*Computer Numerical Controls*

*Manual Guide*

*for Lathe*

*Operator's Manual*

**FANUC**

**TECHNOLOGY AND MORE**

In this manual we have tried as much as possible to describe all the various matters.
However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities.
Therefore, matters which are not especially described as possible in this manual should be regarded as "impossible".

# SAFETY PRECAUTIONS

This manual includes safety precautions for protecting the user and preventing damage to the machine. Precautions are classified into Warnings and Cautions according to their bearing on safety. Also, supplementary information is described as Notes. Read the Warnings, Cautions, and Notes thoroughly before attempting to use the machine.

---
⚠ **WARNING**
> Applied when there is a danger of the user being injured or when there is a damage of both the user being injured and the equipment being damaged if the approved procedure is not observed.

---
⚠ **CAUTION**
> Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

---
**NOTE**
> The Note is used to indicate supplementary information other than Warning and Caution.

---

\* Read this manual carefully, and store it in a safe place.

# *1.1*      FOR SAFETY DURING OPERATION

To ensure safety while using a machine featuring the MANUAL GUIDE function that is made according to this programming manual, observe the following precautions:

---

⚠ **WARNING**

1  Confirm, on the screen, that the data has been entered correctly before proceeding to the next operation. Attempting operation with incorrect data may cause the tool to strike the work-piece or machine, possibly breaking the tool or machine or injuring the operator.

2  Before starting the machine using the tool compensation function, carefully determine the direction of compensation and the compensation value, and ensure that the tool will not strike the work-piece or machine. Otherwise the tool or machine may be damaged or the operator may injured.

3  Before playing back an operation after teaching it, check the tool paths on the confirmation screen to ensure that the tool will not strike the work-piece or machine. Otherwise, the tool or machine may be damaged or the operator may be injured.

4  To use constant surface speed control, set a value allowed for the work-piece and work-piece holding unit for the maximum spindle speed data item. If the set value is not appropriate, the work-piece or holding unit may fly due to centrifugal force, possibly breaking the machine or injuring the operator.

5  Set all necessary parameters and data items before starting MANUAL GUIDE operations. Note that if the cutting conditions are not suitable for the work-piece, the tool may be damaged or the operator may be injured.

6  After teaching a machining program created using MANUAL GUIDE functions, do no run the machine on that program immediately. Instead, confirm every step of the resultant program, and make sure that the tool path and machining operation are correct and that the tool will not strike the work-piece or machine. Before starting production machining, run the machine with no work-piece attached to the machine to make sure that the tool will not strike a work-piece or the machine. If the tool strikes the machine and/or work-piece, the tool and/or machine may be damaged, and even injuries the operator.

---

⚠ **CAUTION**

After pressing the power-on button, do not touch any keys on the keyboard until the initial screen appears. Some keys are used for maintenance or special operations such that pressing such a key may cause an unexpected operation.

---

# TABLE OF CONTENTS

# APPENDIX

# I. GENERAL

# 1    OVERVIEW

## Overview of the manual

This manual describes the functions related to 1-path Lathe MANUAL GUIDE of Series 16*i*/18*i*/21*i*-TA/TB.

For other functions, refer to the operator's manual for the Series 16*i*/18*i*/21*i*-TA/TB.

The specifications and usage of the MANUAL GUIDE may vary according to the specifications of the operator's panel of a machine tool. Be sure to read the manual provided by the machine tool builder.

The functions of the CNC machine tool system are determined not only by the CNC, but by the combination of the machine tool, the power magnetic circuit in the machine tool, the servo system, the CNC and the operator's panel.

It is impossible to cover all possible combinations of all functions, programming methods, and operations in a single manual.

This manual explains only the MANUAL GUIDE operations provided for the CNC. For individual CNC machine tools refer to applicable manuals from the machine tool builders. The manuals from machine tool builders take precedence over this manual.

This manual explains as many detailed functions as possible. However, it is not possible to describe all of the items which cannot be done or which the operator must not do. Therefore, please assume that functions other than those described in this manual cannot be performed.

Detailed information and special conditions are explained in notes. The readers may encounter new technical terms in the notes not previously defined or described. In this case, read this manual through first, then review the details.

# 2 GUIDANCE HANDWHEEL

## Guidance Cutting and Guidance Handwheel

When a handwheel of a general-purpose lathe is rotated, the tool moves along only one axis. To carry out machining along a tapered or arc figure, an operator must rotate two handwheels simultaneously in synchronization. It is almost impossible to ensure accurate machining with this method.
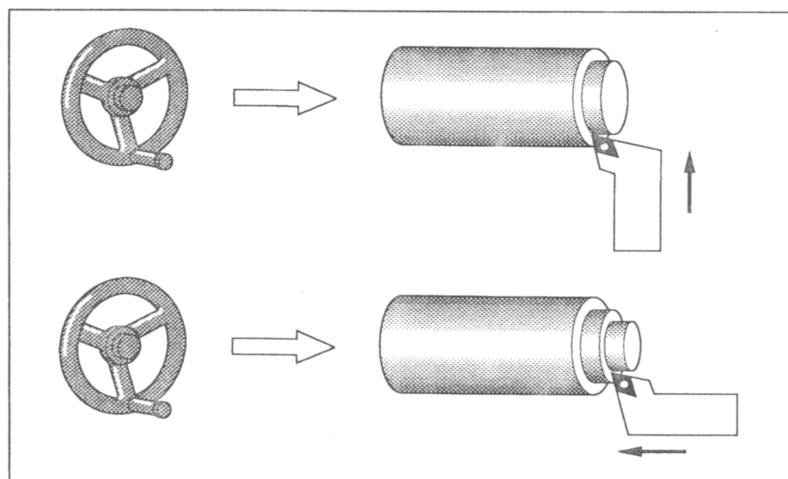


**Machining with a General Purpose Lathe**

The Guidance Cutting eliminates the limitation of the general purpose lathe and enables sophisticated machining.

A guidance handwheel is provided so that an operator of the general purpose lathe can easily implement the function.

When the MANUAL GUIDE is used, just a few turns of the single guidance handle are required to carry out machining along a tapered or arc figure.
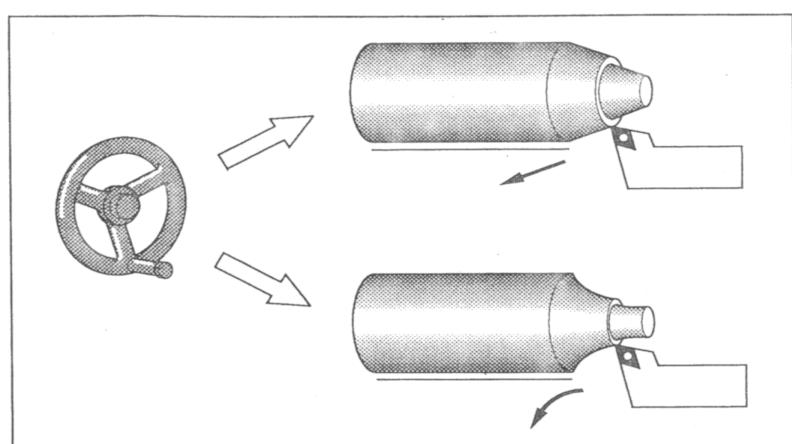


**Rotating the Guidance Handwheel for Machining**

- 4 -

After cutting a workpiece by using the above guidance cutting, this cutting motion can be taught into a machining program as an one block motion command, and these taught-in motion command can be used for playback machining.

## Synchronous Feed and Guidance Handwheel

During running a machining program under automatic executing mode, an operator can control a tool motion by using guidance handwheel.

By selecting a synchronous feed mode on guidance handwheel, an operator can make a tool moved by the feedrate which is synchronized with handwheel rotating instead of inputted feedrate in the machining program.

# 3

# ALL-IN-ONE SCREEN

In MANUAL GUIDE, basically, only one screen called All-in-one Screen is used for all the operations for it.

Title area                                                    CNC status area

| FANUC MANUAL GUIDE | | | |
|---|---|---|---|
| Actual Position & Servo Motor Load Meter | Distance to go | Spindle speed & Spindle mode | Program & Process Number & Others |
| | Next block motion | Feedrate & Handwheel mode | |

Status indicator window

Program (Process data)

Graphic window

Program (Figure data)

Key-in buffer

Soft-keys        Pop-up window        Program window
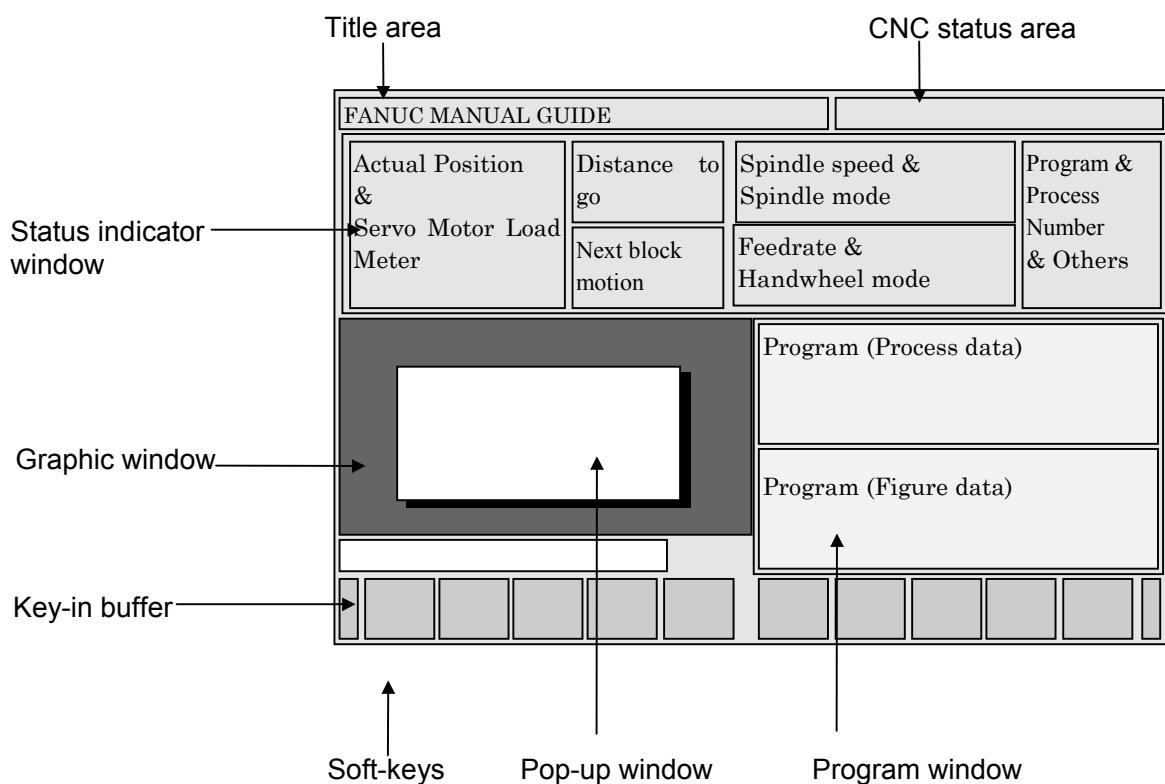
Title area :
    The title of MANUAL GUIDE is displayed always.
CNC status area :
    Following CNC status information are displayed always.
    • Mode
    • Alarm status
    • Reset or Emergency stop status
    • Actual time
Status indicator window :
    Following CNC status information are displayed always.
    • Actual machine position and Servo load meter
    • Moving distance of actual and next block
    • Feedrate and Guidance handwheel mode
    • Spindle rotating speed and mode
    • Actual program and process number
    • Actual command by machining program

Graphic window:
 Following graphical drawing are displayed as occasion demands.
- Taught-in tool path of guidance cutting and tracing cutting
- Animated drawing of machining simulation
- Tool path of machining simulation

Program window :
 Following program information are displayed as occasion demands.
- Actual process name
- Process data of actual process
- Figure data of actual process

Pop-up window :
 Following supplemental screens are displayed as occasion demands.
- Tool file
- Set-up guidance for work coordinate shift amount
- Program list
- Process list
- Supplemental program data

Key-in buffer :
 Inputting numerical data and its comment are displayed as occasion demands.

Soft-keys :
 Following soft-keys comment for indicating its contents are displayed as occasion demands.
- Machining type menu
- Data item menu
- Teach-in type menu
- Operating key
- Pop-up window menu

# 4 SYMBOLS USED

The following explains how keys and buttons are indicated in this manual.

(1) Functions buttons are indicated in bold type :
Example) **PRGRM**, **OFSET**

(2) Numeric keys to be entered from key-board are underlined :
Example) 12.345

(3) The input key is indicated in bold type in the same way as the function buttons. Generally, the input key follows numeric key input :
Example) 12.345   **INPUT**

(4) Soft-keys are enclosed in brackets [ ] :
Example) [ LIST ], [LINE ]

(5) The cursor keys are indicated by the following symbols :
Example) $\uparrow$, $\downarrow$, $\leftarrow$, $\rightarrow$

(6) The page keys are indicated by the following symbols :
Example) $\uparrow$, $\downarrow$

# II. OPERATION

# 1    OVERVIEW OF THE PROCEDURE

# *1.1*    **THE MANUAL GUIDE CONCEPT**

By using MANUAL GUIDE, an operator can handle a newly equipped machine from simple cutting using a handwheel to complicated machining step by step. He will be able to carry out them without making a detour.

On MANUAL GUIDE, machining operations can be classified to 3 steps by an operator's programming skill.



MANUAL GUIDE Performance Chart

**Cycle Cutting**
- Bar machining
- Grooving
- Threading
- Necking
- Drilling
- Tapping

**Guidance / Single Cutting**
by guidance handwheel

**Manually Cutting**
by normal handwheel

Complexity of Machining

Operator's Programming Skill
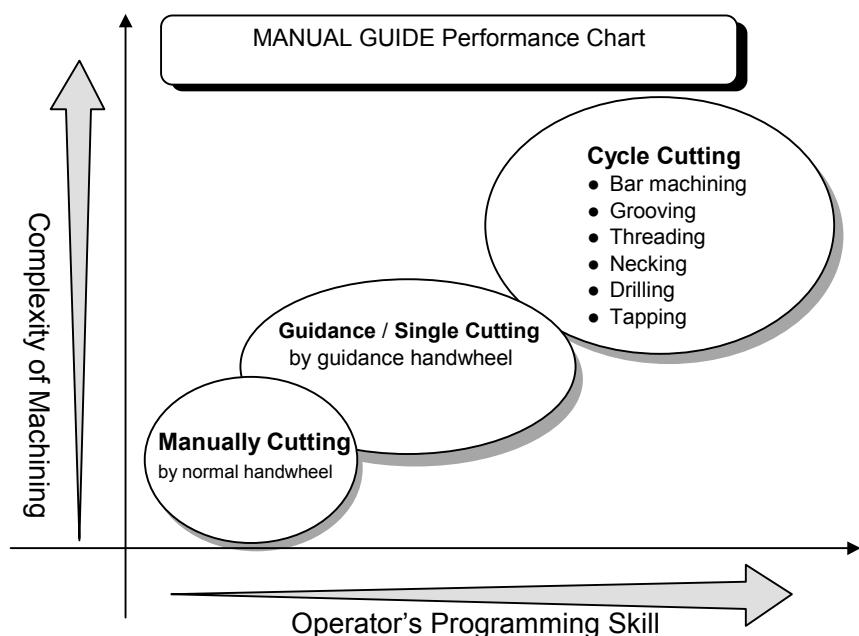
## **Manually Cutting**

As the $1^{st}$ step after machine equipment, without any data inputting nor programming, by using normal handwheel for X/Z-axis, an operator can carry out a simple machining such as straight longitudinal or radial cutting, still more drilling.

