen



TURRET PAB
(displayed for two-saddle specification models with the mirror image function activated)

The turret on which a draw finger for bar puller is mounted is designated.

Enter "0" to select "0 = A TURRET" and "1" to select "1 = B TURRET".

TOOL NO. PTN

T-OFFSET PON

T-POSIT. PXT
PZT

BAR OUT FL

The same value as set at the bar material preparation parameter No. 10 INITIAL BAR STOCK-OUT LENGTH is initially set.

START PS

The same value as set at the bar material preparation parameter No. 11 BAR PULLER STARTING POSITION is initially set.

GRIP DISTANCE D

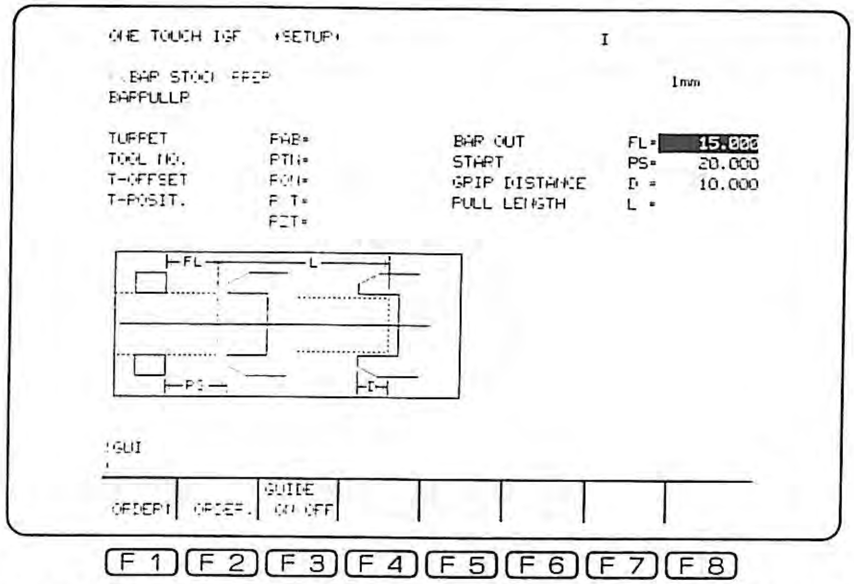
The same value as set at the bar material preparation parameter No. 12 BAR PULLER GRIPPING DISTANCE is initially set.

Supplement

Guide Drawing Display for Data Input:

It is possible to display the guide drawing for data input.

Each time the function key [F3] (GUIDE ON/OFF) is pressed, the guide drawing display is turned on and off.



After the completion of data setting, press the function key [F2] (ORDER ↓).

The one-touch IGF system will check the entered data and the CHUCK·CENTER page of the SETUP screen will appear when the data has been correctly entered. An error will occur if there is faulty data.

Note: *Pressing the function key [F1] (ORDER ↑) will return the CRT to the start-up page of the one-touch IGF.*

SECTION 20 ONE-TOUCH IGF ON IN-PROCESS GAUGING FUNCTION SPECIFICATION (OPTIONAL)

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 process edit operation.

1. Parameters Used for Gauging Cycles

The following parameter setting screen is provided to set one-touch 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.

```
ONE TOUCH IGF        PARAMETER

 - 4 -                                                   1mm
                     *DIMENSION PARAMETER*

40 TOOL INDEX POSITION X-AXIS                     500.000
41 TOOL INDEX POSITION Z-AXIS                     800.000
42 REAR MACHINING Z-AXIS CLEARANCE                  4.000
43 THRU-HOLE CUTTING DEPTH                          3.000
44 ROUGHING NOSE-R MAXIMUM VALUE                     1.200
45 FINISHING NOSE-R MAXIMUM VALUE                    0.800
46 MAX.AIRCUT DISTANCE FOR RECESSED SHAPE          10.000
47                                                  0.000
48 GAUGE PT TO APPROACH PT1 DISTANCE (APP)          2.000
49 1ST GAUGE PT TO APPROACH PT2 DIST(APPS)          1.000
50 GAUGE PT TO IMAGINARY PT DISTANCE (IMP)          2.000
51 NG LIMIT (DNG)                                   0.040
52 OK LIMIT (DOK)                                   0.020
! OF
! OB
! OB
!
ORDER↑ | ORDER↓ |  |  |  |  | QUIT |
```

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

Note 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 OSP5020L Gauging Manuals.*

Note 2: *Parameters DNG and DOK can be set during the process editing operation regardless of the data set on the parameter setting screen.*

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.

The procedure for making a gauging process in the process edit mode is explained taking the work-piece used in Section 4 of the Operation Manual - BASIC as an example. A gauging process for each of four processes, rough turning, finish turning, copying, and grooving is detailed below.

2-1. Gauging Cycles for Rough, Turning, Finish Turning, and Copy Processes

This is the procedure to make a gauging process for the 5th process "FIN. OD←".

Operating Procedure:

1) Display the PROCESS EDITING screen.

2) Move the cursor to the blank process.

   A gauging process can be made anywhere after the process for which the gauging process is made.

   In the case of two-saddle specification models, make a gauging process for the turret on which a sensor is mounted.

   Move the cursor to the blank process following the FIN. OD← process, using the cursor control keys, ↑ and ↓.

   If no blank process is left, press the function key [F8] (EXTEND) until the function "INSER" is assigned to the function key [F3]. Then, press the function key [F3] (INSERT). A blank process will be created.

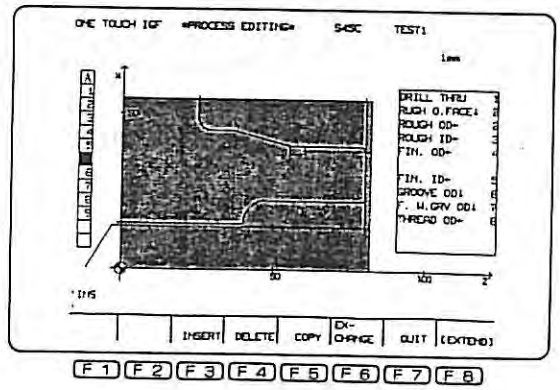
3) Display the MACH. METHOD AMEND screen and press the function key [F1] (METHOD AMEND).

Start-up page of the one-touch IGF screen

↓ [F1] (GRAPHIC EDIT) + TEST1 + **WRITE**

Start-up page of graphic edit screen

↓ [F4] (PROCESS EDIT)



↓ [F1] (METHOD AMEND)

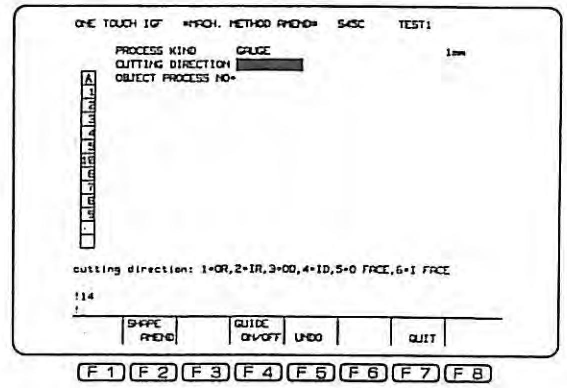
MACH. METHOD AMEND screen

4) Designate the machining method.

   Select "14 = GUG".

5) Enter "1 = OR" for CUTTING DIRECTION.

   The guide drawings indicating gauging motion patterns can be displayed on this screen.



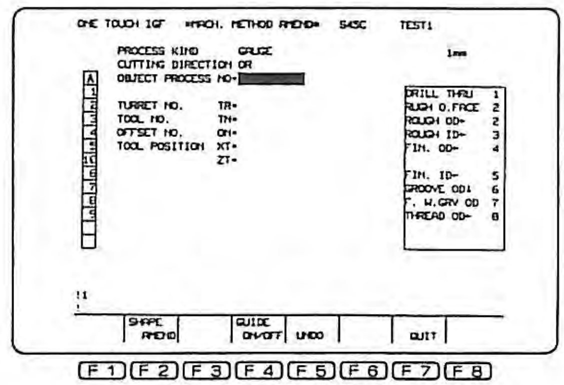
6) Select the process for which a gauging process is executed.

   Key in "5" to select "FIN. OD←" from the process table at the right of the screen.

7) Enter "12" for TOOL NO. TN to select the turret position in which a sensor is mounted.

   Enter "12" for OFFSET NO. ON. This is the offset number of sensor.

   Enter "500" for TOOL POSITION XT and "800" for TOOL POSITION ZT.



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 one-touch IGF

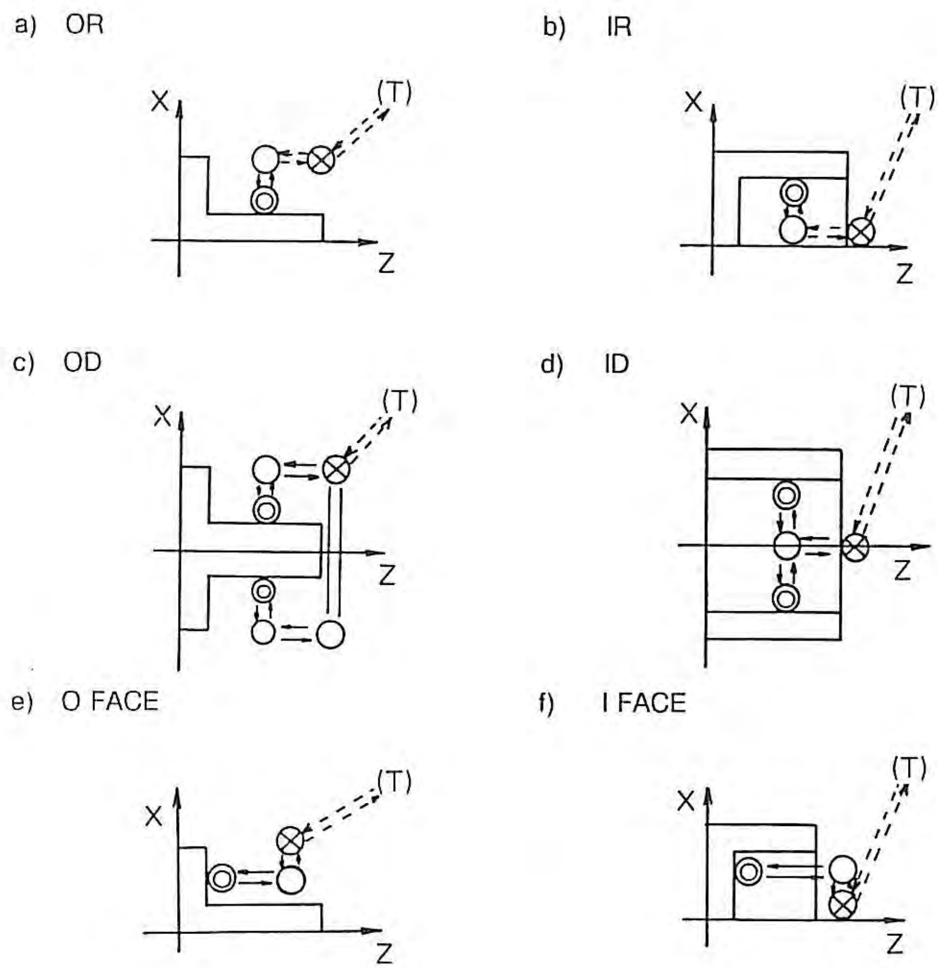


Fig. 20-1

In case the gauging direction set in step (5) and the process selected do not match each other*, an error occurs.

* The surface on which the gauging cycle can be made is not preset 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←	○	×	○	×	○	×
2. COPY	2. ID←	×	○	×	○	×	○
3. FINISH	3. O.FACE ↓	○	×	○	×	○	×
	4. I.FACE ↑	×	○	×	○	×	○
	5. M.OD←	○	×	○	×	○	×
	6. M.ID←	×	○	×	○	×	○
	7. M.FACE ↓	○	×	○	×	○	○
	8. M.OD→	○	×	○	×	○	×
	9. M.ID→	×	○	×	○	×	○
	10. M.FACE ↑	×	○	×	○	○	○
	11. OD→	○	×	○	×	○	×
	12. ID→	×	○	×	○	×	○
6. GROOVE	1. OD ↓	○	×	○	×	○	×
	2. ID ↑	×	○	×	○	×	○
	3. FACE←	○	×	○	×	○	○
7. R. GRV	1. WIDE OD ↓	○	×	○	×	○	×
8. F. GRV	2. WIDE ID ↑	×	○	×	○	×	○
	3. WIDE FACE←	○	○	○	×	○	○
	5. V OD ↓	○	×	○	×	×	×
	6. V ID ↑	×	○	×	○	×	×
	7. V FACE←	×	×	×	×	○	○

○: Gauging possible
×: Gauging impossible

8) Display the MACH. SHAPE AMEND screen.

Press the function key [F2] (SHAPE AMEND).

9) Enter the machining shape (gauging motion).

Move the position of RF POINT (magenta) to the position where the sensor contacts the workpiece.

In this example, move the position of RF POINT to the right side of the groove.

Enter "90.000" for REF. PT X and "85.000" for REF. PT Z.

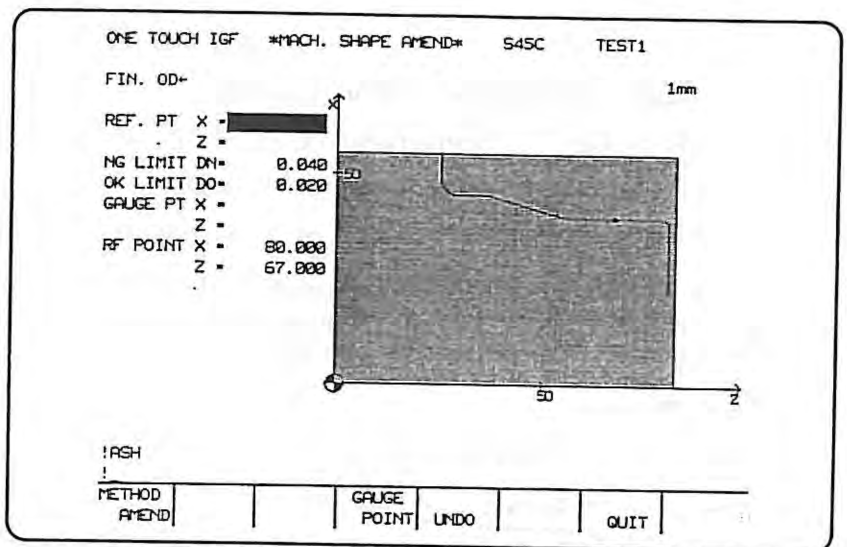
For the location of the reference point, refer to Fig. 20-1 and Fig. 20-2.

The values for NG LIMIT and OK LIMIT are not changed.

* NG LIMIT — An alarm occurs if the difference between the target (commanded) value and the actual measured value is outside this limit.

* OK LIMIT — Offset is carried out if the difference between the target (commanded) value and the actual measured value is outside this limit.

The same values as set at the one-touch IGF dimension parameter are initially set for NG LIMIT and OK LIMIT.



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

10) Designate GAUGE PT X and GAUGE PT Z.

Move the cursor to GAUGE PT X and press the function key [F4] (GAUGE POINT). The same value as set for RF POINT X will be set for GAUGE PT X.

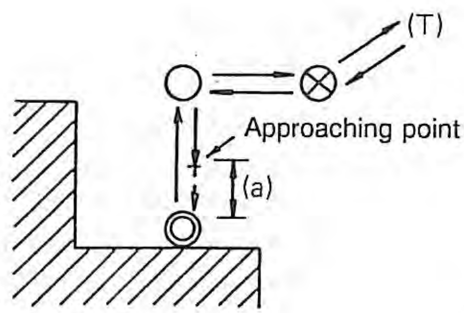
Press the function key [F4] (GAUGE POINT) again. The same value as set for RF POINT Z will be set for GAUGE PT Z.

It is possible to directly enter the values through the keyboard without using the function key [F4] (GAUGE POINT).

Remarks:

1. RF POINT X and RF POINT Z are not displayed unless the shape in the process to be gauged contains the face vertical to the gauging direction.

2. 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 cycle start point

◎ : Gauging point

ⓐ : Relative value of the gauging target point value and the approach point (APP: parameter set value)

←—— : Rapid positioning

←---- : Positioning in cutting feedrate

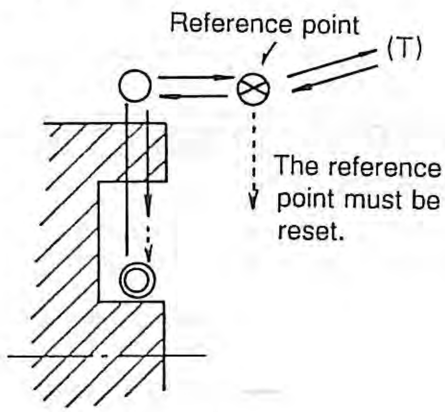
Fig. 20-2

The gauging starting point is defined as follows.

- OR, IR, OD Starting point (X, Z) = (REF. PT X, GAUGE PT Z)
- ID Starting point (X, Z) = (Program zero, GAUGE PT Z)
- O FACE, I FACE Starting point (X, Z) = (GAUGE PT X, REF. PT Z)

3. There are cases in which the workpiece interferes with the sensor if gauging is executed with the reference point set at the preset reference point position, depending on the workpiece shape. Change the setting for REF. PT in such cases.

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.

Fig. 20-3

11) Terminate the gauging process data input.

Press the function key [F7] (QUIT). Check the GAUGE OR↓ process has been made. To correct the gauging process data, move the cursor to the gauging process (GAUGE OR↓ in this example) and advance to the MACH. METHOD AMEND or MACH. SHAPE AMEND screen.

2-2. Gauging Cycle for Grooving Process

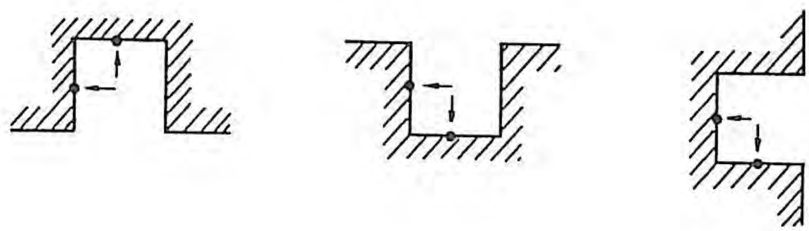
For single-groove and wide-groove cut processes, gauging is executed for the following two positions:

Groove depth
Groove width

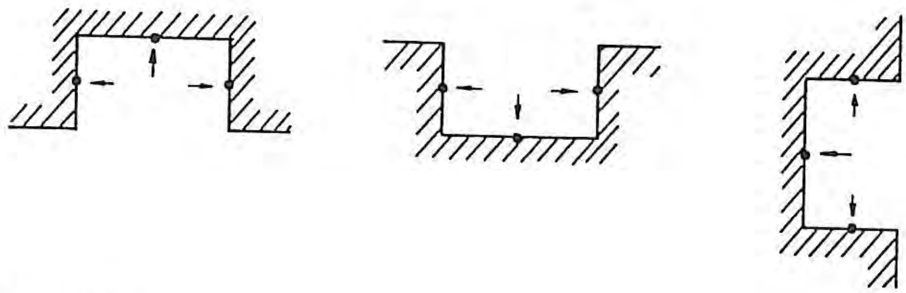
For the V-groove cut process, gauging is executed only for groove depth.

The relationship between the direction of groove cut, the gauging direction, and the position of RF POINT is illustrated on the following page.

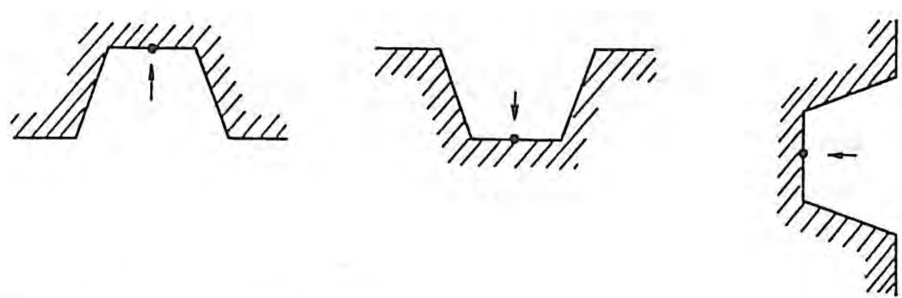
(1) Groove (Single groove)



(2) Wide Groove



(3) V Groove



←, ↑, ↓ : Gauging direction
● : Reference point

This is the procedure to make a gauging process for the left side face of the wide groove created in the 9th process.

Operating Procedure:

1) Display the PROCESS EDITING screen.

2) Move the cursor to the blank process.

3) Display the MACH. METHOD AMEND screen and press the function key [F1] (METHOD AMEND).

4) Designate the machining method.

   Select "14 = GUG".

   Enter "5 = O FACE" for CUTTING DIRECTION.

   The gauging motion pattern guide drawing can be displayed on this screen.

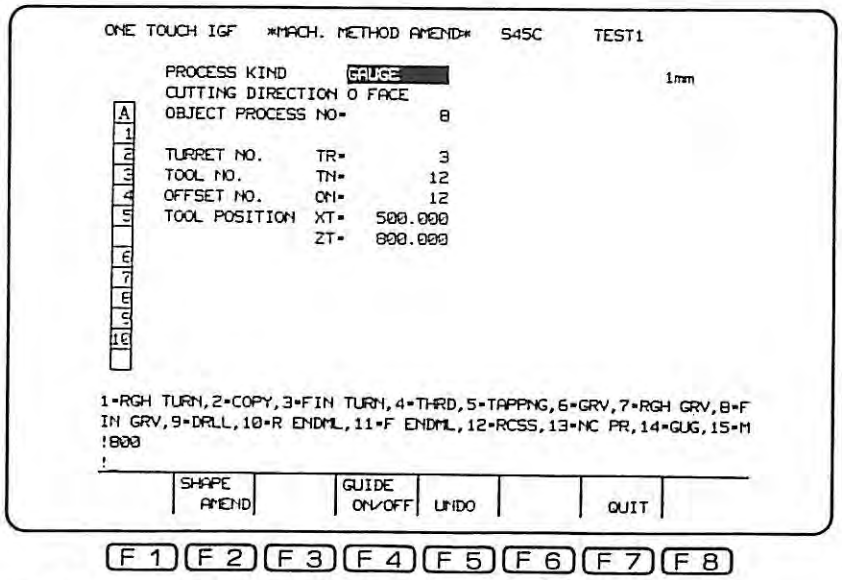
   Select the process for which a gauging process is executed.

   Key in "8" to select "F. W. GRV OD ↓ " from the process table at the right of the screen.

   Enter "12" for TOOL NO. TN to select the turret position in which a sensor is mounted.

   Enter "13" for OFFSET NO. ON. This is the offset number of sensor.

   Enter "500" for TOOL POSITION XT and "800" for TOOL POSITION ZT.



5) Display the MACH. SHAPE AMEND screen.

   Press the function key [F2] (SHAPE AMEND).