Since this normal handwheel is available at anytime during idling status of MANUAL GUIDE, an operator carry out this handwheel cutting seeing a position display on an All-in-one screen.

This should be familiar to an operator has been using a manual lathe, mechanically driven lathe.

In addition to moving tool function, auxiliary function switches are prepared on a machine operator's panel.

## Guidance / Single Cutting

As the 2nd skill up step, an operator can carry out a taper and circular cutting by using guidance handwheel.

There are 2 types for this cutting.

1) Guidance Cutting

By inputting a target line or circle figure data, an operator can carry out an approach cutting or along cutting by using a guidance handwheel.

Still more, by teaching these cutting motion blocks into CNC memory, an operator can carry out a playback machining repeatedly by using them, and at this time, he can control a tool motion by using a guidance handwheel in synchronous mode.

2) Single Cutting

By inputting a tool motion path of line or circle, an operator can carry out these cutting motion blocks, and he can control a tool motion by using a guidance handwheel in synchronous mode.

At the time of inputting a tool path data, simple figure calculating such as end coordinate calculating can be used.

In addition to the above cutting function, similar to the manually cutting method, auxiliary function switches prepared on a machine operator's panel can be used. Still more, these auxiliary function can be taught-in with the above cutting motions and used for playback machining.

## Cycle Cutting

As the final skill up step, an operator can carry out a complicated machining by inputting necessary program data for cycle cutting.

The following types of cycle cutting are available

1) Bar machining (Outer, Inner, End face)
2) Grooving (Standard, Trapezoidal)
3) Threading (General, Metric, Unified, PT, PF)
4) Necking (DIN509-E, DIN509-F, DIN76)
5) Drilling (Center-drilling, Drilling, Reaming, Boring)
6) Tapping

Especially, in Bar machining, strong contour calculating functions are prepared for inputting a free contour form.

In these cycle cutting procedures, necessary auxiliary functions such as spindle rotation or tool changing are automatically outputted, so an operator can carry out a complicated machining by only pushing a cycle starting button.

# *1.2*   **FLOWCHART OF OPERATIONS**

The following shows the general procedure from preparing to start making a machining program to executing a playback machining by using MANUAL GUIDE.

Reference

```
┌─────────────────────┐
│  Setting parameters │               APPENDIX
└─────────────────────┘
           │
┌─────────────────────┐
│ Setting tool file data │            II. 9.   SETTING DATA
└─────────────────────┘
           │
      ◇ Guidance cutting ? ◇───── no
           │ yes
┌─────────────────────┐
│   Guidance cutting  │
└─────────────────────┘
           │
┌─────────────────────┐
│  Teach-in and Save  │              II. 4. GUIDANCE CUTTING
└─────────────────────┘
           │
┌─────────────────────────────┐
│ Repeat cutting or check drawing │
└─────────────────────────────┘
           │
      ◇ Tracing cutting ? ◇───── no
           │ yes
┌─────────────────────┐
│   Tracing cutting   │
└─────────────────────┘
           │
┌─────────────────────┐
│  Teach-in and Save  │              II.5.   TRACING CUTTING
└─────────────────────┘
           │
┌─────────────────────────────┐
│ Repeat cutting or check drawing │
└─────────────────────────────┘
           │
      ◇ Cycle motions ? ◇───── no
           │ yes
┌──────────────────────────────────┐
│ Input data, Check drawing, Test cutting │
└──────────────────────────────────┘
           │                        II.6.   CYCLE MOTIONS
┌─────────────────────┐
│        Save         │
└─────────────────────┘
           │
  no ──── ◇ Program end ? ◇
           │ yes
┌─────────────────────────┐
│ Check by animated drawing │        II.8.   PLAYBACK
└─────────────────────────┘                 MACHINING
           │                                 and
┌─────────────────────┐                      MACHINING
│  Playback machining │                      SIMULATION
└─────────────────────┘
```

# 2    DESCRIPTION OF THE KEYBOARD

## 2.1     9.5"/10.4" LCD-MOUNTED TYPE CNC CONTROL UNIT
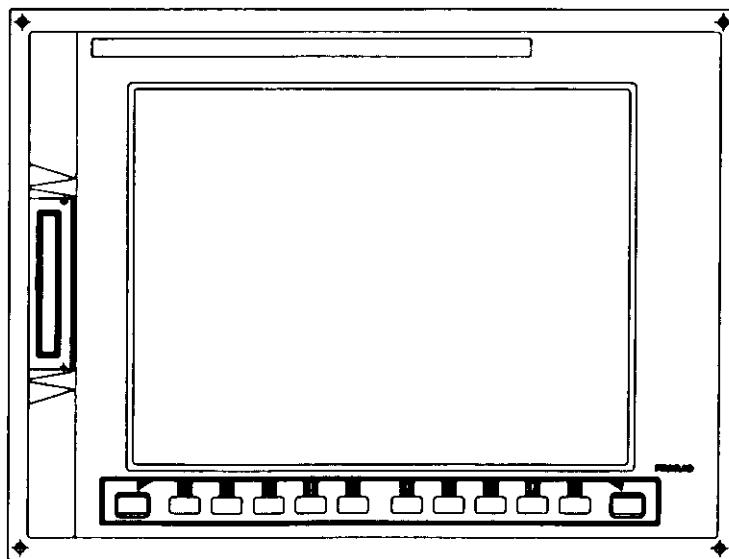


## 2.2     STAND-ALONE TYPE SMALL MDI UNIT
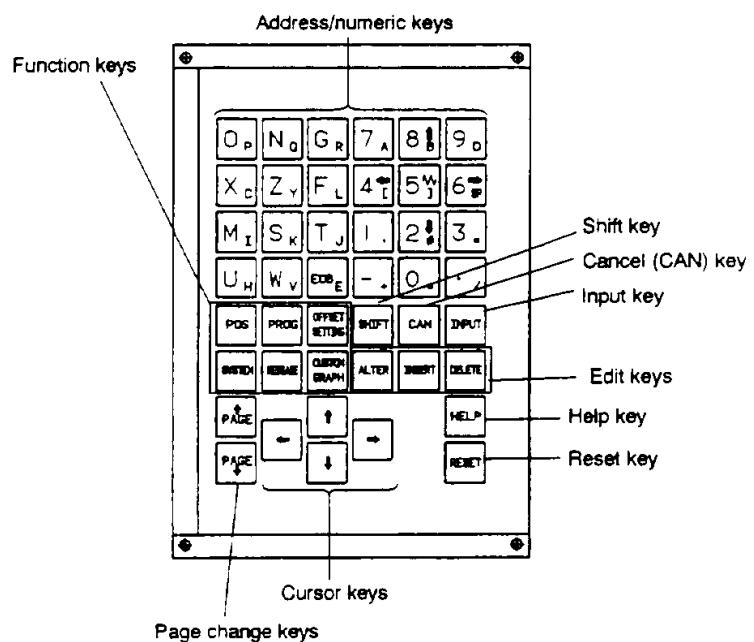
## *2.3*    STAND-ALONE TYPE STANDARD MDI UNIT
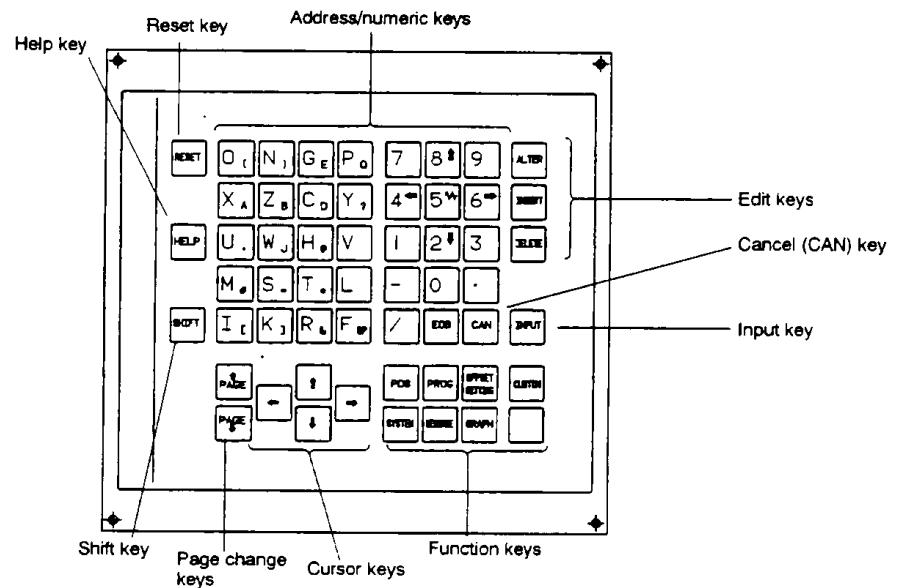
# 3  DESCRIPTION OF COORDINATE

# *3.1*    **MACHINE COORDINATE SYSTEM**
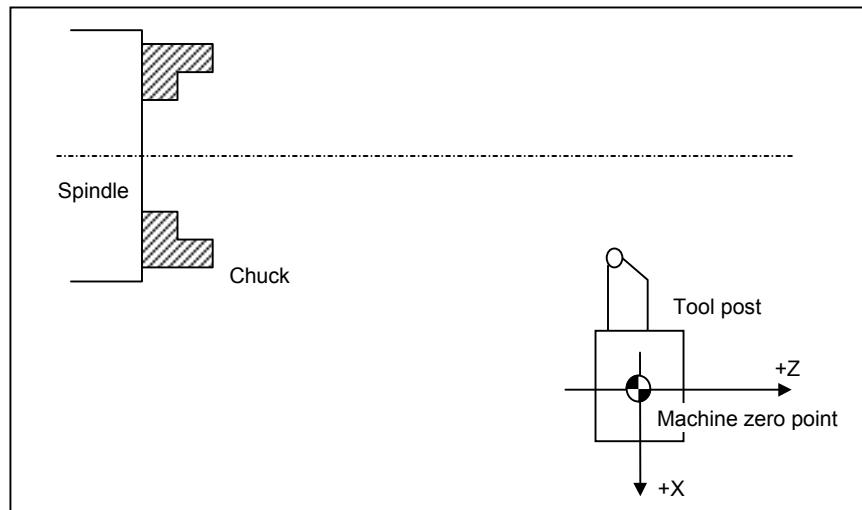


The above figure shows the state in which a tool post has returned to the reference position. The X and Z machine coordinates of the position of the tool post are 0. When the tool post is manually returned to a reference position, the X and Z machine coordinates of the machine reference position are automatically set to 0.

The directions of the X- and Z-axes are specified so that the tool post moves always from a workpiece or the spindle when the tool post moves in the positive X and Z directions.

This machine coordinate system is kept unchanged, regardless of whether the program reference position, explained later, is set on the end face of aworkpiece.

# *3.2*     **WORKPIECE COORDINATE SYSTEM**

The machine coordinate system previously mentioned is peculiar to a specific machine. It is determined uniquely regardless of theworkpieces and tools used.

Another coordinate system is required for each tool to prevent programmed positions from changing between tools because of the difference in tool dimensions. This coordinate system must be set with respect to the dimensions of a specific tool. Such coordinate systems are called Workpiece Coordinate System.

A workpiece coordinate system is determined using the Tool Geometry Offset. For details of it, refer to the CNC Operator's Manual.

The origin of the workpiece coordinate system is set on the end face of workpiece always in MANUAL GUIDE.

In the following sample drawing, $OFS_X$ and $OFS_Z$ means the distance with sigh between 2 points, 1st is the position at the tool post is at the machine zero point, 2nd is the position at the imaginary nose is at the center of the end face of the workpiece (origin of the workpiece coordinate). These distance data are set into the geometry offset amount.
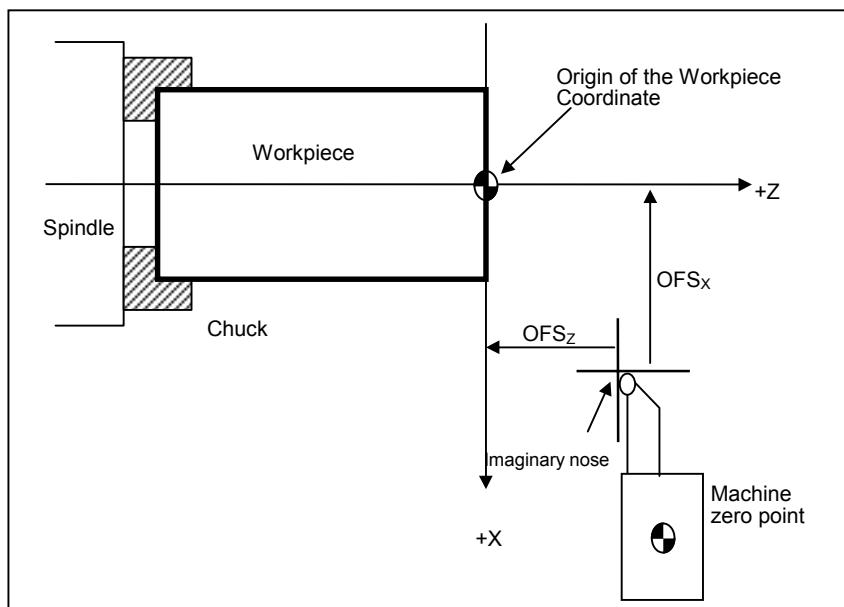
At the time of changing workpiece and its length is changed, instead of measuring the above geometry offset value for all tools, Workpiece Shift offset compensation is used for shifting the workpiece coordinate origin point.