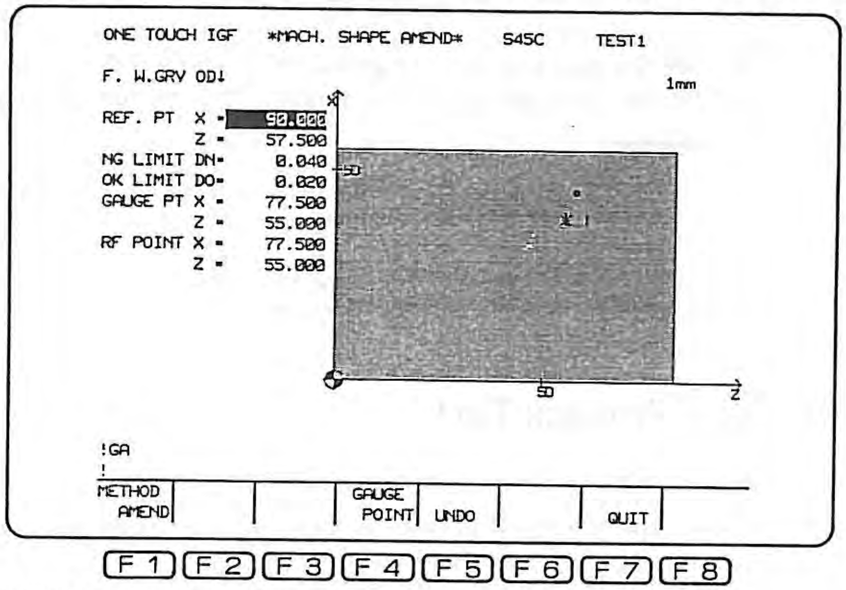
6) Enter the machining shape (gauging motion).

Move the position of RF POINT (magenta) to the left side of the groove.

Enter "90.000" for REF. PT X and "57.500" for REF. PT Z.

The values for NG LIMIT and OK LIMIT are not changed.

Move the cursor to GAUGE PT X and press the function key [F4] (GAUGE POINT) twice.



7) Terminate the gauging process data input.

Press the function key [F7] (QUIT). Check the GAUGE OFACE← process has been made.

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 is changed or deleted.

(1) After a Machining Process Change

If any of the TURRET, KIND, or DIRECTION of the machining process is 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.

4. Gauging Process Test

In the one-touch IGF process test, the gauging process is processed in the following manner.

(1) On the ONE TOUCH 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 within this distance.

5. NC Gauging Process Program

The gauging process is converted into an NC program whose configuration is shown below.

Animation Data

N0001 | 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 0] NMS01
Sensor indexing
Positioning at gauging cycle start point
Call of gauging cycle subprogram (OWXMA, ODOAA, etc.)
Retraction of turret to the turret index position
— (1)

NMS01 | Next Process

Call of gauging cycle ON condition cancel subprogram (OMSSF)
Call of gauging cycle ON counter check subprogram (OCNCK)
— (2)

M02

Program areas in ▭ 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.

Program Example:

```
N0001 G13
N0002 G00 X500 Z800
N0003 G50            S4200
N0004 G97            S620 M41  M03  M08  P0060
N0005 G00 X118 Z82        T020202
N0006     X49.4
N0007 G87 N0008
N0008 G81
N0009 G00 X44.6
N0010 G01      Z80        G42  F0.18
N0011     X78
N0012     X80  Z79
N0013          Z55
N0014     X90  Z36.34
N0015          Z29
N0016 G02 X98  Z25        I4
N0017 G01 X110.8
N0018 G40
N0019 G80
N0020 G01 X113 Z26.1
N0021 G00 X118
N0022     X500 Z800       T0200
N0023                     M05  M09       P0070
N0100 G13
N0101                                    P0080
N0102 IF[VDIN[32]_EQ_0]   NMS01      <------ Judgment of gauging cycle ON
                                             conditions
N0103 G00 X118 Z85        T121212    <------ Sensor indexing
N0104     X90                        <------ Positioning at reference point
N0105          Z67                   <------ Positioning at gauging cycle start
                                             point
N0106 CALL OWMXA MSP = 80 APP = 2 APPS = 1 IMP = 2 DNG = 0.04
      DOK = 0.02 TOFN = 0.2          <------ Call of gauging cycle subprogram
N0107          Z85
N0108     X118
N0109     X500 Z800       T1200      <------ Retract to turret indexing position
NMS01
   :                      :
   :                      :
N1000 G14
N1001 G13
N1002                                P0090
N1003 CALL OMSSF                     <------ Call of gauging cycle ON condition
                                             cancel subprogram (A-turret side)
N1004 CALL OCNCK                     <------ Call of gauging cycle ON counter
                                             check subprogram
N1005 G14
N1006                                P0090
N1007 CALL OMSSF                     <------ Call of gauging cycle ON condition
N1008 M02                                    cancel program (B-turret side)
```

SECTION 21 ONE-TOUCH IGF WITH LOAD MONITOR SPECIFICATION (OPTIONAL)

The one-touch IGF equipped with the load monitor specification allows an NC program for the load monitor function to be made simply.

- By designating the monitor axis for individual tools, the program to monitor the load of the designate axis in the process where the designated tool is used.
- For rapid feed operation, load monitoring is ignoring (M216 is output.).

1. Setting Monitor Axis

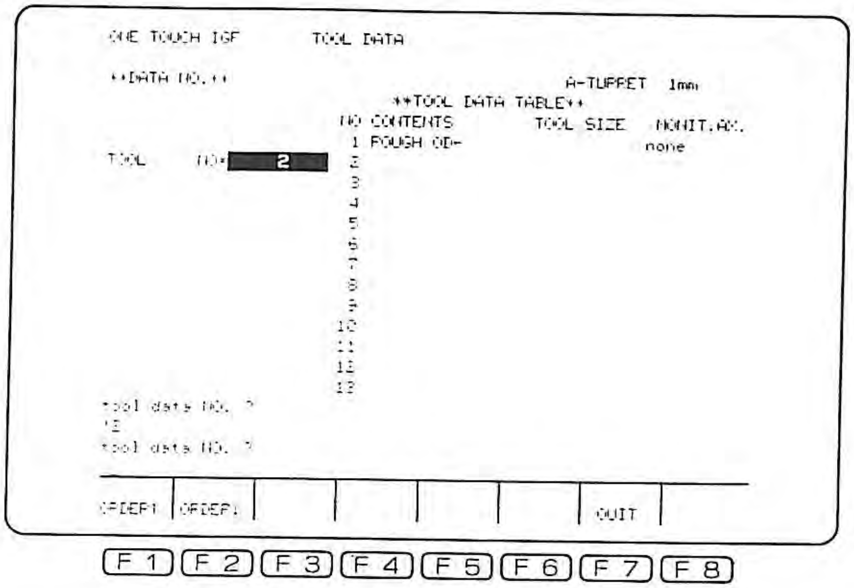
(1) Setting Monitor Axis in the Tool Data Setting Mode

1) Press the function key [F3] (TOOL DATA) after displaying the start-up screen of the one-touch IGF.

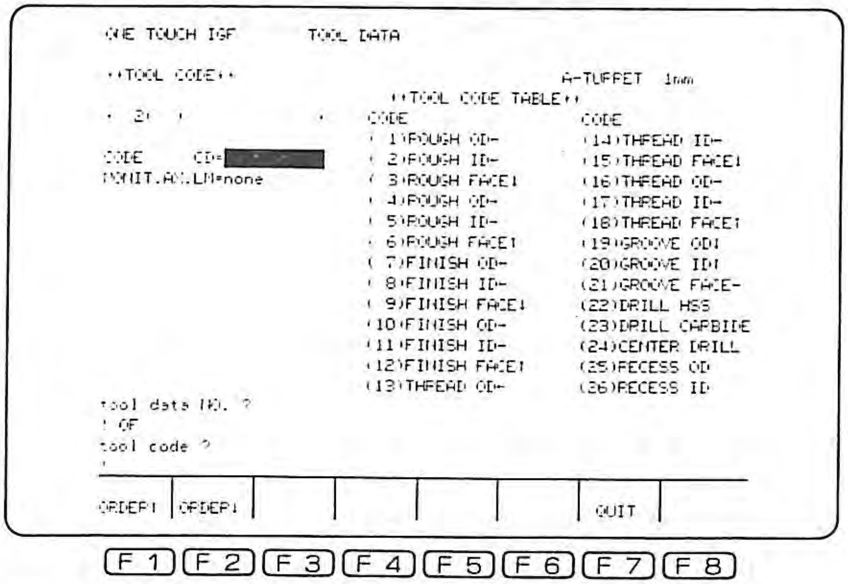
2) Select the turret with the turret selection key.

This selection is not required for the single saddle machine.

3) When the process is proceeded, the screen indicated below is displayed. Select the tool data number for which the data is set or modified.



4) Press the function key [F2] (ORDER ↓). The screen changes as indicated below.



5) Input the tool code.

The following prompt is displayed on the console line.

MONIT.AX. (0 = NONE, 1 = X, 2 = Z)

* The prompt will differ slightly according to the specification.

6) Input the monitor axis.

Examples:

For X-axis . . . . 1 WRITE
For X and Z axes . . . . 1 + 2 WRITE

(2) Setting Monitor Axis in Process Edit Operation

1) Display the MACH. METHOD AMEND page of the PROCESS EDITING screen. For the procedure to display this page, refer to Section 11, 4. "Editing the Machining Method".

```
ONE TOUCH IGF   *MACH. METHOD AMEND*   S45C      TEST1

     PROCESS KIND        DRILL                                  1mm
     CUTTING DIRECTION THRU(HSS)  CUTTING SPEED  V =         20
 A                                FEEDRATE       F =      0.150
 1                                SPINDLE(CW/CCW)MS=CW
 2   TOOL DATA NO.    NO=        0 COOLANT(ON/OFF)MC=ON
 3   MONIT.AX.        LM=NONE      DEPTH(THRU)     DE=     3.000
 4   TURRET NO.       TR=        1 PECKING(ON/OFF)SP=ON
 5   TOOL NO.         TN=        1 DEPTH(STEP)     D1=     3.000
 6   OFFSET NO.       ON=        1       (RETRACT) D2=    56.000
 7   TOOL GROUP NO.   TG=          RPM(CENTER)     S =     1200
 8   OFFST GROUP NO.OG=            FEEDRATE(CENT)  F =     0.080
 9   FORM CODE NO.    FC=        1 DEPTH(CENTER)   D =     3.000
     TOOL ANGLE       A1=   118.000
     DRILL LENGTH     L =   104.000
     DRILL DIA.       D =    28.000
     TOOL POSITION    XT=   500.000
MONIT.AX. (0=NONE,1=X,2=Z,8=SPINDLE)

!
!
        | SHAPE |       | GUIDE  |      |      |      | QUIT |
        | AMEND |       | ON/OFF | UNDO |      |      |      |

  [F 1] [F 2] [F 3] [F 4] [F 5] [F 6] [F 7] [F 8]
```

2) Input the tool data.

The following prompt is displayed on the console line.

MONIT.AX. (0 = NONE, 1 = X, 2 = Z)

* The prompt will differ slightly according to the specification.

3) Input the monitor axis.

Examples:

For Z-axis . . . . **2 WRITE**
For X-axis . . . . **1 WRITE**

4) The name of the monitor axis will be displayed for MONIT. AX. LM.

2. Others

(1) It is not necessary to input the data for PART NO.

(2) For the load monitor function, refer to OSP5020L Special Functions Manual (Ⅲ).

SECTION 22 ONE-TOUCH IGF WITH ATC SPECIFICATION (OPTIONAL)

The tool number and the tool group number differ depending on whether or not the machine is equipped with the ATC.