In the following sample drawing, $WSF_Z$ means the distance with sigh between the end face of the old workpiece and the new one. This distance data is set into the Z-workpiece shift offset amount.
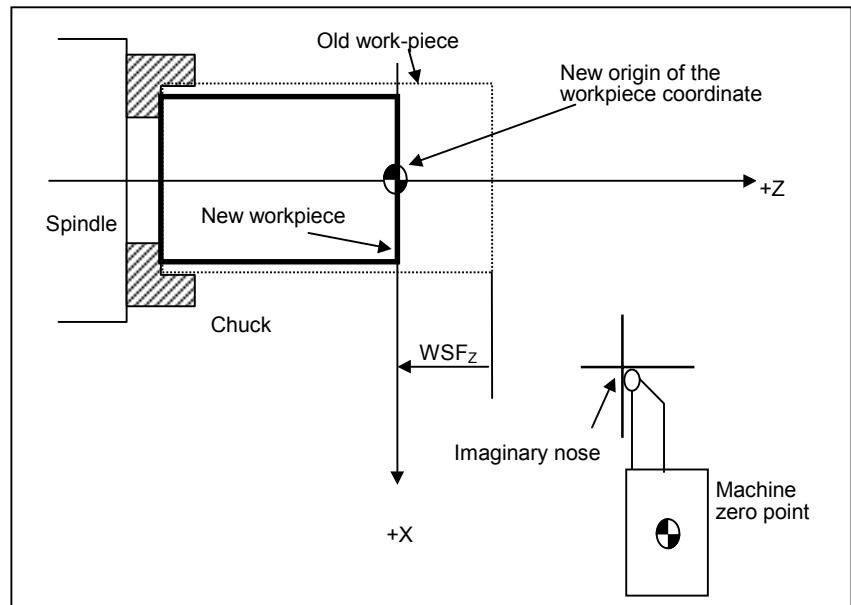


By using the Set-up Guidance, described later in Chapter 9, workpiece shift amount are automatically set.

# *3.3*     PROGRAM COORDINATE SYSTEM

When a machining program is created by MANUAL GUIDE, the dimensions are specified similarly with workpiece coordinate system.

# 4 GUIDANCE CUTTING OPERATIONS

As described in Chapter I.2 GUIDANCE HANDWHEEL, an operator can carry out machining along a tapered or arc figure by using a guidance handwheel. And also, he can enter the motion block into CNC temporary memory, this entering operation is called "Teaching-in". These entered, taught-in, motion blocks can be used repeatedly for cutting the same part during corresponding process is selected.

In order to use these taught-in motion blocks for later occasion, they can be saved into CNC kept memory, called "Playback Machining Program", and this operation is called "Save into Playback Machining Program". By using this playback machining program, an operator can carry out same machining motions repeatedly later on.

# *4.1*     SELECTING THE GUIDANCE CUTTING

By the following operations, an operator can select a Guidance Cutting mode.
1)   Push **RESET** key on a LCD/MDI keyboard.
2)   Push soft-key [GUIDNC].

---

**NOTE**
When the soft-key [GUIDNC] is not displayed on a screen, it will be displayed by pushing left-end softkey called Super Return.
There may be a case that menu keys for selecting machining type are prepared on a machine operator's panel, then refer to the manual prepared by machine tool builder.

---

# *4.2* **INPUTTING PROCESS DATA**

By selecting Guidance Cutting, the following Process Data items are displayed in a Program Window. As to supplemental process data, they will be displayed in a pop-up window, and it can be displayed by pushing soft-key [DETAIL].

## *4.2.1* **Main Process Data**

```
GUIDANCE CUTTING

TOOL NUM   =
FEED/REV.  =
S-SPEED    =
TOOL KIND  =
TOOL DATA  =
```

**TOOL NUM.**    Number of tool used for guidance cutting
**FEED/REV.**    Feedrate at playback machining, and it can be changed to FEED/MIN. by pushing soft-key [FEED/MIN.].
**S- SPEED**    Cutting speed at playback machining, and it can be changed to REV/MIN. by pushing soft-key [REV/MIN.].
**TOOL KIND**    Tool kind used for guidance cutting. Choose from the following soft-key menu.

| | |
|---|---|
| [GENERAL-OUT] | Tool used for outer machining |
| [GENERAL-INN] | Tool used for inner machining |
| [GENERAL-FACE] | Tool used for facing |
| [GROOVE-OUT] | Tool used for outer grooving |
| [GROOVE-INN] | Tool used for inner grooving |
| [GROOVE-FACE] | Tool used for face grooving |
| [DRILL] | Tool used for drilling |

**TOOL DATA**    Tool nose data used for guidance cutting. Choose from the following soft-key menu when one of the GENERAL-OUT,INN or FACE is selected in TOOL KIND input field.

GENRRAL-OUT          GENERAL-INN          GENERAL-FACE
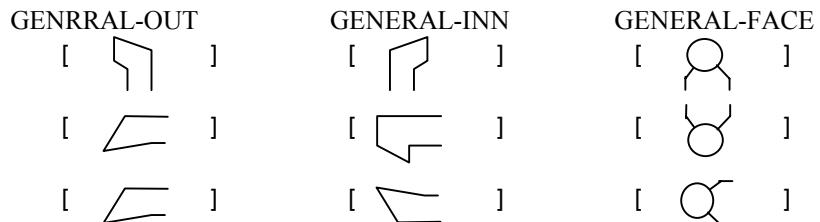
When one of the GROOVE-OUT,INN or FACE is selected in TOOL KIND, tool width must be inputted in TOOL DATA input field.
When the DRILL is selected in TOOL KIND, tool diameter must be inputted in TOOL DATA.

# *4.3*    **TEACH-IN CUTTING MOTIONS**

After inputting necessary process data, a cursor moves into Figure Data Window automatically, and then, an operator can begin guidance cutting and teach-in operation. This status is called 'Teach-in mode'.

> **NOTE**
> The maximum block number can be inputted in teach-in mode of a guidance cutting process is 80.

## *4.3.1*    **Figure Data Window**

As described above, at the time that a cursor is in a figure data window, an operator can carry out a guidance cutting using guidance handwheel, and teach-in these cutting motions into CNC for later playback machining.

In this figure data window, taught-in cutting motion blocks are displayed, and an operator can edit the contents of them.

```
                          TOOL DATA =

Figure          ┌─► T101 ;
Data                G97 S300 ;
Window              M03 ;
                    G0 X200. Z50. ;
                    G0 X100. Z20. ;
                    G01 X90, ;
```
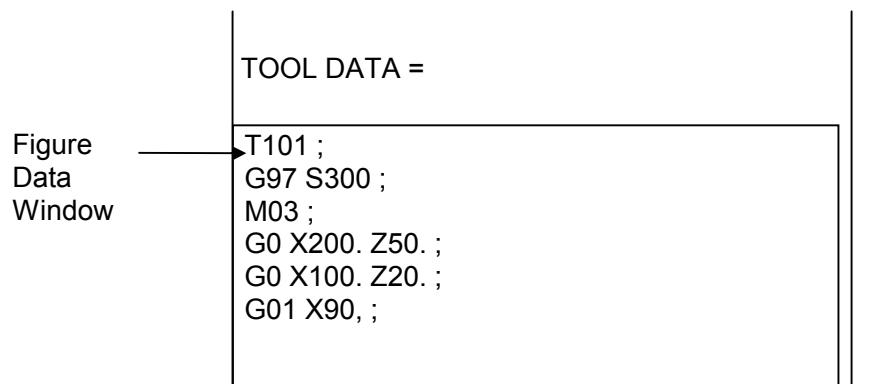
During Teach-in mode, the following soft-keys are displayed and they are used for teach-in operations.

| LINE | CIRCLE ↻ | | | | | TEACH AUX | TEACH RAPID | TEACH CUT | |

## *4.3.2*    **Operation of Teach-in Auxiliary Function Blocks**

On a machine operator's panel, the following auxiliary function switches are prepared.

- CSS (Constant Surface Speed) mode ON
- CSS mode OFF
- FEED/Rev.
- FEED/Min.
- T-code
- Spindle CW
- Spindle CCW
- Spindle Stop
- Coolant ON
- Coolant OFF
- Return to Tool Change Position
- Return to Machine Reference Position

By pushing a necessary switch among these switches, corresponding auxiliary function is carried out.

During Teach-in mode, these carried out auxiliary functions can be taught-in into CNC for later playback machining.

After pushing one or continuously several ones, maximum 10 blocks, of the above auxiliary function switches, by pushing [TEACH AUX], an operator can teach-in these commands into CNC for later playback. When several auxiliary function switches are pushed continuously before pushing [TEACH AUX], all of these commands are taught in CNC at a time.

---

**NOTE**

The above taught-in command blocks can be used for playback machining, but since they are temporary program as it is, these will be eliminated at the time the next machining type is selected. So, in order to use these taught-in command blocks for playback machining at later on, an operator must save them as a playback machining program.

---

## *4.3.3*    **Operation of Teach-in a Rapid Traverse Motion Block**

During Teach-in mode, an operator can move a tool by handwheel or JOG stick on a machine operator's panel.

After moving a tool toward a proper position, by pushing [TEACH RAPID], an operator can teach-in this tool motion into CNC as a rapid traverse block for later playback.

This taught-in block is displayed in a figure data window as following sample.

**G00 X78.010 Z10.053 ;**

Workpiece coordinate of actual position

G-code of rapid traverse motion
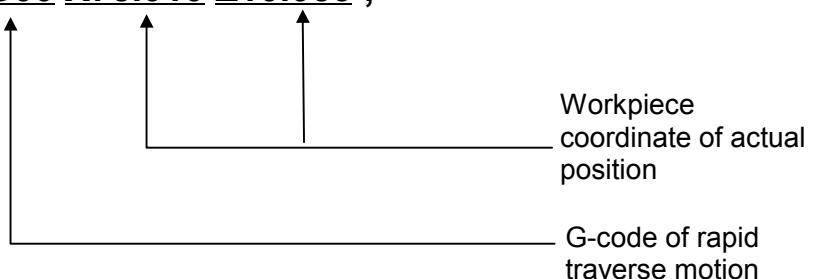
## *4.3.4*    **Operation of Teach-in a Simple Linear Cutting Motion Block**

During Teach-in mode, an operator can cut a workpiece in direction of only X-axis or Z-axis by using a normal handwheel on a machine operator's panel.

After moving a tool toward a proper position, by pushing [TEACH CUT], an operator can teach-in this tool motion into CNC as a linear cutting block for later playback.

This taught-in block is displayed in a figure data window as following sample.

**G01 X78.010 Z10.053 F0.5 ;**

Feedrate inputted in process data window

Workpiece coordinate of actual position

G-code of linear cutting motion
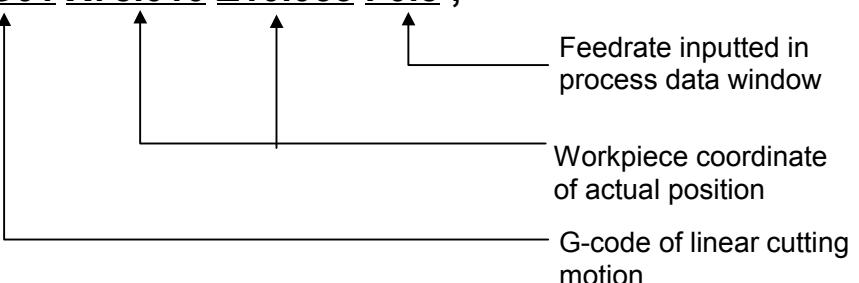
## *4.3.5*    **Operation of Guidance Cutting (Line)**

By using a guidance handwheel on a machine operator's panel, an operator can carry out the following 2 types of linear cutting in simultaneous X and Z-axis direction.

- **Approach Cutting** to an inputted Target Line
- **Along Cutting** to an inputted Target Line

In order to input a target line, the following data input window is displayed in a pop-up window, and it can be called by pushing a [LINE].

```
X1 POS.    =                        (X1,Z1)
Z1 POS.    =
X2 POS.    =
X2 POS.    =
ANGLE      =                              A
                              (X2,Z2)

                                    1.1.1.1.1.1

                                              0
```
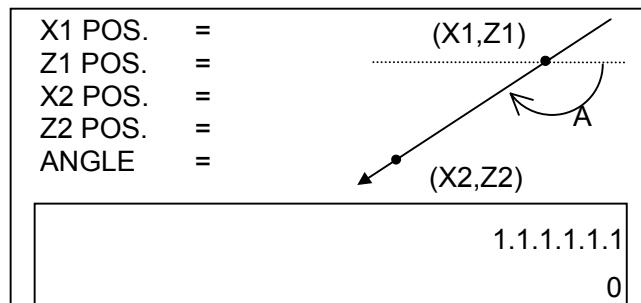
| | |
|---|---|
| **X1/Z1/X2/Z2 POS.** | Absolute coordinate of separated 2 points on a line. |
| **ANGLE** | Angle of line from +Z-axis. A positive angle is clockwise direction from +Z-axis, and minus angle is counter clockwise direction. If this data is inputted, only 1 point coordinate data is necessary among the above 2 points. |

After inputting necessary data in the above window, by pushing [INPUT END], an operator can begin guidance cutting.

---

**NOTE**
   As to the above position data (X1,Z1) and (X2,Z2), instead of inputting numerical data through key-board, an operator can input the actual tool position by pushing [READ POS.].

---

Besides inputting data for target line, the following distance data are displayed in a pop-up window. An operator can move the tool while checking the distance to the target line.

| | |
|---|---|
| **D** | Distance between the current tool position and the target line. |
| **DX/DZ** | Distance between the current tool position and the target line along X or Z-axis. |

> **NOTE**
> At guidance cutting, the Nose-R compensation is done automatically, and the nose radius value of inputted tool number in a process data window is used for it.
> So, the above distance data display the distance between 2 imaginary nose point of the current tool position and the position of that a nose circle is touched to the target line.

## (1) Approach Cutting

By switching the guidance handwheel mode on a machine operator's panel, an operator can select an Approaching Cutting mode.

In an approach cutting mode, a tool moves along a line perpendicular to the target line according as an operator rotates a guidance handwheel.

During approach cutting mode, the following guidance is displayed in a pop-up window.



```
X1 POS.   =     10.000
Z1 POS.   =    -10.000
X2 POS.   =     80.000
Z2 POS.   =  -100.000
ANGLE     =

D         3.679
DX        7.893   DZ     10.148
```

In the guidance displayed in a pop-up window, pink and green arrows indicate the direction in which the tool is moved. The direction in which the handwheel is rotated determines the direction of approach. At the end of hollow arrow, the handwheel and the direction of its rotation are displayed.

The feedrate of the tool motion depends on the speed at which the guidance handwheel is rotated.

## (2) Along Cutting

By switching the guidance handwheel mode on a machine operator's panel, an operator can select an Along Cutting mode.

In an along cutting mode, a tool moves along a line parallel to the target line according as an operator rotates a guidance handwheel.

During along cutting mode, similar to the former approach cutting mode, the guidance for handwheel rotation is displayed in a pop-up window.

In the guidance, blue arrows indicate the direction of along cutting. The direction in which the handwheel is rotated determines the direction of along cutting. At the end of solid arrow, the handwheel and the direction of its rotation are displayed.

The feedrate of the tool motion depends on the speed at which the guidance handwheel is rotated.

## (3) Limit Setting

Normally, a limit is set such that a tool does not exceed the target line. When the target line is inputted, an opposite area of target line from the tool position is set to the limit area.

By pushing a [LMTOFF], this soft-key display is changed to [LMT ON] and limit becomes unavailable. In order to make this limit be valid, push [LMT ON].

During this limit is valid, a black line is displayed at the top of the approaching arrow in a guidance drawing.

## *4.3.6*    **Operation of Guidance Cutting (Circle)**

By using a guidance handwheel on a machine operator's panel, an operator can carry out the following 2  types of circular cutting in simultaneous X and Z-axis direction.

- **Approach Cutting** to an inputted Target Circle
- **Along Cutting** to an inputted Target Circle

In order to input a target circle, the following data input window is displayed in a pop-up window, and it can be called by pushing a [CIRCLE].
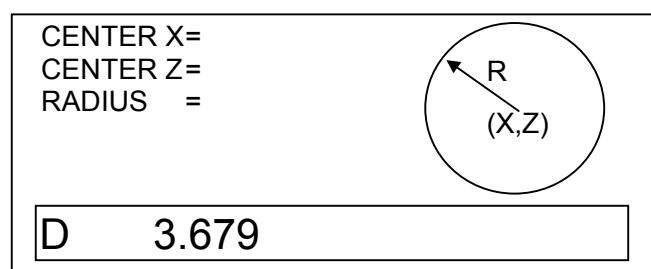


**CENTER X/Z**        Absolute coordinate of circle center.
**RADIUS**             Radius of circle.

After inputting necessary data in the above window, by pushing [INPUT END], an operator can begin guidance cutting.

---

**NOTE**
As to the above circle center position data (X,Z), instead of inputting numerical data through key-board, an operator can input the actual tool position by pushing [READ POS.].

---

Similar to the guidance cutting for line, besides inputting data for target circle, the following distance data are displayed in a pop-up window. An operator can move the tool while checking the distance to the target circle.

D                Distance between the current tool position and the target circle.
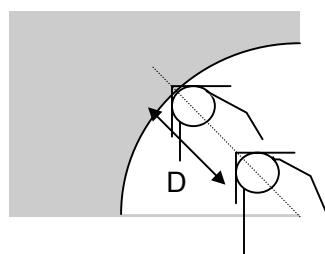
> **NOTE**
> Similar to line cutting, the Nose-R compensation is done automatically, and the nose radius value of inputted tool number in a process data window is used for it.
> So, the above distance data display the distance between 2 imaginary nose point of the current tool position and the position of that a nose circle is touched to the target line.

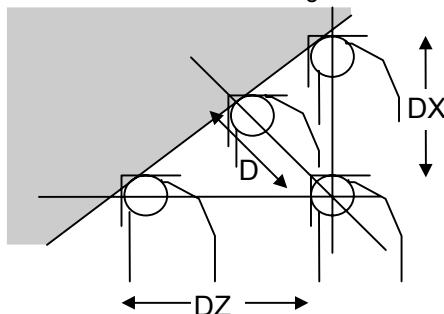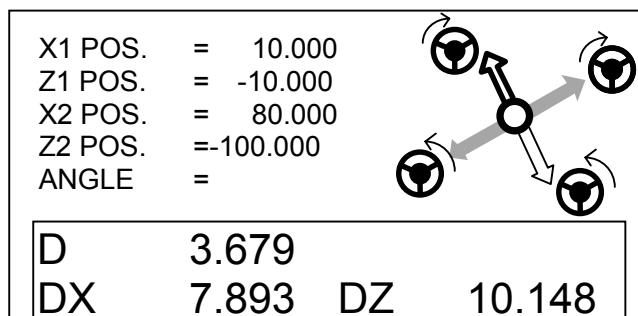After inputting necessary data in the above window, by pushing [INPUT END], an operator can begin guidance cutting. By switching the guidance handwheel mode on a machine operator's panel, an operator can select one of Approaching Cutting or Along Cutting.

## (1) Approach Cutting

By switching the guidance handwheel mode on a machine operator's panel, an operator can select an Approaching Cutting mode.
In an approach cutting mode, a tool moves along a line connecting the center of the tool nose and the center of the target circle according as an operator rotates a guidance handwheel.
During approach cutting mode, the following guidance is displayed in a pop-up window.



```
CENTER X =    50.000
CENTER Z =-100.000
RADIUS   =    50.000


D     6.576
```

In the guidance displayed in a pop-up window, pink and green arrows indicate the direction in which the tool is moved. The direction in which the handwheel is rotated determines the direction of approach. At the end of hollow arrow, the handwheel and the direction of its rotation are displayed.
The feedrate of the tool motion depends on the speed at which the guidance handwheel is rotated.

## (2) Along Cutting

By switching the guidance handwheel mode on a machine operator's panel, an operator can select an Along Cutting mode.
In an along cutting mode, a tool moves along a circle having the same center as the target line according as an operator rotates a guidance handwheel.
During along cutting mode, similar to the former approach cutting mode, the guidance for handwheel rotation is displayed in a pop-up window.

In the guidance, blue arrows indicate the direction of along cutting. The direction in which the handwheel is rotated determines the direction of along cutting. At the end of solid arrow, the handwheel and the direction of its rotation are displayed.

The feedrate of the tool motion depends on the speed at which the guidance handwheel is rotated.
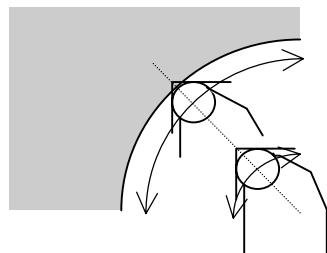


## (3) Limit Setting

Similar to the former line guidance cutting, normally, a limit is set such that a tool does not exceed the target line.

When the target line is inputted, an opposite area of target line from the tool position is set to the limit area.

By pushing a [LMTOFF], this soft-key display is changed to [LMT ON] and limit becomes unavailable. In order to make this limit be valid, push [LMT ON].

During this limit is valid, a black line is displayed at the top of the approaching arrow in a guidance drawing.

⚠ **CAUTION**
Be careful tool nose and workpiece because it interferes, when the following figure is cut by guidance machining,
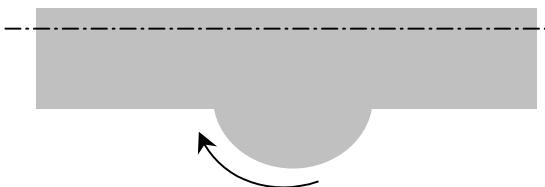
## *4.3.7*  **Operation of Teach-in a Cutting Block**

As described in former description, during Teach-in mode, an operator can cut a workpiece by using a normal handwheel or guidance handwheel.

After cutting a workpiece toward a proper position, by pushing [TEACH CUT], an operator can teach-in this tool motion into CNC as a cutting block for later playback.

This taught-in block is displayed in a figure data window as following sample.

Sample of Linear Cutting)

**G01 X78.010 Z10.053 F0.25 ;**

Feedrate inputted in process data window

Workpiece coordinate of actual position

G-code of linear cutting

Sample of Circular Cutting)

**G02 X78.010 Z10.053 R20.0 F0.25 ;**

Feedrate inputted in process data window

Radius of cutting arc path

Workpiece coordinate of actual position
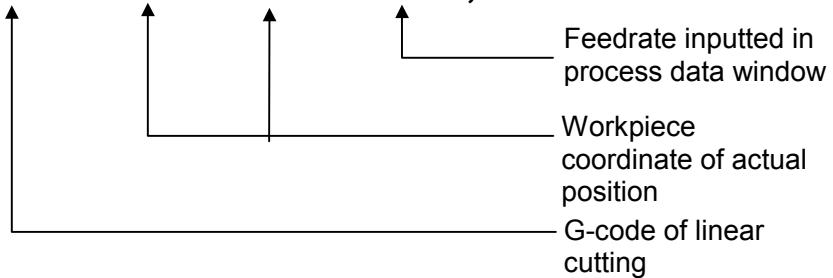
G-code of circular cutting
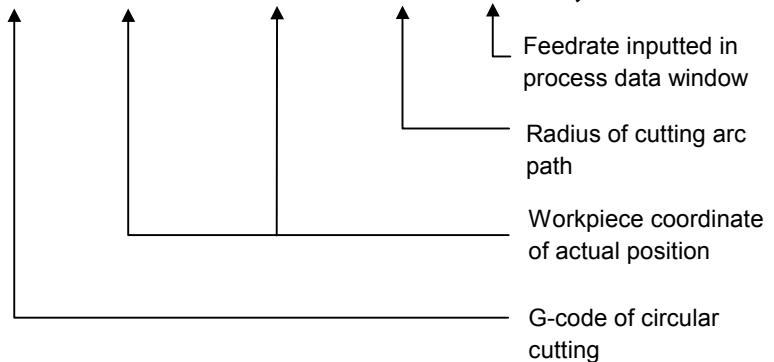
# *4.4*     EDITING TAUGHT-IN BLOCKS

Taught-in "Auxiliary Function Block", "Rapid Traverse Block" and "Cutting Motion Block" can be displayed in a figure data window as described in 4.3. And an operator can edit these blocks too.

## *4.4.1*     Figure Data Window

By using 4-directional cursor keys ↑ ↓ → ← on a LCD/MDI unit, a cursor in a figure data window can be moved for pointing the data word of taught-in blocks.

As described above, at the time that a cursor is in a figure data window, an operator can carry out a guidance cutting using guidance handwheel, and teach-in these cutting motions into CNC for later playback machining.

In this figure data window, taught-in cutting motion blocks are displayed, and an operator can edit the contents of them.

By pushing a cursor key ↑ or ↓, the contents of figure data window are shifted up and down, and an operator can see the whole blocks.

```
                              TOOL KIND   =
                              TOOL DATA   =
Figure Data                  ┌──────────────────────────┐
Window ─────────────►        │ G97 S300 ;               │
                             │ M03 ;                    │
Cursor ─────────────►        │ ▌T101▐       ;           │
                             │ G1 X100. Z20. F0.5 ;     │
                             │ G1 X90, Z25. F0.5 ;      │
                             │                          │
                             │                          │
                             └──────────────────────────┘
```
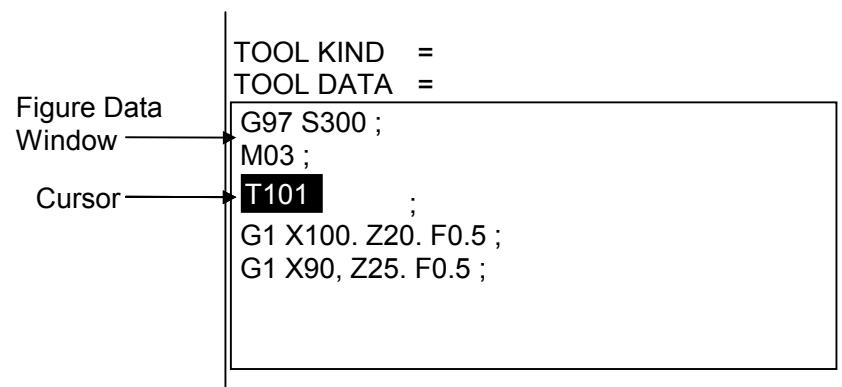
## *4.4.2*    **Operations for Figure Data Editing**

### (1) Alteration of numeric value

After pointing the objective word (=Address + Numeric data) by cursor, input only new numeric data through numeric keys on LCD/MDI unit, and push INPUT key.

By these operations, only numeric value can be altered.

### (2) Alteration of Address and numeric value

After pointing the objective word (=Address + Numeric data) by cursor, input new address and numeric data through address and numeric keys on LCD/MDI unit, and push INPUT key.

By these operations, address and numeric value can be altered.

### (3) Insertion of a new word

After pointing the word after that a new word should be inserted, input new word (=Address + Numeric data) through address and numeric keys , and push INSERT.

By these operations, a new word can be inserted after the pointed word.

### (4) Insertion of a new motion block

After pointing the character ";", end of the motion block after that a new motion block should be inserted, input necessary word address data such as G-code, axis motion word (X_____ Z_____) etc. After inputting whole necessary word address data, lastly, push ";" and INSERT.

By these operations, a new motion block word can be inserted after the pointed word.

### (5) Deletion of a word

After pointing the objective word (=Address + Numeric data), push DELETE.

Prompting message for confirmation of deleting will be displayed, so push DELETE again.

By these operations, this objective word can be deleted.

### (6) Deletion of a motion block

After pointing some word (=Address + Numeric data) in the block which must be deleted, push "EOB" and **DELETE**. Prompting message for confirmation of deleting will be displayed, so push **DELETE** again.

By these operations, this objective motion block can be deleted.

# *4.5*     **CHECKING CUTTING MOTIONS ON A GRAPHIC WINDOW**

Taught-in "Rapid Traverse Block" and "Cutting Motion Block" can be displayed graphically in a graphic window.

## *4.5.1*     **Graphic Window**

In a graphic window, drawing of taught-in motion blocks are displayed in line form.
Cutting motion blocks are displayed by solid lines, and rapid traverse blocks are displayed by dotted line.
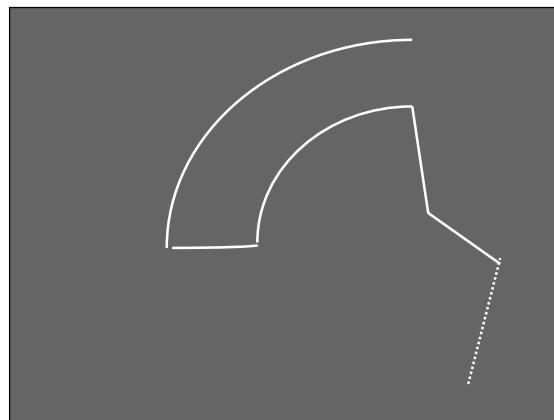


> **NOTE**
> The 1st block, since its starting point is not clear, is not displayed as a line drawing.

# *4.6*      PLAYBACK MACHINING

Taught-in "Auxiliary Block", "Rapid Traverse Block" and "Cutting Motion Block" can be executed in the form among of the following 2 types machining.

- One-shot playback machining
- Whole processes playback machining

## *4.6.1*      One-shot Playback Machining

During a guidance cutting process is selected, in other word, its teach-in blocks are displayed in a figure data window, an operator can execute an actual machining by using taught-in blocks in its process, and it is called One-shot Playback Machining.