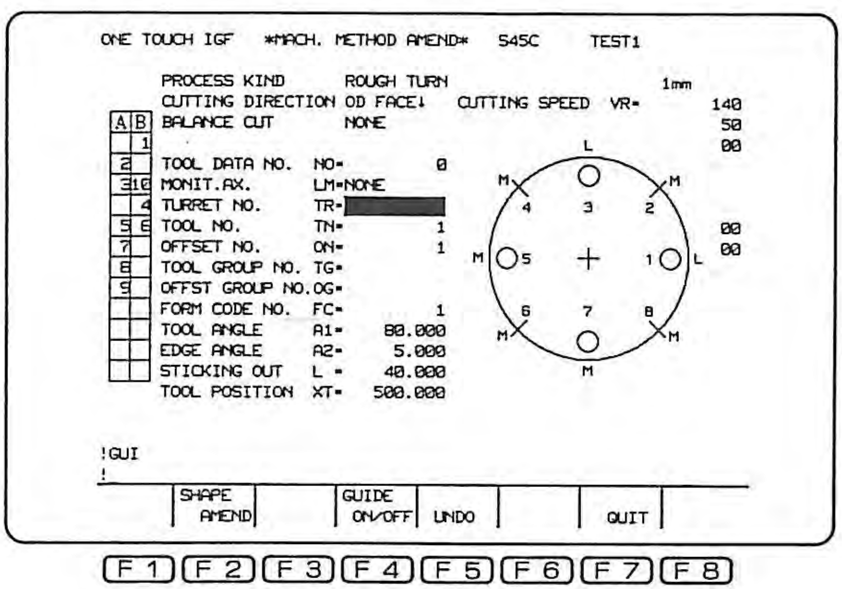
	Without ATC	With ATC
Tool Number	1 to 12	A side: 1 to 96, B side: 1 to 12
Tool Offset Number	1 to (Number of tool offset pairs in NC) Number of tool offset pairs: Standard: 32 Optional: 64 or 96	1 to (Number of tool offset pairs in NC) Number of tool offset pairs: Standard: 32 Optional: 64 or 96
Tool Group Number	1 to 12	1 to 24
Offset Group Number	1 to 3	1 to 3

When an attempt is made to newly register the process data in the process edit mode with the model LR15-M ATC, it is necessary to set the data for TURRET NO. on the MACH. METHOD AMEND screen. For the procedure to display the MACH. METHOD AMEND screen, refer to Section 11, 4. "Editing the Machining Method".

TURRET NO. refers to the number of the turret position where the selected tool is mounted or the selected tool is to be mounted using the ATC.

Set the L-tool to any of the turret positions marked with "L" (1, 3) on the screen, and the M-tool to any of the turret positions marked with "M" (2, 4, 5, 6, 7, 8).

When a L-tool is set to the "M" position or a M-tool is set to the "L" position, an alarm occurs.



SECTION 23 CONVERSION FUNCTION (OPTIONAL)

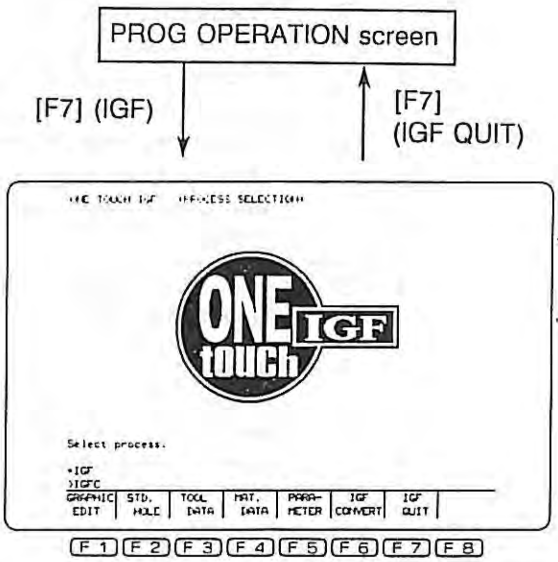
The conversion function converts the LAP function, nose R compensation function, and compound fixed cycle function programmed using the one-touch IGF into the program in the general programming format. The part program created using other than the one-touch IGF can also be converted using this function, however the contents of the part program after conversion can not be guaranteed.

1. Operating Procedure

Conversion is executed on the start-up page of the one-touch IGF screen.

Proceed as follows:

1) Press the function key [F6] (IGF CONVERT).
2) Key in the input file name, ",", output file name, ";", and option parameter through the keyboard, and press the **WRITE** key.



Note 1: *Input file name refers to the name of the part program created through the one-touch IGF.*

Note 2: *Output file name refers to the name of the program in the general programming format. (If the output file name is not entered, a program having the same name as the input file is created, resulting in deletion of the input file.)*

Note 3: *For the option parameter, refer to the table on the following page.*

Option Parameter:

Code	Contents	When Omitted
A	The program of the designated input file name is converted into the program for the OSP5020L/OSP5000L-G/OSP500L-G without LAP, nose R compensation, and compound fixed cycle functions, and stored in the bubble memory with an output file name assigned.	Conversion into the program for the OSP5020L/OSP5000L-G/OSP500L-G
B	The program of the designated input file name is converted into the program for the OSP3000L and stored in the bubble memory with an output file name assigned.	Conversion into the program for the OSP5020L/OSP5000L-G/OSP500L-G
D	The blank material shape designation start command (G83) and the blank material shape designation block are deleted from the program of the designated input file name, and the program is stored to the bubble memory with an output file name assigned. LAP4 is converted into LAP3.	
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 an output file name assigned.	Conversion into the program for A-turret

Example: To convert the file A.MIN into the program for the OSP3000L (file name B.MIN) with spaces inserted

>IGFC A.MIN, B.MIN;BS **WRITE**

or

>IGFC A, B;BS **WRITE** (when the extension name .MIN is omitted)

Note: *When LAP4 is converted into LAP3 by entering ";D", tool path will be changed. This may result in an occurrence of tool interference in ID turning operation as indicated in Fig. 23-1. In this case, change the cycle reference point. If machining on the descending step is present in the blank material copy turning cycle, do not attempt conversion.*

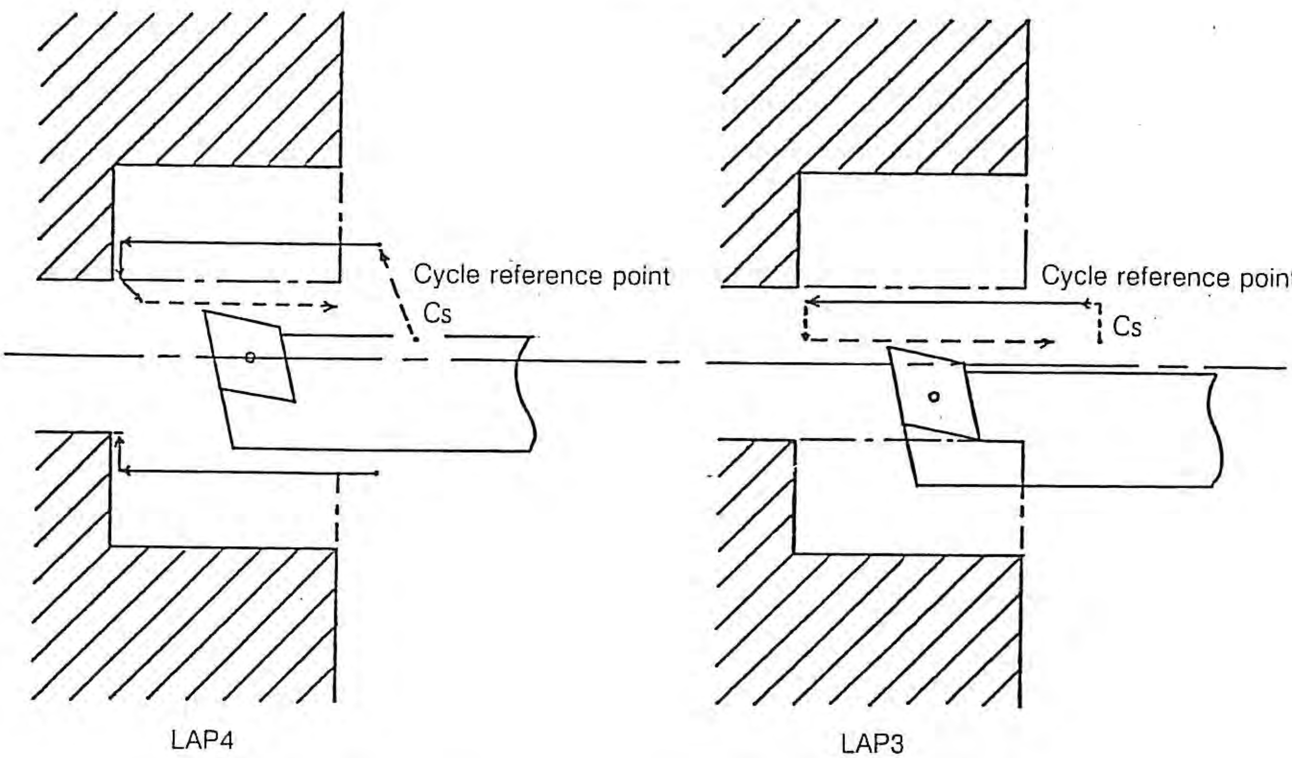


Fig. 23-1

2. Items to be Converted

(1) High-speed Round Bar Turning C cycle

- G85 : Bar turning rough cut cycle
- G87 : Finish cut cycle
- G81 : Start of contour definition, longitudinal
- G82 : Start of contour definition, transverse
- G83 : Start of blank material shape definition
- G80 : End of contour definition

Tool path in the machining cycle called by the G85 command (Fig. 23-2) is converted to the general cutting program.

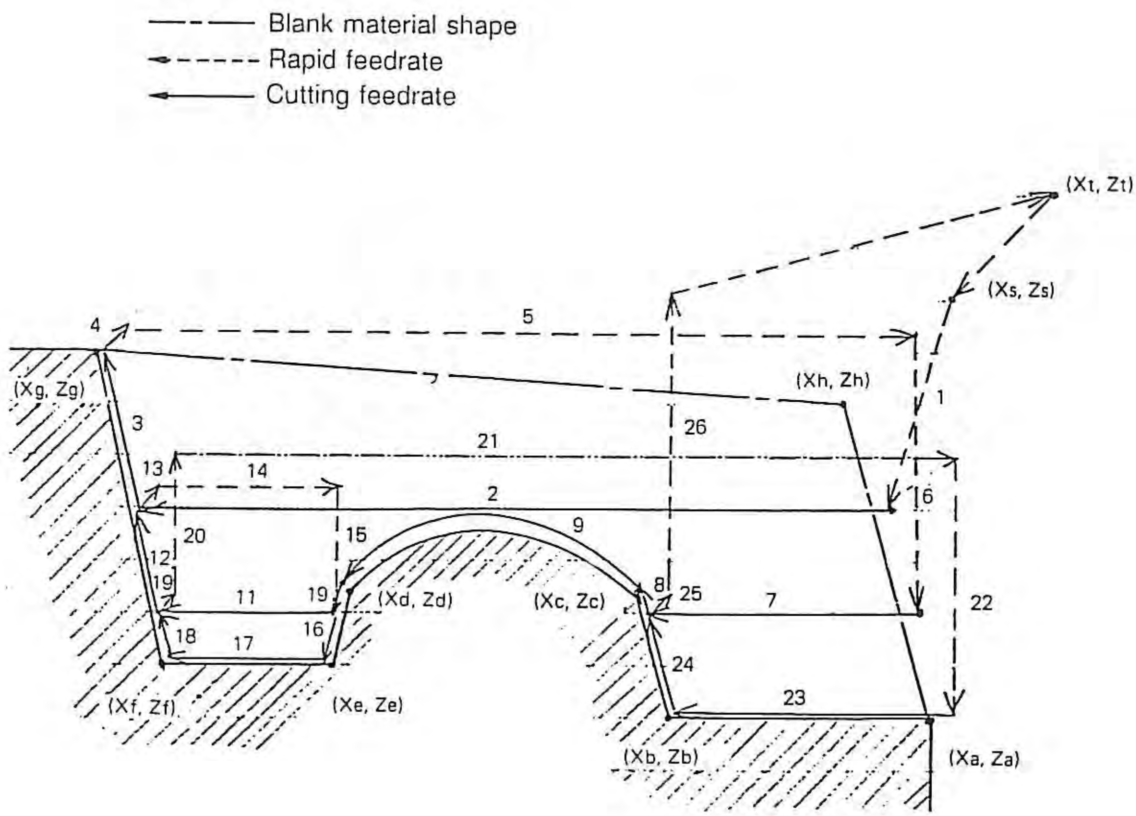


Fig. 23-2 Tool path in the high-speed round bar turning cycle

Input Program

N0001	G00	X_t	Z_t	STM	
N0002		X_s	Z_s		
N0003	G85	N0004 DFUWM85			
N0004	G83				
N0005	G01	X_a	Z_a		
N0006		X_h	Z_h		
N0007		X_g	Z_g		
N0008	G81				
N0009	G00	X_a	Z_a	G42	
N0010	G01	X_b	Z_b	E	
N0011		X_c	Z_c		
N0012	G03	X_d	Z_d	I_d	K_d
N0013	G01	X_e	Z_e		
N0014		X_f	Z_f		
N0015		X_g	Z_g		
N0016	G40				
N0017	G80				