- Operations for one-shot playback machining
    1) By using cursor key ↑ ↓, point the block from which playback machining is started. And as another method, pushing [HEAD] and pointing the cursor to the top of taught-in blocks, it can be started from the top of the process.
    2) By pushing [EXEC], playback machining of the taught-in blocks in its process can be started.

---

**NOTE**

Taught-in blocks by the operations described in this chapter are temporary one, and as it is, they will be canceled at the time next process is selected. If an operator wants to use these taught-in blocks for later playback machining, it is necessary to save them as a playback machining program.
As to the operations for this saving, refer to next chapter.

---

## *4.6.2*    **Whole Processes Playback Machining**

In order to use taught-in blocks for whole processes playback machining, it is necessary to save them into CNC memory as a playback machining program.

- Operations for saving as a new playback program
    1) Push [SAVE], and a Program List is displayed in a pop-up window.

Cursor

```
PROGRAM LIST
01  CIRCLE MACHINING 5M12S  1997/05/12 10:12
10:LINE MACHINING            4M32S  1997/06/10 11:21
```

Program

Program Name (Max 20ch.)

Processing time (Max 99minuites)

Lastly modified date and time

- 2) Input 2 digits number for program number, and push [NEW]. After then, a new program including current taught-in blocks is made.
- 3) Move a cursor onto the program name field, and Input program name through key-board, and push **INPUT**.   The maximum character number of it is 20.

- Operations for adding to an old playback program
    1) Push [SAVE], and a Program List is displayed in a pop-up window.
    2) Select the program into which current taught-in blocks are added by positioning a cursor to its program number.
    3) Push [ADD], and after then, the current taught-in blocks are added the end of the selected playback program.

## *4.7*    ONE-SHOT MACHINING SIMULATION OF A INPUTTING PROCESS

Taught-in "Auxiliary Block", "Rapid Traverse Block" and "Cutting Motion Block", can be checked by an one-shot playback machining simulation. And this machining simulation is done with tool path drawing only.

By pushing a [CHECK], it can be displayed by pushing a right-end soft-key [ + ], the following soft-keys are displayed.

| | | | | | HEAD | RETURN | EXEC | SINGLE |
|---|---|---|---|---|---|---|---|---|

| | |
|---|---|
| **[HEAD]** | Return to the top of this process. |
| **[RETURN]** | End a check drawing and return to the programming screen. |
| **[EXEC]** | Start the continuous simulation drawing. |
| **[SINGLE]** | Cause a single block stop while continuous simulation is being executed. Or, starts single block simulation. |

In order to return to the initial screen, push **RESET**, and then push [RETURN].

# 5 SINGLE CUTTING OPERATIONS

As described in Chapter I.2 GUIDANCE HANDWHEEL, an operator can enter cutting motions through key-board directly and control tool motions by using a guidance handwheel as a synchronous feed mode. And, these entered cutting motions are memorized into CNC temporary memory, and can be used repeatedly for cutting the same part during corresponding process is selected.

In order to use these entered cutting figures for later occasion, they can be saved into CNC kept memory, called "Playback Machining Program", and this operation is called "Save into Playback Machining Program". By using this playback machining program, an operator can carry out same machining motions repeatedly later on.

# *5.1*    SELECTING THE SINGLE CUTTING

By the following operations, an operator can select a Single Cutting mode.
1)    Push **RESET** key on a LCD/MDI keyboard.
2)    Push soft-key [SINGLE].

> **NOTE**
> When the soft-key [SINGLE] is not displayed on a screen, it will be displayed by pushing left-end softkey called Super Return.
> There may be a case that menu keys for selecting machining type are prepared on a machine operator's panel, then refer to the manual prepared by machine tool builder.

# *5.2* INPUTTING PROCESS DATA

By selecting Single Cutting, the following Process Data items are displayed in a Program Window.

## *5.2.1* Main Process Data

```
SINGLE CUTTING

TOOL NUM.  =
FEED/REV.  =
S-SPEED    =
TOOL KIND  =
TOOL DATA  =
```

**TOOL NUM**        Number of tool used for single cutting

**FEED/REV.**       Feedrate at playback machining, and it can be changed to FEED/MIN. by pushing soft-key [FEED/MIN.].

**S-SPEED**         Cutting speed at playback machining, and it can be changed to REV/MIN. by pushing soft-key [REV/MIN.].

**TOOL KIND**       Tool kind used for guidance. Into details, refer to the Chapter 4.2.

**TOOL DATA**       Tool nose data for single cutting. Into details, refer to the Chapter 4.2.

# *5.3*   **ENTERING CUTTING MOTIONS**

After inputting necessary process data, a cursor moves into Figure Data Window automatically, and then, an operator can enter the cutting motions directly through key-board, and carry out each cutting motion by using a guidance handwheel as a synchronous feed mode. And at this time, teaching of motion blocks done by using normal handwheel and auxiliary function blocks done by using switches on a machine operator's panel are possible too.

---
**NOTE**
The maximum block number can be entered in a single cutting process is 80.
---

## *5.3.1*   **Figure Data Window**

As described above, at the time that a cursor is in a figure data window, an operator can carry out cutting of entered motion blocks by using a guidance handwheel.
These entered motion blocks are memorized into CNC temporary memory, and can be used for playback machining.
Besides, similar to the guidance cutting of former chapter, an operator can teach-in motion blocks by normal handwheel and auxiliary function. This status is called "Teach-in mode", similar to guidance cutting.
In this figure data window, entered and taught-in cutting motion blocks are displayed, and an operator can edit the contents of them.

```
CUT SPEED =

T101 ;
G97 S300 ;
M03 ;
G0 X200. Z50. ;
G0 X100. Z20. ;
G1 X90, ;
```

Figure Data Window →
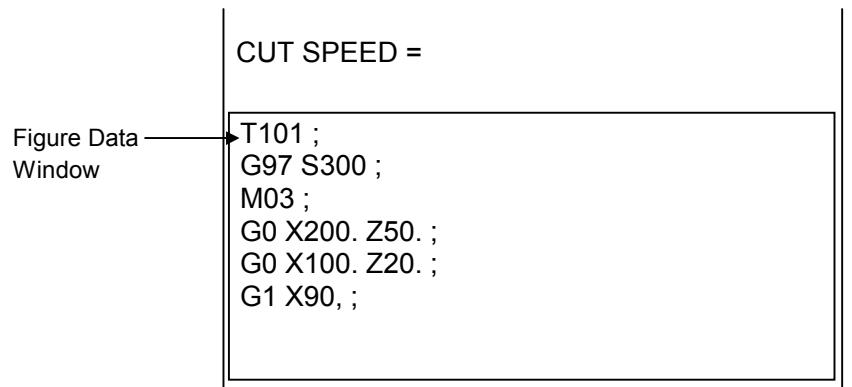
During Teach-in mode, the following soft-keys are displayed and they are used for entering a new motion block and teach-in operations.

| POS. | LINE | CW | CCW |  |  | TEACH AUX. | TEACH RAPID | TEACH CUT |  |

## *5.3.2*    **Operation of Teach-in Auxiliary Function Blocks**

Similar to a guidance cutting process, the following auxiliary function switches on a machine operator's panel can be used. And, these auxiliary functions can be taught-in into CNC by [TEACH AUX].

- CSS (Constant Surface Speed) mode ON
- CSS mode OFF
- FEED/Rev.
- FEED/Min.
- T-code
- Spindle CW
- Spindle CCW
- Spindle Stop
- Coolant ON
- Coolant OFF
- Return to Tool Change Position
- Return to Machine Reference Position

## *5.3.3*    **Operation of Teach-in a Positioning Block**

Similar to a guidance cutting process, tool positioning motions done by normal handwheel or JOG stick can be taught-in by [TEACH RAPID].

This taught-in block is displayed in a figure data window as following sample.

**G00 X78.010 Z10.053 ;**

Workpiece coordinate of actual position

G-code of rapid traverse motion (same with cutting)
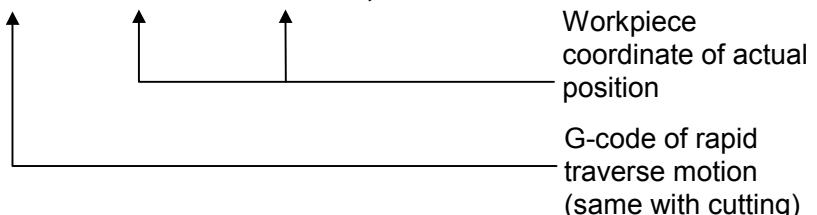
## *5.3.4*    **Operation of Teach-in a Simple Linear Cutting Motion Block**

Similar to a guidance cutting process, an operator can cut a workpiece in direction of only X-axis or Z-axis by using a normal handwheel, and these cutting motions can be taught-in by [TEACH CUT].

This taught-in block is displayed in a figure data window as following sample.

**G01 X78.010 Z10.053 F0.5 ;**

Feedrate inputted in a process data window

Workpiece coordinate of actual position

G-code of linear cutting motion
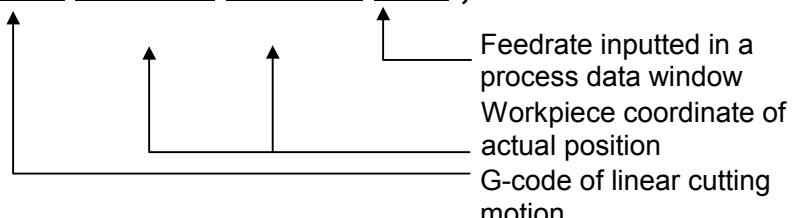
## *5.3.5*    **Operation of Entering a Tool Path (Positioning/Line/Circle)**

After positioning a cursor to the end of taught-in blocks in a figure data window, an operator can input a tool path data with its end point calculating.

When the coordinate of the figure end point is unknown, it can be calculated and the figure can be entered.

**Positioning Motion**

By pushing a [POS.], the following figure inputting window is displayed in a pop-up window.

```
END POINT X    X=
END POINT Z    Z=
                              (X,Z)
```
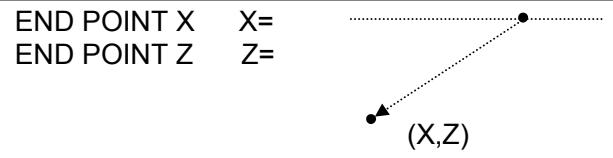
**END POINT X/Z**       Absolute coordinate of end point

> **NOTE**
> As to the above position data (X,Z), instead of inputting numerical data through key-board, an operator can input the actual tool position by pushing [READ POS.].

**Linear Cutting Motion**

By pushing a [LINE], the following figure inputting window is displayed in a pop-up window.

```
END POINT X    X=
END POINT Z    Z=
ANGLE          A=            A
FEEDRATE       F=
                     (X,Z)
```
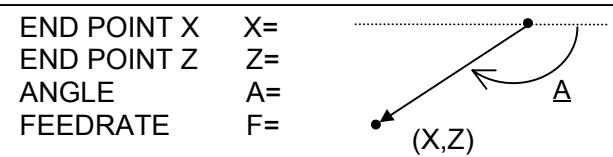
**END POINT X/Z**       Absolute coordinate of end point
**ANGLE**               Angle of line from +Z-axis. A positive angle is clockwise direction from +Z-axis, and minus angle is counter clockwise direction. If this angle data is inputted, end point either X or Z need not be entered, and not inputted data can be calculated.
**FEEDRATE**            Feedrate of this cutting motion (data inputted in process data is set as a default value)

> **NOTE**
> As to the above position data (X,Z), instead of inputting numerical data through key-board, an operator can input the actual tool position by pushing [READ POS.].

## Circular Cutting Motion

By pushing a [CW] / [CCW], the following figure inputting window is displayed in a pop-up window.



| | |
|---|---|
| END POINT X | X= |
| END POINT Z | Z= |
| RADIUS | R= |
| CENTER X | I= |
| CENTER Z | K= |
| FEEDRATE | F= |

**END POINT X/Z**    Absolute coordinate of end point

**RADIUS**    Radius of this circle

**CENTER X/Z**    Absolute coordinate of center point of this circle. If these center data (I/K) are inputted, end point either X or Z need not be entered, and not inputted data can be calculated.

If end point (X/Z) and center point (I/K) are inputted, radius data need not be inputted.

**FEEDRATE**    Feedrate of this cutting motion (data inputted in process data is set as a default value)

> **NOTE**
> As to the above position data (X,Z), instead of inputting numerical data through key-board, an operator can input the actual tool position by pushing [READ POS.].

## *5.3.6*   **Displaying of Entered Tool Motion Blocks**

As described in former description, an operator can input a figure data for each tool motion blocks.

After inputting necessary figure data for each tool motions blocks, by pushing **INSERT**, calculation of end points coordinate is done, and the result of it is displayed in a figure data window as following sample.

Sample of Positioning)

**G00 X78.010 Z10.053 ;**

Workpiece coordinate of end point

G-code of Positioning

Sample of Linear Cutting)

**G01 X78.010 Z10.053 F0.25 ;**

Inputted feedrate

Workpiece coordinate of end point

G-code of linear cutting

Sample of Circular Cutting)

**G02 X78.010 Z10.053 R20.0 F0.25 ;**

Inputted feedrate

Radius of circle

Workpiece coordinate of end point
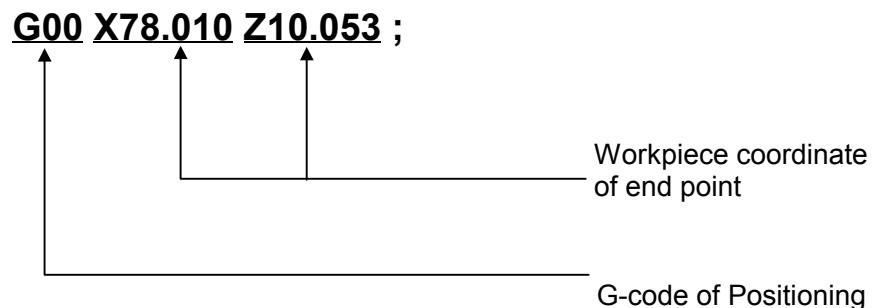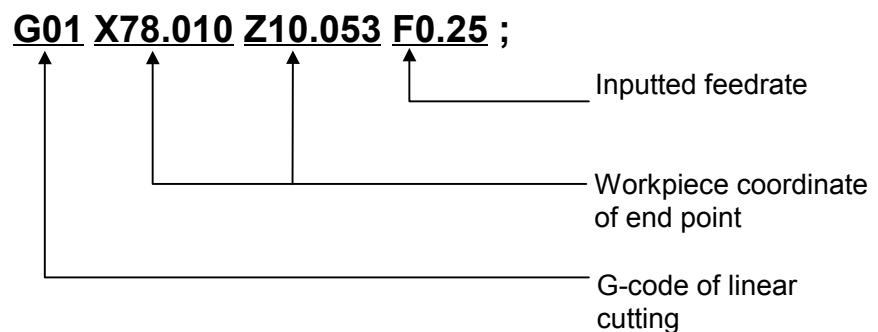
G-code of circular cutting
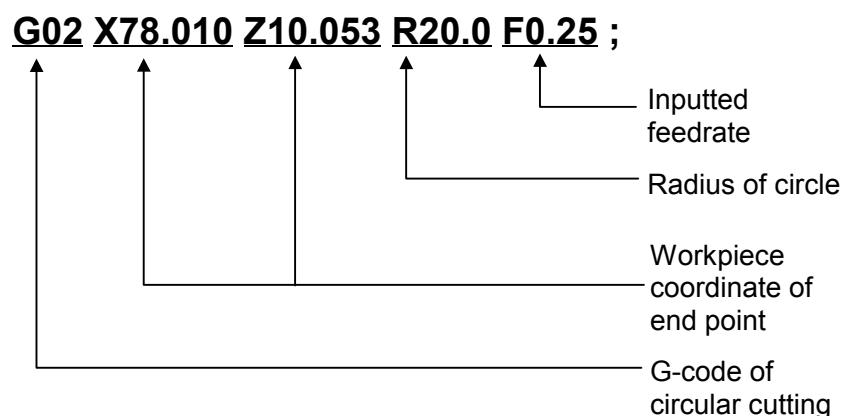
# *5.4*    **EDITING TAUGHT-IN OR ENTERED BLOCKS**

Taught-in or entered "Auxiliary Function Block", "Positioning Block" and "Cutting Motion Block" can be displayed in a figure data window as described in 5.3. And an operator can edit these blocks too.

## *5.4.1*    **Figure Data Window**

By using 4-directional cursor keys ↑ ↓ → ← on a LCD/MDI unit, a cursor in a figure data window can be moved for pointing the data word of taught-in blocks.

By pushing a cursor key ↑ or ↓, the contents of figure data window are shifted up and down, and an operator can see the whole blocks.

```
CUT SPEED  =

G97 S300 ;
T101        ;
G1 X100. Z20. F0.5 ;
G1 X90, Z25. F0.5 ;
```

Figure Data Window
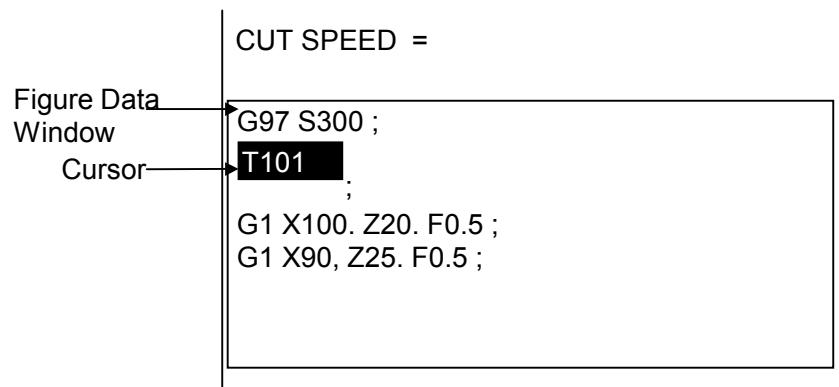
Cursor

## *5.4.2* **Operations for Figure Data Editing**

### (1) Alteration of numeric value

After pointing the objective word (=Address + Numeric data) by cursor, input only new numeric data through numeric keys on LCD/MDI unit, and push INPUT key.

By these operations, only numeric value can be altered.

### (2) Alteration of Address and numeric value

After pointing the objective word (=Address + Numeric data) by cursor, input new address and numeric data through address and numeric keys on LCD/MDI unit, and push INPUT key.

By these operations, address and numeric value can be altered.

### (3) Insertion of a new word

After pointing the word after that a new word is inserted, input new word (=Address + Numeric data) through address and numeric keys, and push **INSERT**.

By these operations, a new word can be inserted after the pointed word.

### (4) Insertion of a new motion block

After pointing the character ";", end of the motion block after that a new motion block should be inserted, select a figure type by pushing a corresponding soft-key menu. By this operation, a figure data inputting window, pop-up window, is displayed on a screen.

After inputting whole necessary figure data, push **INSERT**.

By these operations, a new motion block can be inserted.

### (5) Deletion of a word

After pointing the objective word (=Address + Numeric data), push DELETE.

By these operations, this objective word can be deleted.

### (6) Deletion of a motion block

After pointing some word (=Address + Numeric data) in the block which must be deleted, push ";" and **DELETE**.

By these operations, this objective motion block can be deleted.

> **NOTE**
> As to inputted data such as angle and center point, they are used for calculating end point, they cannot be changed after calculating.

# *5.5*    CHECKING CUTTING MOTIONS ON A GRAPHIC WINDOW

Entered and taught-in tool motion block can be displayed graphically in a graphic window. It is similar to the case of Guidance Cutting. Into details, refer to the Chapter 4.

# *5.6*     **PLAYBACK MACHINING**

Entered and taught-in tool motion blocks and auxiliary function blocks can be executed in the form among of the following 2 types machining.

- One-shot playback machining
- Whole processes playback machining

These are similar to the case of Guidance Cutting. Into details, refer to the Chapter 4.

> **NOTE**
>     In single cutting motions, as described in this
>     chapter, inputting motion blocks are the path of a
>     tool, so Nose-R compensation is never be done.

## *5.7*    ONE-SHOT MACHINING SIMULATION OF A INPUTTING PROCESS

Entered and taught-in motion blocks can be checked by an one-shot playback machining simulation. And this is similar to the case of Guidance Cutting. Into details, refer to the Chapter 4.

# 6

# CYCLE CUTTING OPERATIONS

In cycle cutting, all of the motions, tool changing, approaching, auxiliary functions and cutting and so on, are automatically done.
Following cutting types are available.
1)   Bar machining (Roughing)
2)   Bar machining (Finishing)
3)   Threading
4)   Grooving (Standard)
5)   Grooving (Trapezoidal Roughing)
6)   Grooving (Trapezoidal Finishing)
7)   Center drilling
8)   Drilling
9)   Reaming
10)  Boring
11)  Tapping
As to the details of each cycle cutting type, refer to Part III "Types of Cycle Motions". In this chapter, general operations for cycle cutting are described.

# *6.1*     SELECTING THE CYCLE CUTTING

By the following operations, an operator can select a necessary cycle cutting type.
1)     Push **RESET** key on a LCD/MDI keyboard.
2)     Push soft-key [CYCLE].
3)     Push soft-key for each cutting cycle type.

| BAR ROUGH | BAR FINISH | THREAD | GROOVE | GROOVE TRAPE. ROUGH | | GROOVE TRAPE. FINISH | CENTER | DRILL | REAM | BORE |
|---|---|---|---|---|---|---|---|---|---|---|

By pushing a [ + ], the 2nd menu as follows is displayed.

| TAP | | | | | | * | * | * | * | * |
|---|---|---|---|---|---|---|---|---|---|---|

> **NOTE**
> There may be a case that some particular machining function are added by MTB, and at this time, menu for these functions are displayed in the above [ * ] soft-keys.
> Into details, refer to the MTB's manual.

# *6.2* INPUTTING PROCESS DATA

By selecting each cycle cutting type, necessary process data items are displayed in a Program Window. As to supplemental process data, they will be displayed in a pop-up window, and it can be displayed by pushing soft-key [DETAIL].

## *6.2.1* Sample of Process Data

Example) Bar machining (Roughing)

```
BAR MACHINING

AREA        =       CUT ST Z    =
TOOL NUM    =       FIN X       =
FEED/REV.   =       FIN Z       =
S-SPD/MIN   =       INFEED AM   =
S-DIRECT    =
CUT ST X    =
```

| | |
|---|---|
| **AREA** | Select from [OUTER]/[INNER]/[FACE]/[OUTER REVERS] /[INNER/REVERS]/[FACE REVERS] |
| **TOOL NUM.** | Number of tool used for cycle cutting |
| **FEED/REV.** | Feedrate at playback machining, and it can be changed to FEED/MIN. by pushing soft-key [FEED/MIN.]. |
| **S-SPD/MIN** | Cutting speed at playback machining, and it can be changed to REV/MIN. by pushing soft-key [REV/MIN.]. |
| **S-DIRECT.** | Spindle rotating direction. |
| **CUT ST X/Z** | Coordinate of cutting start position. These are automatically set by inputting figure data. |
| **FIN X/Z** | X/Z direction amount of finishing. |
| **INFEED AM** | Infeed amount of the first cutting motion. |

## *6.2.2*    **Supplemental Process Data**

```
EDGE ANGL =         COOLANT    =
NOSE ANGL =         SPDL GEAR=
RESE.AMNT =
RELESE     =
APPROACHX=
APPROACHZ=
```

By pushing a [DETAIL], a detail data window is displayed.
These supplemental process data are automatically set always, so in ordinary case, an operator need not to call this detail data window excepting the case of checking these data. When more detailed control for machining motions is necessary, check and change these data if necessary.

| | |
|---|---|
| **EDGE ANGL** | Angle of the cutting edge. At the case that it was set to the value over than 90 degrees, cutting edge compensation will be done automatically. Into details, refer to III.1 BAR MACHINING descriptions. Parameter No.9809 value is automatically set into this data at initially inputting this process. |
| **NOSE ANGLE** | Angle of the tool tip, its default value can be set by parameter. Parameter No.9810 value is automatically set into this data at initially inputting this process. |
| **RESE.AMNT** | Travel amount in X-axis(OUTER/INNER : diameter) or Z-axis(FACE : radius) for retraction after cutting. |
| **RELEASE** | Type of tool positioning after cutting. Default value is [STNDRD]. |

        [STNDRD]    The tool is retracted after cutting along the inputted figure.

        [RAPID]    The tool is retracted immediately after cutting.

| | |
|---|---|
| **APROCH X/Z** | X/Z coordinate of approaching intermediate point. |
| **COOLANT** | Coolant specification at playback. Choose from the following soft-key menu. Default value is [ON]. |

        [ON]        : (M8)

        [OFF]      : (M9)

| | |
|---|---|
| **SPDL GEAR** | Spindle gear at playback. Choose from the following soft-key menu. Default value is [AUTO]. |

        [AUTO]

        [LOW]

        [MID1]

        [MID2]

        [HIGH]

## *6.3*   INPUTTING FIGURE DATA (OTHER THAN BAR MACHINING)

After inputting necessary process data, a cursor moves into Figure Data Window, and then, an operator can enter figure data through key-board.

For figure data used in cycle cutting except for Bar machining, specify the necessary figure data such as start point or end point that are required for cycle cutting.

Use the numeric keys and the INPUT key to enter data.

Example) Figure data for Threading

```
                            CUT ST Z      =

Figure  ──────▶   START        X =
Data              START        Z =
Window            END          X =
                  END          Z =
```
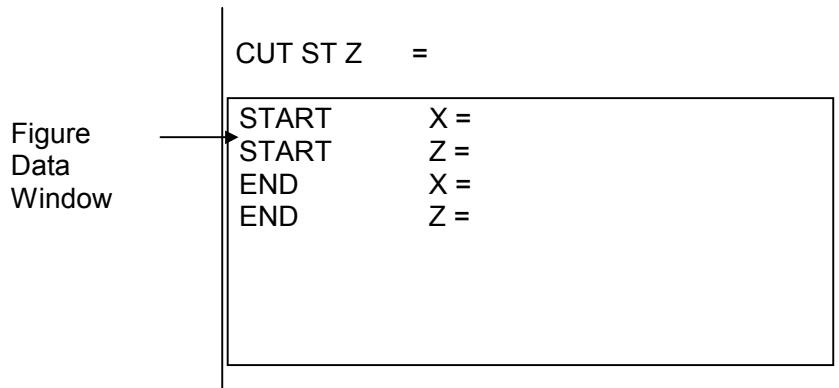
| | |
|---|---|
| **START X/Z** | X or Z coordinate of thread start point. |
| **END X/Z** | X or Z coordinate of thread end point. |

# *6.4*     INPUTTING CONTOUR FIGURE DATA (BAR MACHINING)

For the figure data used in Bar machining, specify the final contour figure.

When the coordinate of the figure end point is unknown, it can be calculated by inputting necessary data described on a blue print.

## (1) Inputting Start Point

At the top of contour figure, enter Start Point figure data.
Figure data window)

```
┌─────────────────────────────────────┐
│      X ████████ Z            ;       │
│                                      │
│                                      │
│                                      │
└─────────────────────────────────────┘
```

**START X/Z**          Absolute coordinate of figure start point

## (2) Selecting a figure type

A menu of figure type appears in soft-keys. Select a figure type by pushing the corresponding soft-key.

| LINE | CW | CCW | ROUND | CHAMF | TANGNT | | | | CALC. |
|------|----|----|-------|-------|--------|--|--|--|-------|

> **NOTE**
> As to Start Point figure data, different from other figure data such as line or circle whose data are inputted in a pop-up window, X and Z data are inputted in a figure data window directly.

## (3) Inputting contour figure data

When a figure has been selected, a pop-up window for inputting contour figure data appears. Enter the data that is designated on the blue print.

Example) Contour figure data for Line

```
CONTOUR FORM (LINE)
 END POINT X    X=
 END POINT Z    Z=
 ANGLE          A=
 OVERRIDE       P=
```

(X,Z)   A

Soft-keys for figure data inputting pop-up window)

| | | | | CAN | | | | | |
|--|--|--|--|-----|--|--|--|--|--|

By pushing a [CAN], an operator can cancel the inputting of the current figure data, and can return to figure type selecting screen.

After inputting whole necessary contour figure data, by pushing a INSERT, a pop-up window for contour figure data is closed and inputted data are displayed in a figure data window as following example.

Example) Figure data window

```
        X    20.000 Z     0.000;
        X           Z         A    30.000 P        ;
```
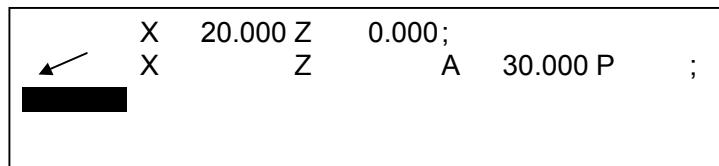
> **NOTE**
>
> When **INSERT** has been pushed, necessary contour calculation is automatically done and end point coordinate of the corresponding figure is memorized internally and is displayed in a graphic window.

# *6.5*     EDITING CONTOUR FIGURE DATA (BAR MACHINING)

After doing necessary editing operations following after each descriptions, by pushing a [CALC.], contour calculation for new contour figures will be done.