Blank material shape designation block (N0005 – N0007)

Shape designation block (N0009 – N0017)

Output Program Created through Entering ";A" and Pressing the **WRITE** key

N0001	G00	X_t	Z_t	STM	
N0002		X_s	Z_s		
N0003		X_1	Z_1	F	
N0004	G01	X_2	Z_2		
N0005		X_3	Z_3	F	
N0006	G00	X_4	Z_4		
N0007		X_5	Z_5		
N0008		X_6	Z_6		
N0009	G01	X_7	Z_7	F	
N0010		X_8	Z_8		
N0011	G03	X_9	Z_9	I_9	K_9
N0012	G01	X_{10}	Z_{10}		
N0013		X_{11}	Z_{11}		
N0014		X_{12}	Z_{12}		
N0015	G00	X_{13}	Z_{13}		
N0016		X_{14}	Z_{14}		
N0017		X_{15}	Z_{15}		
N0018	G01	X_{16}	Z_{16}	F	
N0019		X_{17}	Z_{17}		
N0020		X_{18}	Z_{18}		
N0021	G00	X_{19}	Z_{19}		
N0022		X_{20}	Z_{20}		
N0023		X_{21}	Z_{21}		
N0024		X_{22}	Z_{22}		
N0025	G01	X_{23}	Z_{23}	F	
N0026		X_{24}	Z_{24}		
N0027	G00	X_{25}	Z_{25}		
N0028		X_{26}	Z_{26}		
N0029		X_t	Z_t		

Output Program Created through Entering ";D" and Pressing the **WRITE** key

N0001	G00	X_t	Z_t	STM	
N0002		X_s	Z_s		
N0003	G85	N0004		DFUWM85	
N0004	G81				
N0005	G00	X_a	Z_a	G42	
N0006	G01	X_b	Z_b	E	
N0007		X_c	Z_c		
N0008	G03	X_d	Z_d	I_d	K_d
N0009	G01	X_e	Z_e		
N0010	G01	X_f	Z_f		
N0011		X_g	Z_g		
N0012	G40				
N0013	G80				

Shape definition block (N0004 – N0013)

(2) Tool Nose Radius Compensation

- G41, G42 : Nose radius compensation command
- G40 : Nose radius compensation cancel command

Example:

The tool path created with the tool nose radius compensation function activated (from G41 (G42) to G40) is converted into the general cutting program.

Input Program						Output Program					
N0001	G00	X_0	Z_0			N0001	G00	X_0'	Z_0'		
N0002	G01	X_1	Z_1	G42 F1	T010101	N0002	G01	X_1'	Z_1'	F1	T010101
N0003		X_2	Z_2			N0003		X_2'	Z_2'		
N0004		X_3	Z_3			N0004		X_3'	Z_3'		
N0005		X_4	Z_4			N0005		X_4'	Z_4'		
N0006		X_5	Z_5			N0006		X_5'	Z_5'		
N0007	G40					N0007	G00	X_6'	Z_6'		
N0008	G00	X_6	Z_6					:			
		:						:			
		:									

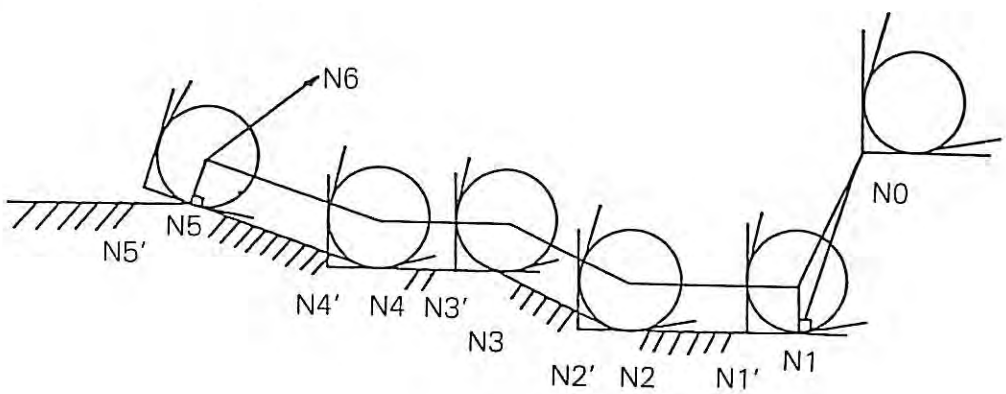


Fig. 23-3 Tool path with the nose radius compensation function activated

(3) Compound Fixed Cycle

- G71 : Longitudinal thread cutting compound fixed cycle
- G72 : Transverse thread cutting compound fixed cycle
- G73 : Longitudinal grooving compound fixed cycle
- G74 : Transverse grooving compound fixed cycle
- M32 : Straight infeed along thread face (on left face)
- M33 : Zigzag infeed
- M34 : Straight infeed along thread face (on right face)
- M73 : Infeed pattern I
- M74 : Infeed pattern II
- M75 : Infeed pattern III

a) Conversion of G71 into G33 (longitudinal thread cutting fixed cycle) of OSP5000

Input Program	Output Program
N0001 G00 X_s Z_s	N0001 G00 X_s Z_s
N0002 G71 X_a Z_a I_i B_d D_d U_u H_h L_1	N0002 G33 X_{a_1} Z_a I_i K_1 L_l F_f J_j M23
$ F_f J_j M23 M32 M75	N0003 X_{a_2} K_2
:	N0004 X_{a_3} K_3
:	N0005 X_{a_4} K_4
	:
	:

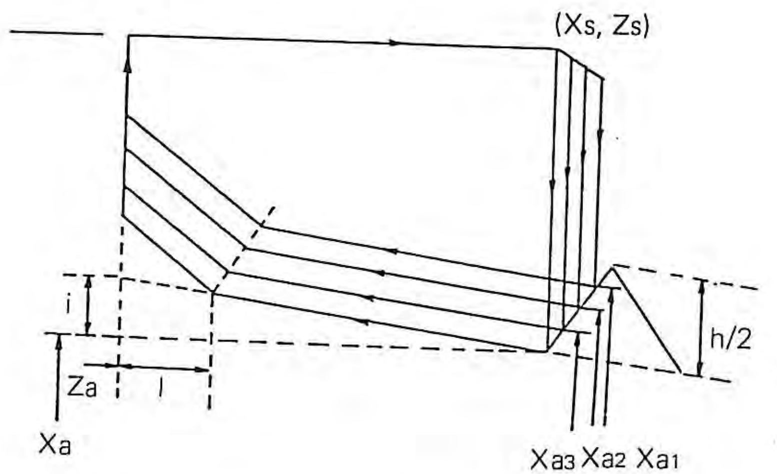


Fig. 23-4 Tool path in the longitudinal thread cutting compound fixed cycle (OSP5000)

b) Conversion of G71 into G33 (longitudinal thread cutting fixed cycle) of OSP3000

Input Program										Output Program					
N0001	G00	X_s	Z_s							N0001	G00	X_s	Z_s		
N0002	G71	X_e	Z_e	I_i	D_d	U_u	F_f	H_h	M_m	N0002			Z_{s_0}		
		:								N0003	G33	X_{e_0}	Z_e	I_i	F_f
		:								N0004	G00		Z_{s_1}		
										N0005	G33	X_{e_1}	Z_e	I_i	F_f
												:			
												:			

Z_{s_0}, Z_{s_1}, ... in the output program is identical to the Z-axis shift amount obtained by the B command in the input program. Z-axis is not shifted unless a B command is designated. (The G00 Z - block is not output.)

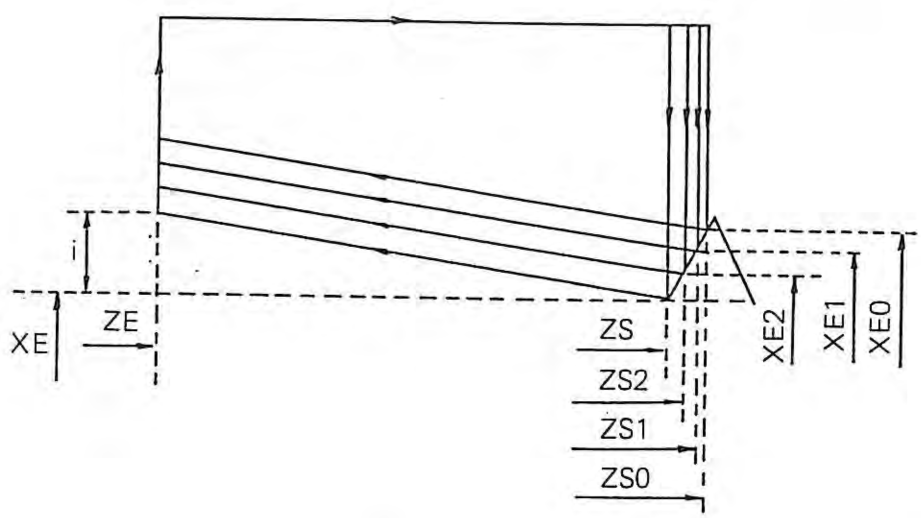


Fig. 23-5 Tool path in the longitudinal thread cutting compound fixed cycle (OSP3000)

Note 1: *An error occurs when an attempt was made to convert the thread cutting compound fixed cycle to the OSP3000 program in the following cases:*

- *M23 (chamfering) and the value for L (chamfer size) are designated.*
- *Transverse thread cutting compound fixed cycle (G72)*

Note 2: *When a J command (number of threads) is designated, the F command value is converted into the value F/J.*

PART II ERROR LIST

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
2105	Device name
2151	Command character
2152	Command syntax
2156	Option
2801	Process number designate error
2802	Process number over error
2803	Auto deciding no use. error
2805	No data setting error
2806	IGF file read error
2807	Input data overflow (+/−) error
2808	Material name input error
2809	Numerical data error
2810	Numerical digit error
2816	File & machine unit system unconformable error
2820	IGF data program overflow error
2821	Program name error
2822	File name error
2823	Program read buffer over error
2825	Spindle rotate direction unmatched error
2826	Process combination impossible error
2827	Data set error
2828	Process change error
2837	IGF file version no. unconformable error
2838	IGF file machine code unconformable error
2839	IGF file spec. code unconformable error
2841	Undefined gauging object process error
2842	Gauging object process unconformable error
2843	Operation expression error
2845	Turret No. error
2846	Incorrect input data item
2847	Shape definition impossible
2848	R-CHF/C-CHF/recess indefinite
2850	Copy command
2851	Continuous designation

Error No.	Error Name
2853	Process sequence table are full
2855	IGF buffer capacity shortage
2856	Process kinds and cutting direction unconformable
2857	B TURRET impossible
2858	Auto process decision
2859	Shape table overflow
2860	Shape element designate
2862	Shape insert impossible
2863	Shape delete impossible
2864	Undefined start point data
2865	Start point blank envelope outside
2866	Shape acceptance impossible
2867	No guide
2868	Undefined shape
2869	Face generating selection impossible
2870	Side generating arc selection impossible
2871	Chamfering cannot be created
2872	Process editing move
2873	Shape change impossible
2874	Start point under end point
2875	Start point under cutting end point
2876	USP file read
2877	Illegal process data
2878	Tool No. no setting
2879	Simultaneous machining
2895	Chamfering direction incorrect
2896	Blank designate data not complete
2898	Bar stock designate data not complete
2899	Chuck and Tail stock designate data not complete
2900	Expression: right part error
2901	Expression: calculation error
2902	Expression: syntax
2903	Expression: buffer over
2904	Program bad direct: G-code
2905	Program factor: M-code error
2906	Program factor: common var. error
2907	Program factor: sequence name error
2908	Program factor: buffer over error
2909	Program factor: illegal char error

Error No.	Error Name
2910	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
2920	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
2930	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
2940	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
2946	Multi cycle: U (W) greater than H error
2947	Multi cycle: X, Z illegal order error
2948	Multi cycle: angle error