## *6.5.1*     Changing Inputted Contour Figure Data

(1)  Move a cursor on the data to be changed in a figure data window.
(2)  At the time only numeric value to be changed, input only new numeric data, and push INPUT.
(3)  At the time both address and numeric value to be changed, input new address and numeric data, and push INPUT.

## *6.5.2*     Inserting a new Contour Figure Data

If the inputted contour figure data were not enough for contour calculating, a warning message for it will be displayed and necessary data must be newly added.

(1)  Move a cursor on a contour figure data other than " ; " in a block that the data should be inserted newly in a figure data window.
(2)  Input a new "Address + Numeric data" and push **INSERT**.

> **NOTE**
> However, newly inserted contour figure data is limited to the one that is displayed in a pop-up window for contour figure data inputting.
> If the other data is inputted, that inputting will be neglected.

## *6.5.3*     Deleting a needless Contour Figure Data

If a needless contour figure data for calculating, it must be deleted.
(1)  Move a cursor on a contour figure data to be deleted.
(2)  Push a **DELETE**.
(3)  The message for confirming the operation.
     To cancel the operation, push a **CAN**. To delete a data, push **DELETE** again.

## *6.5.4*     Inserting a New Contour Figure

(1)  Move a cursor on the " ; " of the figure block to be followed by a new figure.
(2)  Select the figure type of a newly inserted figure.
(3)  A pop-up window for contour figure data inputting of the new figure is displayed.
(4)  Input necessary contour figure data.
(5)  Push **INSERT**.

## 6.5.5    Deleting a Contour Figure

(1) Move a cursor on the " ; " or the left end address, figure type, of the figure block to be deleted.
(2) Push **DELETE**.
(3) The message for confirming the operation and the soft-keys similar to 6.5.3 are displayed. To cancel the operation, push **CAN**. To delete the figure, push **DELETE** again.

## 6.5.6    Changing a Figure

In order to change a figure type of the already inputted contour figure, after deleting the old figure block, insert a new figure block.

# *6.6*  RECALL OF FIGURE DATA FOR ROUGHING AND FINISHING

On Bar machining process or Trapezoidal Grooving process, entered figure of Roughing or Finishing process can be recalled for next process.

When a cursor is positioned on a "Area" item, with keeping entered figure data, Roughing and Finishing can be changed as follows.

- Bar machining process
  **[BAR-F]**   Bar Roughing -> Bar Finishing
  **[BAR-R]**   Bar Finishing -> Bar Roughing

- Trapezoidal grooving
  **[GV-TRR]**   Trapezoidal Grooving Roughing -> Trapezoidal Grooving Finishing
  **[GV-TRF]**   Trapezoidal Grooving Finishing -> Trapezoidal Grooving Roughing

In the above case, in addition to the figure data, some data used for both roughing and finishing commonly are kept at the same time.

# *6.7*      **CHECKING CUTTING MOTIONS ON A GRAPHIC WINDOW**

Contour figure (Bar machining) can be displayed graphically in a graphic window.
It is similar to the case of Guidance Cutting.
Into details, refer to the Chapter 4.

# *6.8*    **PLAYBACK MACHINING**

Entered cycle motions can be executed in the form among of the following 2 types machining.
- One-shot playback machining
- Whole processes playback machining

These are similar to the case of Guidance Cutting. Into details, refer to the Chapter 4.

# *6.9*    ONE-SHOT MACHINING SIMULATION OF A INPUTTING PROCESS

Entered cycle motion program can be checked by an one-shot playback machining simulation. And this is similar to the case of Guidance Cutting. Into details, refer to the Chapter 4.

# 7

# EDITTING OF PLAYBACK MACHINING PROGRAMS

Besides editing inputted data or taught-in program at newly making a new machining process, it is possible to edit a playback machining program that was already saved.

# *7.1*     **CHANGING PROGRAM NAME**

    1)    Push **RESET** key on a LCD/MDI keyboard.

    2)    Push [PROGRM] on an initial screen, then program list window appears.

Cursor

| PROGRAM LIST | | | |
|---|---|---|---|
| 0 :CIRCLE MACHINING | 5M12S | 1997/05/12 10:12 |
| 10:LINE MACHINING | 4M32S | 1997/06/10 11:21 |

Program Number

Program Name
(Max 20ch.)

Processing time
(Max 99minuites)

Lastly modified
date and time

    3)    For changing a program name, move a cursor onto the program name field, and input program name through key-board, and push **INPUT**. The maximum character number of it is 20.

# *7.2* **EDITING THE CONTENTS OF MACHINING PROCESS**

By the following operations, an operator can select a program or process to be edited.
1) Push **RESET** key on a LCD/MDI keyboard.
2) Push [PROGRM] on an initial screen, then program list window appears.
3) Select a program to be edited by moving a cursor on a program list window, and push [EDIT], then process list window appears.



4) Select a process to be edited by moving a cursor on a process list window, and push [EDIT], then the process list is closed and selected program is displayed in a program window.
5) If necessary, modify the contents of machining process. As to the operations for each machining process, refer to the following corresponding explanations.

| | |
|---|---|
| **Guidance cutting** | 4.4 Editing Taught-in Blocks |
| **Single cutting** | 5.4 Editing Taught-in Blocks |
| **Cycle motions** | 6.3 Inputting Figure Data (Other than Bar Machining) |
| | 6.5 Editing Contour Figure Data (Bar Machining) |

6) Machining simulation by tool path drawing or one shot playback machining can be done, if necessary.
7) Push [SAVE] in order to replace the old process by the newly modified one. By pushing left end soft-key, this replacement is not done, and the newly modified process is canceled.
8) By pushing [SAVE] in the above case, a process list window appears again, and editing of other processes can be continued.
9) Push [RETURN], then a program list window is closed.

# 7.3     DELETING / COPYING A PROGRAM

On a program list window, deleting and copying of a program can be done.

## 7.3.1     Deleting a Program

1) Push **RESET** key on a LCD/MDI keyboard.
2) Push [PROGRM] on an initial screen, then program list window appears.
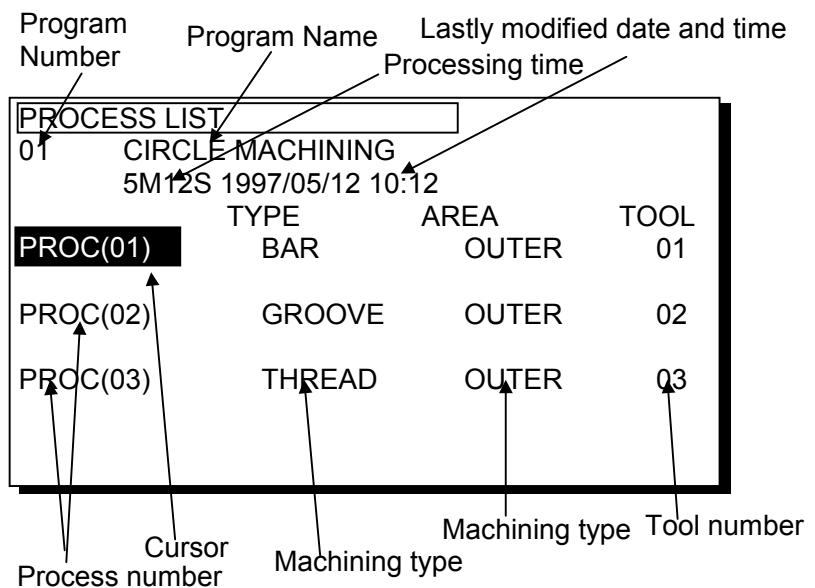3) Select a program to be deleted by moving a cursor on a program list window, and push **DELETE**.
   By this pushing, the message for confirming the operation is displayed.
   To cancel the operation, push **CAN**. To delete a data, push **DELETE** again.

## 7.3.2     Copying a Program

1) Push **RESET** key on a LCD/MDI keyboard.
2) Push [PROGRM] on an initial screen, then program list window appears.
3) Select a program to be copied by moving a cursor on a program list window, and input a program number of newly made program, after then push [COPY].

# *7.4*     PROCESS EDITING

## *7.4.1*     Moving a Process

1) Push **RESET** key on a LCD/MDI keyboard.
2) Push [PROGRM] on an initial screen, then program list window appears.
3) Select a program to be edited by moving a cursor on a program list window, and push [EDIT], then process list window appears.
4) Move the cursor to the process immediately before the place where you want to move the process.   Enter the number of the process to move with numeric keys and push [MOVE]. Then the process moves.
5) After moving a process, process number in a process list window will be renumbered automatically.

## *7.4.2*    **Copying a Process**

1)  Push **RESET** key on a LCD/MDI keyboard.
2)  Push [PROGRM] on an initial screen, then program list window appears.
3)  Select a program to be edited by moving a cursor on a program list window, and push [EDIT], then process list window appears.
4)  Select a process to be copied by moving a cursor on a process list window, and push [COPY]. Then, this selected process is copied just after it.
5)  After copying a process, process number in a process list window will be renumbered automatically.

## *7.4.3*    **Deleting a Process**

1)  Push **RESET** key on a LCD/MDI keyboard.
2)  Push [PROGRM] on an initial screen, then program list window appears.
3)  Select a program to be edited by moving a cursor on a program list window, and push [EDIT], then process list window appears.
4)  Select a process to be deleted by moving a cursor on a process list window, and push **DELETE**.
    By this pushing, the message for confirming the operation is displayed.
    To cancel the operation, push **CAN**. To delete a data, push DELETE again.
5)  After deleting a process, process number in a process list window will be renumbered automatically.

# 8

# PLAYBACK MACHINING AND MACHINING SIMULATION

By using a playback machining program that was already saved, playback machining and machining simulation can be done.

# 8.1      PLAYBACK MACHINING

1)   Push **RESET** key on a LCD/MDI keyboard.
2)   Push [PROGRM] on an initial screen, then program list window appears.

Cursor

```
PROGRAM LIST
 0 :CIRCLE MACHINING      5M12S      1997/05/12 10:12
10:LINE MACHINING         4M32S      1997/06/10 11:21
```

Program
Number        Program Name        Processing time
              (Max 20ch.)         (Max 99minuites)   Lastly
                                                      modified date
                                                      and time

3)   Select the program for playback machining on a program list window by moving a cursor onto its program number field.
4)   Push Playback Start button on a machine operator's panel or [EXEC.], and then the playback machining of the selected one will be started.
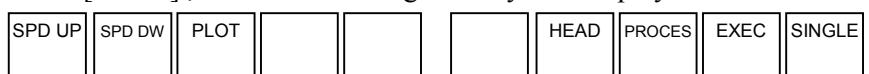
# *8.2*    **MACHINING SIMULATION**

1) Push **RESET** key on a LCD/MDI keyboard.
2) Push [PROGRM] on an initial screen, then program list window appears.

| | PATH | ANIMAT | ANIMAT +PATH | | | | | | |
|---|---|---|---|---|---|---|---|---|---|

## *8.2.1*    **Machining Simulation with Tool Path Drawing**

Push [PATH] , then the following soft-keys are displayed.

| SPD UP | SPD DW | PLOT | | | | HEAD | PROCES | EXEC | SINGLE |
|---|---|---|---|---|---|---|---|---|---|

**[SPD UP]**    Increase the machining simulation speed.
**[SPD DW]**    Reduce the machining simulation speed.
**[PLOT]**    Draw the machining figure entered in cycle cutting processes.
**[HEAD]**    Return to the top of this process, and initialize a workpiece.
**[PROCES]**    Start simulating the machining in units of processes. The output of M01 causes a single block stop. (To output M01 at the end of process, set a parameter No.9774#4(M1O) to 1)
**[EXEC]**    Start the continuous simulation drawing.
**[SINGLE]**    Cause a single block stop while continuous simulation is being executed. Or, starts single block simulation.

In order to return to the initial screen, push **RESET**, and then push left end soft-key, super return.

## *8.2.2*    **Machining Simulation with Solid Model Animated Drawing**

Push [ANIMAT] , then the following soft-keys are displayed.

| SPD UP | SPD DW | PLOT | ROTATE | CIRC | | HEAD | PROCES | EXEC | SINGLE |
|---|---|---|---|---|---|---|---|---|---|

**[SPD UP]**    Increase the machining simulation speed.
**[SPD DW]**    Reduce the machining simulation speed.
**[PLOT]**    Draw the machining figure entered in cycle cutting processes over the workpiece figure.
**[ROTATE]**    Rotate the workpiece.
**[CIRC]**    Change the workpiece figure between full circle and 3/4 drawing.
**[HEAD]**    Return to the top of this process, and initialize a workpiece.
**[PROCES]**    Start simulating the machining in units of processes. The output of M01 causes a single block stop. (To output M01 at the end of process, set a parameter No.9774#4(M1O) to 1)
**[EXEC]**    Start the continuous simulation drawing.
**[SINGLE]**    Cause a single block stop while continuous simulation is being executed. Or, starts single block simulation.

## *8.2.3*    Machining Simulation with Animated Drawing and Tool Path

Push [ANIMAT+PATH] , then the following soft-keys are displayed.

| SPD UP | SPD DW | PLOT | | | | HEAD | PROCES | EXEC | SINGLE |
|---|---|---|---|---|---|---|---|---|---|

At this machining simulation mode, instead of tool path drawing, animated drawing for solid model cannot be done. So, soft-keys, [ROTATE] and [CIRC.] cannot be used.
As to other soft-keys, they are similar to 8.2.3.
In order to return to the initial screen, push **RESET**, and then push left end soft-key, super return.

# 9    SETTING DATA

Before creating a program in MANUAL GUIDE mode, data items such as Tool Data and Common Parameters must be set.

# *9.1*    **TOOL DATA**

## *9.1.1*    **Tool File Window**

By pushing a [TOOL], the tool file window is displayed.

| TOOL FILE | | | | | | | |
|---|---|---|---|---|---|---|---|
| TOOL NO. | GEOMETRY | | | WEAR | | | TIP |
| | X | Z | R | X | Z | R | |
| 1 | -10.000 | -20.000 | 0.400 | 0.000 | 0.000 | 0.000 | 3 |
| 2 | -11.000 | -21.000 | 0.800 | 0.000 | 0.000 | 0.000 | 3 |
| 3 | -12.000 | -22.000 | 0.400 | 0.000 | 0.000 | 0.000 | 3 |
| 4 | -20.000 | -25.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 5 | -25.000 | -26.000 | 0.200 | 0.000 | 0.000 | 0.000 | 0 |
| 6 | -30.000 | -27.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 7 | -35.000 | -28.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 8 | -40.000 | -30.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 9 | -35.000 | -28.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 10 | -40.000 | -30.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 11 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 12 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 13 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 14 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 15 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |
| 16 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0.000 | 0 |

## *9.1.2*    **Operation of Tool File Window**

| | |
|---|---|
| **[RETURN]** | Close this tool file window. |
| **Cursor keys** | Move a cursor to vertically or horizontally. |
| **INPUT, [INPUT]** | Input a numeric data on a key-in buffer into a corresponding data and move a cursor to next data. |
| **[+INPUT]** | Input an added or subtracted (- inputted) value into a corresponding data and move a cursor to next data. |

> **NOTE**
> As to the wear offset data, the following parameters are valid.
> Parameter No.5013: Maximum value of wear offset.
> Parameter No.5014: Maximum value of [+INPUT]
>                    data for wear offset.