Error No.	Error Name
2949	Multi cycle: tool offset error
2950	Multi cycle: cycle start point error
2951	Multi cycle: entry in NOSE-R error
2952	Multi cycle: width error
2953	Chamfering: G01 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
2960	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
2944	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
2970	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
2980	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

Error No.	Error Name
2988	K command
2989	No turret designation
2990	Cannot IGF-convert TG command (N****) error
2991	Cannot IGF-convert OG command (N****) error
2992	Mnemonic
2993	LAP: NOSE-R not canceled error
2994	NOSE-R comp.: NOSE-R > circle-R error
2995	Thread cycle
2996	Synchronize P code
2997	S.M.C. command synchronism
2998	THREAD pitch
2999	Output buffer over error
3000	C-axis connect command
3001	C-axis clamp/unclamp command
3002	C-axis command
3003	Generating calculation
3005	Data word: SB error
3006	Fixed cycle: no spec. error
3007	Fixed cycle: C error
3008	Fixed cycle: I, K error
3009	Fixed cycle: Q error
3010	Fixed cycle: F
3011	Fixed cycle: L
3012	Fixed cycle: D
3013	Fixed cycle: X, Z
3014	Fixed cycle: SA error
3015	Fixed cycle: feed G94 error
3016	A command
3017	Fixed cycle: thread cycle error
3018	Data word: QA error
3019	Data word: X, Y command error
3020	Data word: incremental error
3021	Data word: no X, Y error
3022	Data word: 'Y' error
3023	Data word: distance cal. error
3024	Data word: 'R' error
3025	Generating command code
3026	Generating calculation
3027	G-code

Error No.	Error Name
3028	Syntax
3029	TC
3030	TN
3031	No spec.
3032	ATC TL
3033	MT
3034	U,W command
3036	No spec
3037	NOSE-R comp.:convert of illegal face
3038	C-axis detached
3039	QA command
3500	Automatic process decision
3501	Chucking impossible
3502	Illegal combining information:turning shape
3503	Free blank or average blank undefined
3504	Shape undefined:turning machining
3505	A number of shape element:turning package machining
3506	A number of process:machining sequence determine
3507	Data set incorrect:machining sequence parameter
3508	Machining sequence table setting area overflow
3509	A number of process, balance cut. not created
3510	Process data overflow, balance cut. not created
3511	A number of process :turning special machining
3512	Process data area overflow:turning and special
3513	Generating shape offset cross point cal. impossible
3514	A number of process:M machining
3515	A number of shape element a process of M machining
3516	Process data area overflow:M machining
3517	M machining assignable tool area overflow
3518	Shape undefined:M machining
3519	Temporary storage area overflow:M machining process
3520	Machining/blank shape X minus area enter
3521	A number of shape element a process of turning
3522	Temporary storage area overflow:F endmill process
3523	A number of 凸shape element:turning machining
3524	A number of process:turning machining
3525	A number of shape element:each turning buffer
3526	Arc shape incorrect:turning offset
3527	Remained uncut area

Error No.	Error Name
3528	Illegal setting parameter:M-machining sequence
3529	Tool No. and Tool Offset No. overflow
3530	Decision impossible:M-machining predrill
3531	Decision impossible:grooving tool width
3532	A number of groove, thread recess and tap element
3533	Rev.face and thread process decision impossible
3534	Turning machining tool interference
3535	Turning machining area

2105 Device name

A sector device name not registered was designated when the name of an IGF file, NC program, or USP file was designated.

```
>GE FD2 : A
     ↑
     └── Sector device name
```

2151 Command character

A command not in the command table has been designated.

(This error does not occur if the command is entered using a function key.)

2152 File name

A file name which contains "*" or "?" was designated when the name of an IGF file, NC program, or USP file was designated.

2155 Command syntax

A file name which does not comply with the OSP format was designated when the name of an IGF file, NC program, or USP file was designated.

Code :	3	A device name contains more than 3 characters.
	5	A file name contains more than 20 characters.
	6	A character which is not allowed by the OSP is present.
	7	The main file name or the extension name begins with a numerical character between 0 and 9, or a hyphen "-".

2156 Option

An option character other than "U" was designated when the NC program name was designated.

2801 Process number designate error

The process number specified in the process operation (creation, copy, deletion, etc.) in the process edit is not correct.

2802 Process number over error

When process operation (creation or deletion of process, or creation of balance cut process) was attempted, the number of processes exceeded the maximum allowable size. (Up to 96 processes can be registered.)

2803 Auto deciding no use. error

In prior tool designation, an attempt is made to input "*" at the tool which the automatic process determination function cannot select or at the tool whose tool data is not set.

The tool with which this error occurs cannot be designated as a prior tool.

2805 No data setting error

An attempt to advance the process by pressing the function key [ORDER ↑] although required data has not all been entered in the graphic mode

2806 IGF file read error

Attribute or the size of the IGF data file and USP file designated in the IGF graphic edit process or in the program making process is improper.

2807 Input data overflow (+/−) error

Input data exceeds the input allowable range.

2808 Material name input error

In IGF material data setting operation, improper data setting for material name was attempted.

2809 Numerical data error

More than one decimal point (.) is used in input data.

2810 Numerical digit error

The number of digits of the data to be entered is more than 8.

2816 File & machine unit system unconformable 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.

The unit system used in the USP file and the unit system used in the IGF file differ each other when the USP file is called.

2820 IGF program overflow error

The size of the NC program to be made using the graphic edit function is too large.

2821 Program name error

The 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

The file name does not meet the following requirements.

a) Extension must be .IGF, .MIN, .SUB or .SSB. (.MIN is omissible.)

b) If the extension is .SSB or .SUB, a program name must be specified.

c) The extension name of an USP file must be ".USP".

2823 Program read buffer over error

During the conversion, process data has exceeded the memory capacity and the data cannot be stored in the memory.

The number of characters in the NC program exceeded 1,024 in NC program creation.

2825 Spindle rotate direction unmatched error

In simultaneous four-axis operation with two-saddle specification models, the spindle rotating direction designated for A-saddle differs from that designated for B-saddle.

2826 Process combination impossible error

With two-saddle specification models, simultaneous four-axis operation is attempted using an NC program creation process or multi-machining process.

2827 Data set error

Faulty data is set in shape definition or process editing operation.

Example:

- In basic blank shape definition:

  IN DIA. ID ≧ OUT DIA. OD, or

  IN LENG. IL ≧ OUT LENG. OL

- In basic blank shape definition, a material name which is not listed in the MAT'L TABLE has been designated for MAT'L NAME.

- In shape definition, "E" was entered for the input item which did not allow entry of "E".

- A number larger than 38 was designated for TOOL NO (tool data No.) for B-turret.

2828 Process change error

In process exchange operation in the process edit mode, the cursor is located at C, or a multi-machining process or a gauging process is designated for the exchange between A- and B-turrets.

2837 IGF file version no. unconformable error

The IGF file being loaded does not match the current version.

The USP file being called does not match the current IGF version.

2838 IGF file machine code unconformable error

The machine code of the IGF file being loaded does not match the machine for which the system is set.

The machine code of the USP file being called does not match the machine code of the current IGF file.

Code : ********

Error Code	Machine Type	Error Code	Machine Type
1000000	LC10	10000	LB6
2000000	LC20	20000	LB8
4000000	LC30	40000	LB10
8000000	LC40	80000	LB15
10000000	LC50	100000	LP15
20000000	LS30	200000	LR15
40000000	LH35	400000	LR25
80000000	LH55	800000	LR45

2839 IGF file spec. code unconformable error

Mismatch of IGF file specification

The specification of USP file being called does not match the specification of the current IGF file.

Code : XXX

Bit 0 .... 2-saddle
Bit 1 .... Two-turret model (Mirror image)
Bit 2 .... Multi-machining model
Bit 3 .... LC40-M ATC
Bit 4 .... Tool life management
Bit 5 .... LR15-M ATC
Bit 6 .... Touch sensor
Bit 7 .... Spare
Bit 8 .... Load monitor A
Bit 9 .... Load monitor B
Bit 10 .... Spare
Bit 11 .... Spare

2841 Undefined gauging object process error

The process for which a gauging cycle is executed is not registered as a USP file.

(If the gauging process is registered as a USP file, the process for which the gauging process is executed must be also registered as a USP file.)

2842 Gauging object process unconformable 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 Operation expression error

An error was detected in the calculating expression, or an error occurred during calculation.

Code : 100 .... Improper operator is used.
200 .... Calculation order is improper.
300 .... Calculation stack overflow
400 .... No "]" corresponding to "["
500 .... No "[" corresponding to "]"
600 .... Inconsistency in operand
700 .... Overflow of the number of digits of input data
800 .... More than one decimal point (.) is used in input data.

9XX .... Real number calculation error

XX: Status information

Bit 0 .... 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 0
Bit 5 .... Angle overflow

AXX .... Integer calculation error

XX: Status information

Bit 0 .... 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

2845 Turret No. error

For the LR15M-ATC, L/M designation of the turret position selected does not match the L/M designation of the tool data number.

2846 Incorrect input data item

Calculation is not possible with the data entered.

2847 Shape definition impossible

The shape cannot be defined by the entered data.

2848 R-CHF/C-CHF/recess indefinite

- A shape element which does not allow rounding/chamfering/recessing is present preceding or following the rounding/chamfering/recessing.
- The shape preceding or following the rounding/chamfering cannot be defined as the value for rounding or chamfering is too large. Or calculation is not possible as the value for rounding, chamfering, or recessing is "0".
- Calculation is not possible

2850 Copy command

This error occurs when copy is attempted under the conditions indicated below in the process edit mode.

Code : 1 .... The cursor is located at other than the blank process, or the process numbers from 1 to 96 are all used.

2 .... An attempt was made to copy a gauging process or a multi-machining process to the B-turret side.

2851 Continuous designation

Designation of continuous process execution is not possible in the process edit mode.

Code : 1 .... Continuous process execution was designated with the one-touch IGF for one-saddle models.

The process of the other turret is a blank process or continuous process execution has been already designated for the other turret.

Continuous process execution was designated at the beginning of process sequence.

The previous process is a blank process or a balance cut process.

The process of the turret on the other side is any of the following processes:

Grooving, tapping, NC program creation, Multi-machining, gauging

The position of continuous execution is not correct.

The function key (AMEND) was pressed with the cursor located at "C".

2 .... Continuous process execution is designated in other than the blank process.

2853 Process sequence table are full

No blank process is left in the process table on the screen. Or since a process has been already designated for the last position of the process table, insert operation is not possible.

2855 IGF buffer capacity shortage

The IGF buffer capacity was exceeded due to the operation attempted in the process edit mode and no more process can be created.

Code : 2 .... The IGF buffer capacity is exceeded if process copy is attempted.
3 .... The IGF buffer capacity is exceeded if the contents of the process are corrected.
4 .... The IGF buffer capacity is exceeded if a USP file is called.

2856 Process kinds and cutting direction unconformable

The tool data number which was not set in the tool data of the PBU file was designated in machining method correction in the process edit mode.

When an attempt was made to set the tool data number in machining method correction in the process edit mode, the tool kind and cutting direction designated in the tool data of the PBU file do not agree those in the process being edited.

2857 B TURRET impossible

A gauging process or multi-machining process was designated for the B-turret side in the process edit mode.

2858 Auto process decision

Code : 1 .... Chucking or turning is not possible due to an machining range designation error.
2 .... The designated shape elements intersect.
3 .... The shape of the arbitrary-shaped blank material is not defined.
4 .... The turning shape is not defined.
E .... The shape element which goes across the center axis (Z-axis) is present in the finished shape and blank material shape definition processes.

2859 Shape table overflow

The number of the shape elements in machining shape correction in the process edit mode exceeds 48.

2860 Shape element designate

The shape element designated in shape definition (including insert) cannot be combined with the preceding or following shape element.

Groove, thread, or recess was designated for the undefined shape element.

2862 Shape insert impossible

The function key (INSERT) was pressed when the marker was at the starting point in the shape definition mode or the machining shape correction mode.

2863 Shape delete impossible

The function key (DELETE) was pressed when the marker was at the starting point in the shape definition mode or the machining shape correction mode.

2864 Undefined start point data

Any of the START PT data has not been entered in shape definition.

The function key (BLANK/MACH.) is pressed though the starting point has not been designated in machining shape correction in the process edit mode.

2865 Start point blank envelope outside

The starting point for the turning shape or the arbitrary-shaped blank material shape was designated outside the blank material in shape definition.

2866 Shape acceptance impossible

An attempt was made in shape definition to insert a shape element without establishing the insert mode.

2867 No guide

In the shape definition or process edit mode, the function key (GUIDE ON/OFF) was pressed to enter data for the shape element for which a guide drawing was not provided.

2868 Undefined shape

The function key (ORDER ↑) or (ORDER ↓) was pressed though the undefined shape element was present or the data has not been designated in shape definition.

2869 Face generating selection impossible

An attempt was made to define the shape created through face contour generating machining though the one-touch IGF was not provided with the contour generating machining specification.

2870 Side generating arc selection impossible

An attempt was made to define the shape of CW arc, CCW arc, or rounding R created through side contour generating machining though the one-touch IGF was not provided with the contour generating machining specification.

2871 Chamfering cannot be created

In turning shape definition, an undefined shape element was present when an attempt was made to generate the automatic chamfering. Or the turning shape was not defined in such a manner as "drawing with a single stroke".

2872 Process editing move

An attempt was made to advance to process edit operation though the basic blank material shape has not been defined.

Define the basic blank material shape before advancing to process edit operation.

2873 Shape change impossible

The function key (BLANK SHAPE) was pressed while the finish turning shape was being defined in the machining shape correction mode.

2874 Start point under end point

When entering the hole data in multi-machining shape definition, the coordinate values of the starting point were smaller than the coordinate values of the end point.

2875 Start point under cutting end point

When entering the starting point data for contour generating machining, the coordinate values of the starting point were smaller than the coordinate values of the end point of cut.

2876 USP file read

When a USP file is called in the process edit mode, the properties of the read USP file are not proper. Or the size of the USP file is not proper.

2877 Illegal process data

It does not occur normally.

2878 Tool No. no setting

When process copy or process exchange was attempted in the graphic edit mode, the tool number or the tool group number was not designated.

2879 Simultaneous machining

With the mirror image specification, simultaneous machining by the A- and B-turrets were commanded at the completion of process edit.

2880 IGF-CPU error

An error occurs while one-touch IGF operation is being executed.

Code : 100 .... Time-out in shape definition
101 .... Time-out in setup input
102 .... Time-out in automatic process determination
103 .... Time-out in process edit
104 .... Time-out in process test
105 .... Time-out in program creation
106 .... Time-out in one-touch IGF termination
204 .... MC17 1-bit error
210 .... MC17 ECC error
240 .... MC17 cycle over error

Probable Faulty Locations
: Software, parameter data

Measures to Take : Turn off the power once and back it on again, and re-start one-touch IGF operation from the beginning.

2885 Input data error

When an attempt is made to set the new material data, the material number which can not be designated as the source data is designated.

2895 Chamfering direction incorrect

Faulty data was detected in the starting or end point for chamfering the groove during the execution of process test or when a part program was being created.

2896 Balance cutting process unconformable

An attempt is made to end process edit operation or to carry out process test though the balance cut processes have been separated due to insertion or deletion in the process edit mode.

Delete the process with a balance cut process, or change the position of balance cut process.)

2897 Blank designate data not complete

An attempt was made to execute process test or to create a part program though the basic blank material data has not been designated.

2898 Bar stock designate data not complete

When output of the program for bar feeder or bar puller was designated, an attempt is made to create a part program though the bar material preparation data has not been designated.

2899 Chuck and Tail stock designate data not complete

An attempt is made to create a part program with graphic data though the chuck/center shape data has not been designated on the SETUP screen.

2900 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.)

2901 Expression: calculation error

Calculation alarm

Index	:	TURRET
Character-string	:	None
Code	:	XXYY

XX:

Bit 0 .... Overflow in addition/subtraction
Bit 1 .... Overflow in converting ABS data into integer
Bit 2 .... Conversion from BCD to BIN
Bit 3 .... Conversion from BIN to BCD
Bit 4 .... DROUND, DFIX and DFUP command were programmed in other than mm (inch) unit system.

YY: Floating-point calculation error

Bit 0 .... 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 0
Bit 5 .... Angle overflow

2902 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.

2903 Expression: buffer over

The number of expressions is 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)

2904 Program bad direct: G-code

Illegal G code

A numerical value greater than 199 or less than 0 is assigned to the address character G ($0 \leqq G \leqq 199$).

Index : TURRET

Character-string : None

Code : Hexadecimal number of the programmed numerical value

2905 Program factor: M-code error

Illegal M code

A numerical value greater than 253 or less than 0 is assigned to the address character M ($0 \leqq M \leqq 253$).

Index : TURRET

Character-string : None

Code : Hexadecimal number of the programmed numerical value

2906 Program factor: common var. error

Wrong common variable designation

Index : TURRET

Character-string : None

Code : None .... Characters other than alphanumerics are programmed following programmed.

Others .... Variable number other than $1 \leqq V \leqq 32$ and $901 \leqq V \leqq 932$ is programmed. Variable number other than hexadecimal number of the variable number is programmed.

2907 Program factor: sequence name error

The 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.

2908 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 (this error does not occur usually.)

4 .... Move range of factor data stack is wrong (this error does not occur usually.)

2909 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

2910 Program factor: program name error

The 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

A 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

With two-turret models, the mode is switched between G13 and G14 in the incremental programming mode, tool nose radius compensation mode, LAP mode, or 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

An 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)	
03	]	00	
04	EOR	00	
06	OR	00	
08	AND	00	
0A	NOT	00	
0C	Relative operator	01	LT
		02	LE
		03	EQ
		04	NE
		05	GT
		06	GE
0E		00	
10	Adding/subtracting operator	01	+
		02	–
12	Multiplying/dividing operator	01	*
		02	/
18	Function operator	01	SIN
		02	COS
		03	TAN
		04	ATAN
		05	ATAN2
		06	SQRT
		07	ABS
		08	BIN
		09	BCD
		0A	ROUND
		0B	FIX
		0C	FUP
		0D	DROUND
		0E	DFIX
		0F	DFUP
		10	MOD
1A	[	0	
1C	=	0	
22	Numerical value	Position of decimal point	

2916 Program syntax: 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

The G13 or G14 command is 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

The F word in other than the G04 sequence is negative or "0".

The numerical value of an F word in the G04 sequence is converted into "0.01 sec" unit, it does not satisfy the following inequality:

$0 \leqq F \leqq 99999999$

Index	:	TURRET	
Character-string	:	None	
Code	:	1	F value does not satisfy; $-99999999 \leqq F \leqq 99999999$
		2	F value is either negative or zero.

2920 Data word: 'F' or 'E' error

Illegal F or E command

When an F or E command is converted into "μm/rev" or "0.1 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

The 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
Others .... 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

The numerical value of J word is not:

$0 \leqq J \leqq 99999.999$

Index : TURRET

Character-string : None

Code : 1 .... J value is negative.
2 .... J value does not satisfy:
$0 \leqq J \leqq 99999.999$

2923 Data word: 'K' error

Illegal K command

The numerical value of K word is not:

$-99999.999 \leqq K \leqq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	1 .... K word in circular arc commands Others .... 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

The numerical value of L word in circular interpolation mode is not:

$0 \leqq L \leqq 99999.999$

The chamfering amount in thread cutting fixed cycle calculated from L and K (or I) commands is not 0 through 99999.999.

Numerical value of L word in a gauging cycle is not 0 through 99999.999.

In other modes, numerical value of L word is not:

$-99999.999 \leqq L \leqq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	1 .... In the circular interpolation mode, the L word does not satisfy; $-99999.999 \leqq L \leqq 99999.999$ 2 .... The L value is negative in the circular interpolation mode. Others .... Hexadecimal number of the L value in other than the circular interpolation mode

2925 Data word: 'P' error

Illegal P command

The 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

The numerical value of S is not:

$0 \leqq S \leqq 9999$

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:

$0 < T < 99999999$

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

The numerical value of X word is not:

$-99999.999 \leqq X \leqq 99999.999$

Or the X word programmed in incremental word is not $-99999.999 \leqq X \leqq 99999.999$ when converted into the absolute value.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of the programmed X word

2930 Data word: 'X', 'Z' error

The first block in 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 G30 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

The numerical value of Z word is not:

$-99999.999 \leqq Z \leqq 99999.999$

Or the Z word programmed in incremental word is not $-99999.999 \leqq Z \leqq 99999.999$ 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 G102 or G103 mode.

2932 Data word: arc cal. error

In direct arc radius command, the coordinates of the arc center cannot 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.: 20 μm).

Index : TURRET

Character-string : None

Code :

1 .... The 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 tolerance.

4 .... Arc radius I is zero.

10 .... 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 G00, G01, G02, G03, 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.
Others .... 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 G102 and G103 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 G102 and G103 sequence

2935 Data word: C command error

Alarm of C command

The numerical value of programmed C command calling for spindle orientation is either $C < 0°$, or $C \geqq 360°$.

A C command is programmed at B-turret side on a multi-machining model.

A C command is programmed in other than G00, G01, G50, G101, G102 and G103 modes.

A C command value is outside of $-360° < C < 360°$.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... C command at B-turret side 2 .... C command in other than G00, G01, G50, G101, G102 and G103 3 .... C command calling for zero movement in the G101 mode Others .... Hexadecimal number of programmed C value

2936 Multi cycle: B illegal order error

$B < 0°$ or $B \geqq 180°$

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 the G71, G72, G73 or G74 mode, no D command is programmed or the numerical value of D command is not:

$0 < D \leqq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF .... No D command Others .... Hexadecimal number of D value

2953 Chamfering: G01 mode error

Chamfering commands are programmed in other than G01 mode.

Index : TURRET

Character-string : None

Code : None .... No G code programmed
2 .... G02
3 .... G03
1F G31
20 .... G32
21 .... G33
22 .... G34
23 .... G35
FE G00

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:

$-99999.999 \leqq L \leqq 99999.999$

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:

$-99999.999 \leqq X\ (Z) \leqq 99999.999$

Index : TURRET

Character-string : None

Code : FFFFFFFF .... Either both X or Z command is programmed, or neither X nor Z is programmed.
Others .... Hexadecimal number of X or Z value

2957 LAP: B illegal order error

The B command specifying the tool tip angle in the G88 LAP mode is either $B < 0°$ or $B \geqq 180°$.

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 may digits.

Index	:	TURRET
Character-string	:	None
Code	:	1 .... Programmed D value is not: $-99999.999 \leqq D \leqq 99999.999$
		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 or DB is not: $-99999.999 \leqq DA\ (DB) \leqq 99999.999$
		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: $-99999.999 \leqq XA (XB) \leqq 99999.999$

2 .... In transverse cycle, programmed ZA or ZB command is not: $-99999.999 \leqq ZA (ZB) \leqq 99999.999$

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 0 .... 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 0
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.

2945 Multi cycle: U (W) illegal order error

In the G71 thread cutting cycle, either W command is programmed or the numerical value of U command is not:

$0 < U \leqq 99999.999$

In G72 thread cutting cycle, either U command is programmed or the numerical value of W command is not:

$0 < W \leqq 99999.999$

Index	:	TURRET		
Character-string	:	None		
Code	:	None	....	In G71, W command is programmed, or in G72, U command is programmed.
		Others	....	Hexadecimal number of U (W) value

2946 Multi cycle: U (W) greater than H error

In the 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 the G71, G72, G73 or G74 mode, either X or Z command is not programmed, or the value of them is not:

$-99999.999 \leqq X (Z) \leqq 99999.999$

Index	:	TURRET		
Character-string	:	None		
Code	:	FFFFFFFF	....	Either X or Z command is not programmed.
		Others	....	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 0	....	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 0
		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 from the programmed shift direction.

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of tool offset shift amount

2950 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

2938 Multi cycle: F illegal order error

In the G71, G72, G73 or G74 mode, no F command is programmed or numerical value of F command is not:

$0 < F \leqq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF .... No F command Others .... Hexadecimal number of F value

2939 Multi cycle: H illegal order error

In the G71 or G72 thread cutting mode, no H command is programmed or numerical value of H command is not:

$0 < H \leqq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF .... No H command Others .... Hexadecimal number of H value

2940 Multi cycle: H-U (W) less than D (M73) error

In the 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 or 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:

$0 < I, K \leqq 99999.999$

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. Others .... Hexadecimal number of I or K value

2943 Multi cycle: L illegal order error

In the G73 or G74 grooving cycle, numerical value of L command is not:

$0 < L \leqq 99999.999$

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

2969 LAP: LAP control error

LAP control is impossible. (Overflow of control counter for LAP control)

Index : TURRET

Character-string : None

Code : None

2970 LAP: G-code error

No G80 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 G80 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 0 .... 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 0
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

The tool nose radius compensation mode canceling G40 code is programmed in other than the G00 or G01 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 G00 or G01 mode.
10 .... 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.
40 .... G40 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.
20 .... 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

2980 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

G88 (continuous thread cutting mode) or G82 (end face direction) cannot be converted to the OSP3000 program.

2984 Cannot IGF-convert G32 (N****) error

The G32 code (end face thread cutting command) cannot be converted for OSP3000 programs.

2985 Cannot IGF-convert G72 (N****) error

The G72 code (compound fixed end face thread cutting cycle) cannot be converted for OSP3000 programs.

2986 Cannot IGF-convert B command (N****) error

If B (turret) is not "0", the G71 (longitudinal thread cutting compound cycle) cannot be converted for OSP3000 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 OSP3000 program.

2988 K command

Conversion to the OSP3000 program is not possible when the Z-axis shift amount K is other than "0" in the compound fixed thread cutting cycle.

2989 No turret designation

With two-saddle specification models, G13 or G14 (turret selection code) is not designated in the program to be converted.

2990 Cannot IGF-convert TG command (N****) error

The TG (tool group) command cannot be converted for OSP3000 program.

2991 Cannot IGF-convert OG command (N****) error

The OG (offset group) command cannot be converted for OSP3000 program.

2992 Mnemonic

An attempt was made to convert the improper address character. Or an attempt was made to convert the data Which has more than allowable digits or the data who contains more than one decimal point. Or an attempt was made to convert the common variable or system variable.

2993 LAP: NOSE-R not canceled error

Nose radius compensation mode is not canceled at the end of LAP (sequence containing G80).

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.

2995 Thread cycle

The direction of thread cut is reversed as the shift amount designated in the fixed thread cutting cycle command was too large. Or the coordinate values of the end point of taper are larger than those of the starting point.

Index : TURRET

Character-string : None

Code : 1 .... The coordinate values of the end point of taper are larger than those of the starting point as the values designated by G33 and/or G32 are too large.

2996 Synchronize P code

With two-saddle models, P commands are not assigned orderly from small to large. Negative values are not the case.

Index : TURRET

Character-string : None

Code : 1 .... A P command which has a smaller value than the P command in execution is present.

2 .... Faulty negative values are designated for A-/B-turret.

2997 S.M.C. command synchronism

S, M, C, or constant peripheral speed command for A-turret is different from that for B-turret.

Index : TURRET

Character-string : None

Code : 1 .... Different constant speed commands (G97, G96) are designated.

2 .... Designation of constant peripheral speed command (G110, G111) is not correct.

3 .... Different S commands are designated.

4 .... Different spindle rotating commands (M03, M04, M05) are designated.

5 .... Different spindle speed range commands (M40, M41, M43, M44) are designated.

6 .... Different program stop commands (M00, M01) are designated.

7 .... Different orient-stopped angle (M19) are designated.

8 .... M109 (Cancel of C-axis connect) is not designated by the end of program.

2998 THREAD pitch

Pitch is "0" or negative in the variable lead thread cutting cycle.

Index : AXIS

Character-string : None

Code : None .... Pitch is negative or "0".

XX .... Floating decimal point calculation error

Bit 0 .... 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 0
Bit 5 .... Angle overflow

2999 Output buffer over error

During the conversion, output program buffer becomes full.

3000 C-axis connect command

The C-axis connect command was not executed correctly.

Index : TURRET

Character-string : None

Code : 2 .... The C-axis connect command was designated while the spindle was not at a still.

4 .... The command was designated while the C-axis was clamped.

3001 C-axis clamp/unclamp command

The C-axis clamp/unclamp command was designated when the C-axis was not connected.

Index	:	TURRET
Character-string	:	None
Code	:	1

3002 C-axis command

The C-axis movement command was designated while the C-axis is connected but not unclamped.

Index	:	None
Character-string	:	None
Code	:	1 .... C-axis is not connected. 2 .... C-axis is not unclamped.

3003 Generating calculation

An calculation error has occurred during profile generation function in G101, G102, or G103.

Index	:	TURRET
Character-string	:	None
Code	:	XYY

X:

1 .... Former half of RCON calculation in G101
2 .... Latter half of RCON calculation in G101
3 .... Former half of RCON calculation in G102 or G103
4 .... Latter half of RCON calculation in G102 or G103

YY:

Bit 0 .... 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 0
Bit 5 .... Angle overflow for SIN, COS, TAN, and COT

3005 Data word: SB error

Programmed SB command is not:

$0 \leqq SB \leqq 9999$

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

3006 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

3007 Fixed cycle: C error

Programmed C value is not:

$-360° < C < 360°$

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of programmed C value

3008 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:

$0 \leqq I, K \leqq 99999.999$

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 Others .... Hexadecimal number of I and K values

3009 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

3010 Fixed cycle: F

Programmed F value is either 0 or negative.

No F command

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF .... No F command Others .... Hexadecimal number of programmed F value

3011 Fixed cycle: L

Programmed L value is not:

$0 \leqq L \leqq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	Hexadecimal number of programmed L value

3012 Fixed cycle: D

Programmed D value is not:

$0 \leqq D \leqq 99999.999$

Index	:	TURRET
Character-string	:	None
Code	:	FFFFFFFF .... No D command Others .... Hexadecimal number of programmed D value

3013 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:

$-99999.999 \leqq X, Z \leqq 99999.999$

Index : TURRET

Character-string : None

Code : FFFFFFFF .... No X and Z command
Others .... Hexadecimal number of programmed X and Z value

3014 Fixed cycle: SA error

Programmed SA value is not:

$0 < SA \leqq 20$

No SA command

Index : TURRET

Character-string : None

Code : FFFFFFFF .... No SA command
Others .... Hexadecimal number of programmed SA value

3015 Fixed cycle: feed G94 error

G185 through G188 is programmed in the G94 mode.

Index : TURRET

Character-string : None

Code : None

3016 A command

Calculation error has occurred when the taper angle was designated in the fixed thread cutting cycle for multi-machining specification models.

Or the calculated value for taper side (I ... longitudinal, K ... end face) is not:

$-99999.999 \leqq I, K \leqq 99999.999$

Index : None

Character-string : None

Code : Hexadecimal number of programmed A value

3017 Fixed cycle: thread cycle error

In thread cutting cycle, programmed I or K value is too large.

Index	:	TURRET
Character-string	:	None
Code	:	1

3018 Data word: QA error

QA command is programmed for B-turret.

QA command is programmed in other than the G00 and G01 mode.

Programmed QA value is not:

$0 \leqq QA \leqq 1999$

Index	:	TURRET
Character-string	:	None
Code	:	1 .... QA command is programmed for B-turret. 2 .... QA command is programmed in other than G00 and G01 mode. Others .... Hexadecimal number of QA value

3019 Data word: X, Y command error

In coordinate system conversion, both X and Y are assigned "zero".

Index	:	TURRET
Character-string	:	None
Code	:	None

3020 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

3021 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

3022 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

3023 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

3024 Data word: 'R' error

In a block containing G181 through G184 or G189, either R0 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

3025 Generating command code

Contour generation commands G101, G102, and/or G103 were designated for B-turret.

Index	:	TURRET
Character-string	:	None
Code	:	Commanded code

3026 Generating calculation

Floating point calculation error has occurred in the preparation processing for the function generation of the G101, G102, and G103 main task.

Index	:	TURRET
Character-string	:	None

Code : XYY

X:

1 .... Error in G101 processing
2 .... Error in G102/G103 processing

YY:

Bit 0 .... 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 0
Bit 5 .... Angle overflow for SIN, COS, TAN, and COT

3027 G-code

An attempt was made to convert G101, G102, and/or G103 commanded in the multi-machining specification to the IGF program.

Index : TURRET

Character-string : None

Code : Designated G code

3028 Syntax

The ATC command was designated for B-turret.

Index : TURRET

Character-string : None

Code : None

3029 TC

The TC command value is not: $1 \leqq TC \leqq 4$

Index : TURRET

Character-string : None

Code : Hexadecimal number of the designated TC value

3030 TN

The TN command value is not: $0 \leqq TN \leqq 63$

Index : TURRET

Character-string : None

Code : Hexadecimal number of the designated TN value

3031 No spec.

The ATC command was designated though the ATC specification was not selected.

Index : TURRET

Character-string : None

Code : None

3032 ATC TL

With the LR15-M ATC, the tool offset number and the tool nose radius compensation number designated by the TL command were not correct.

Index : TURRET

Character-string : None

Code : None

3033 MT

With the LR15-M ATC, the tool number designated by the MT command was not correct.

Index : TURRET

Character-string : None

Code : None

3034 U,W command

Index : TURRET

Character-string : None

Code : None

3036 No spec.

The cutter radius compensation ON command (G41, G42) was designated in the contour generating machining mode though the cutter radius compensation function was not selected.

Index : TURRET

Character-string : None

Code : None

3037 NOSE-R comp.:convert of illegal face

The face to be compensated for was changed during the execution of cutter radius compensation or tool nose radius compensation.

Index : TURRET

Character-string : None

Code : None

3038 C-axis detached

The cutter radius compensation ON command was designated while the C-axis connect command was not designated.

Index	:	TURRET
Character-string	:	None
Code	:	None

3039 QA command

The QA command was designated during the execution of tool nose radius compensation.

Index	:	TURRET
Character-string	:	None
Code	:	None

3040 Shape designation

An abnormal command is given in shape definition in the LAP mode.

Code :
- 1 .... Overflow in coordinate value
- 3 .... Circular interpolation command in the first block
- 4 .... No I and K commands in the circular interpolation mode
- 5 .... Faulty circular interpolation command
- 6 .... Excess in blank material shape element number
- 7 .... Excess in finished workpiece shape element number
- 8 .... Faulty depth of cut (More than 33 infeed operations are necessary)
- 9 .... Blank material shape reversed (shape element opposite to the cutting direction)
- 10 .... Finished workpiece shape reversed (shape element opposite to the cutting direction)
- 11 .... No blank material shape element (only one blank material shape designation)

3500 Automatic process decision

An error occurred during the execution of automatic process determination.

3537 Finishing process decision : start and end point

There is a possibility of uncut or overcut at the cutting starting point or cutting end point in the finish machining process.

Code : XXYY

XX: Hexadecimal number of process number

YY:

01 .... An error occurs at the cutting starting point

02 .... An error occurs at the cutting end point

Example: Display of "302"

An error occurs at the cutting starting point of the third process.

3538 R-CHF can not be cut

Rounding R cannot be cut accurately with the tool nose radius compensation function activated. There is a possibility of overcut.

Code : XX: Hexadecimal number of the error-occurring process number

Measures to Take : The cutting tool with the nose R equal to the setting of the dimension parameter No. 44 (ROUGHING NOSE-R MAXIMUM VALUE) cannot cut rounding R accurately.

Set a smaller value to the dimension parameter No. 44 and retry automatic process determination, or correct the shape in the process edit mode.

3539 Prior tool is never used by auto deciding

The tool which can not be selected in automatic process determination (for example a straight tool) is designated as the prior tool.

Code : Designation of the turret, A or B, and the tool data number

Measures to Take : Cancel the prior tool designation on the condition setting screen.

Register the tool which can be designated as the prior tool in the tool data.