## *9.1.3*   **Detail of Tool File Data**

**TOOL NO.**          Commonly used for outputted T-code number and offset number.

**GEOMETRY X/Z/R**   X-axis, Z-axis and Nose-R geometry offset, and these are commonly used with ISO programming mode.

**WEAR X/Z/R**       X-axis, Z-axis and Nose-R wear offset, and these are commonly used with ISO programming mode.

**TIP**              Imaginary tool tip position, and it is commonly used with ISO programming mode.
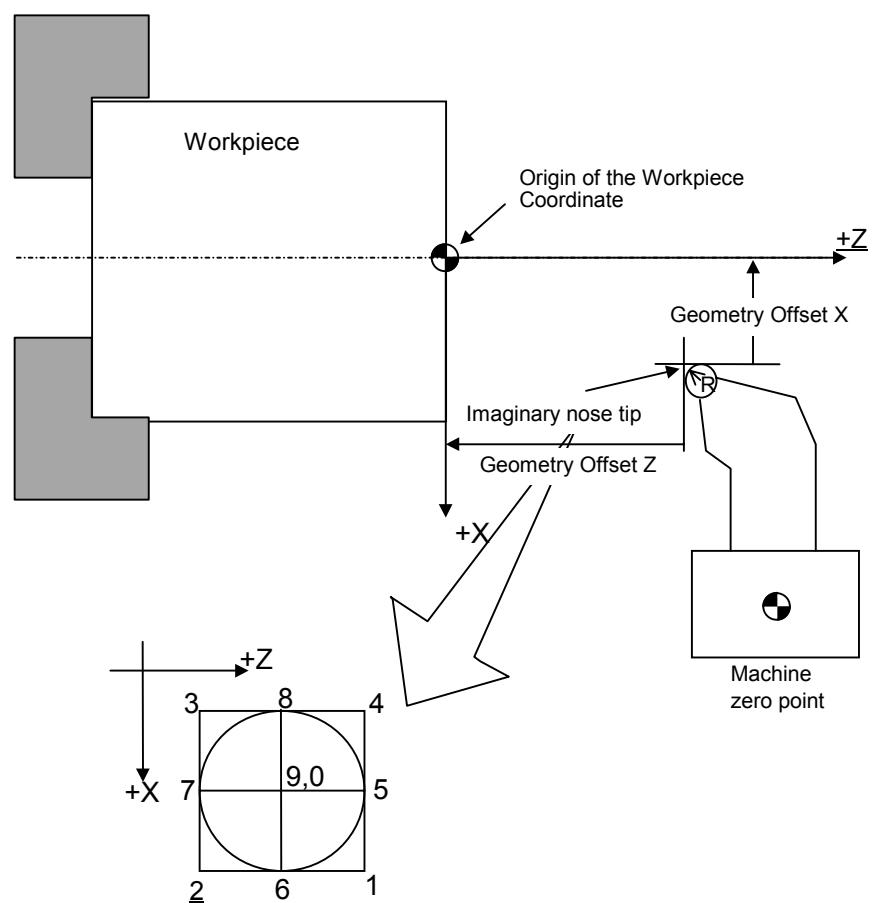
# *9.2*    MEASURING WORKPIECE COORDINATE SHIFT AMOUNT

By pushing a [PREST] in an initial screen, the setup guidance window for measuring workpiece coordinate shift amount appears.

> ⚠ **WARNING**
> When measuring workpiece coordinate shift amount on the X-/Z-axis using this method, keep the spindle rotating, and push the tool against the end surface or periphery of the workpiece and cut it with a feedrate as low as possible, using such as handwheel. Also, be sure to keep the safety door of the machining station closed.
> Refer to the applicable manual supplied by the respective machine tool builders for detailed operations. If you do not know how to use this measurement function, do not use it.
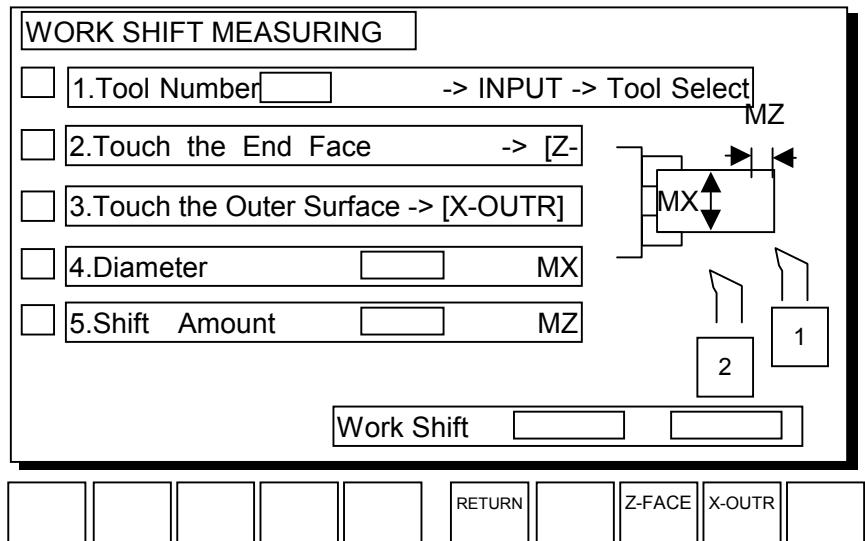> Use of this measurement method without a working knowledge of the operating procedure may bring about the death of, or a serious injury to, the user. Before starting the machine using the tool compensation function, carefully determine the direction of compensation and the compensation value, and ensure that the tool will not strike the workpiece or machine. Otherwise the tool or machine may be damaged or the operator may injured.

## (1) Manual reference position return

First, manually return to the reference position. When reference position return has been performed previously, this is unnecessary.

## (2) **Workpiece shift amount preset window**

In an initial screen, by pushing a [PRESET], a workpiece shift preset window as follows is displayed.

```
┌─────────────────────────────────────────────────────────────┐
│ │WORK SHIFT MEASURING    │                                   │
│                                                              MZ
│ □ │1.Tool Number│      │    -> INPUT -> Tool Select│         │
│                                                        ►│ │◄ │
│ □ │2.Touch  the  End  Face        ->  [Z-│              │   │ │
│                                                      MX│▲     │
│ □ │3.Touch the Outer Surface -> [X-OUTR]│             │▼     │
│                                                              │
│ □ │4.Diameter          │      │      MX│                     │
│                                                        ┌──┐  │
│ □ │5.Shift   Amount     │      │     MZ│          ┌──┐ │1 │  │
│                                                   │2 │ └──┘  │
│                          │Work Shift │    │   │   │ │       │
│                                                   └──┘       │
└─────────────────────────────────────────────────────────────┘
┌──┐┌──┐┌──┐┌──┐┌──────┐┌──┐┌──────┐┌──────┐┌──┐
│  ││  ││  ││  ││RETURN││  ││Z-FACE││X-OUTR││  │
└──┘└──┘└──┘└──┘└──────┘└──┘└──────┘└──────┘└──┘
```

## (3) **Tool index operation**

Move a cursor on a "Tool number", input a tool number to be used for presetting workpiece shift amount, and push **INPUT**.

After then, on a machine operator's panel, push a tool selecting button. By this operation, a turret rotates and selected tool is assigned for measuring workpiece shift amount.

At this time, push CSS OFF switch on machine operator's panel and after then make a spindle rotate by pushing a spindle rotating button. This rotating speed can be changed by common parameter displayed in an initial screen.

## (4) **Measuring Z-axis workpiece shift amount**

Move a tool and make touch it to the end face of a workpiece by rotating handwheel slowly. Then, push [Z-FACE] for teaching the end surface coordinate.

## (5) **Measuring X-axis workpiece shift amount**

After releasing a tool from a workpiece, move tool and make the outer surface of a workpiece. At this time, in order to do measuring of next step precisely, cut a workpiece a little. After then, push [X-OUTR] for teaching the outer surface coordinate.

## (6) **Measuring diameter of cut workpiece and inputting**

By using a measure such as calipers, measure a diameter amount of cut part by the above operations. Of course, before then, the spindle must be stopped by pushing a spindle stop switch.

Move a cursor on a "Diameter MX" part, input a measured diameter amount, and push **INPUT**.

By these operations, workpiece shift amount of X-axis is calculated and is displayed at a lower part of this window.

## (7) Inputting end surface shift amount

In ordinary case, after moving a cursor on a "Shift Amount MZ" part, input "0" value and push **INPUT**. By some reason, if some shifting of end surface is needed, input the shifting amount value and push **INPUT**.

By these operations, workpiece shift amount of Z-axis direction is calculated and is displayed at a lower part of this window.

By pushing a [RETURN], this window can be closed.

> **NOTE**
> Operations for reading actual X and Z workpiece coordinate by pushing [Z-FACE] or [X-OUTR] can be done only one time. The reason for it is as follows.
> Once measured a workpiece coordinate shift amount, workpiece coordinate value is changed according to the measured amount, so if the reading of workpiece coordinate value more than one time, the correct value will never be calculated correctly.
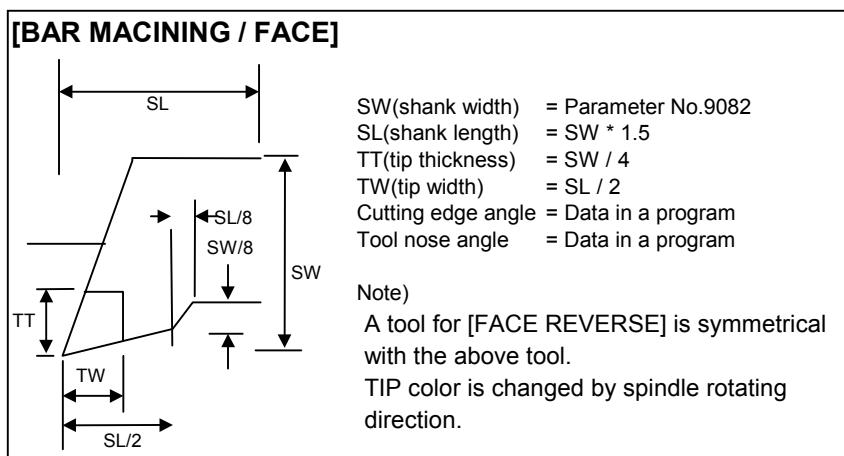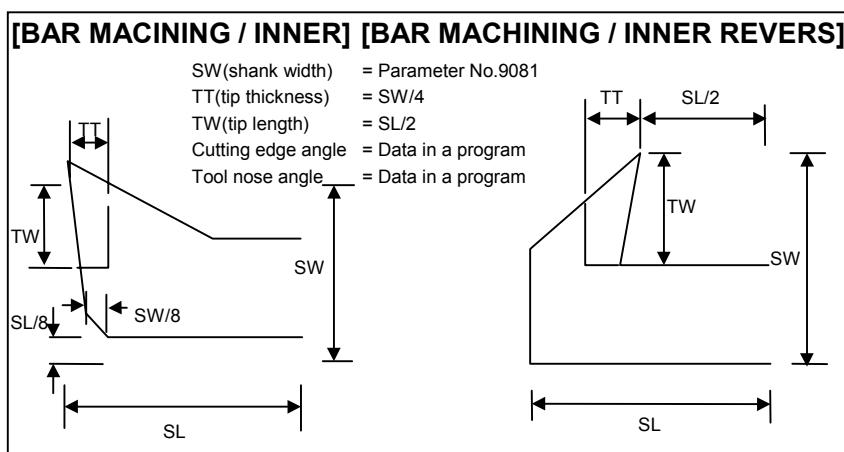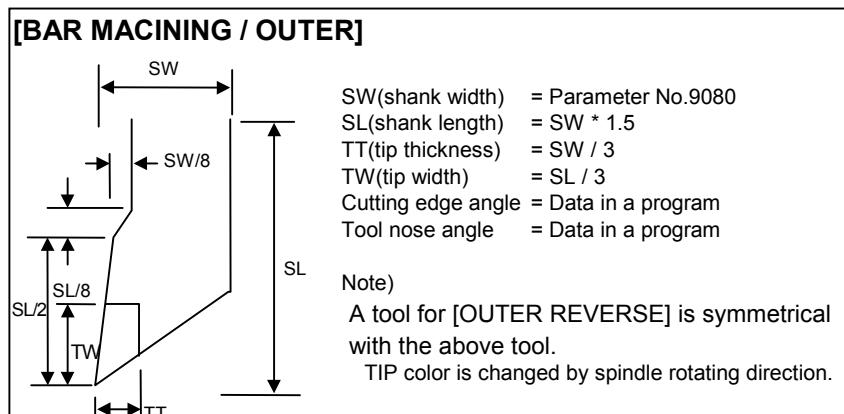> In order to avoid to measure workpiece coordinate more than one time, soft-key [Z-FACE] and [X-OUTER] will be disappear after operations for measuring.
> By closing this measuring window or inputting a tool number again, measuring can be done again.
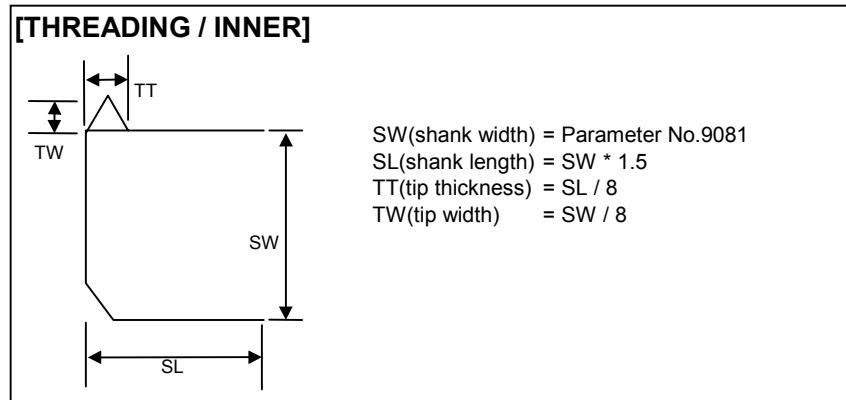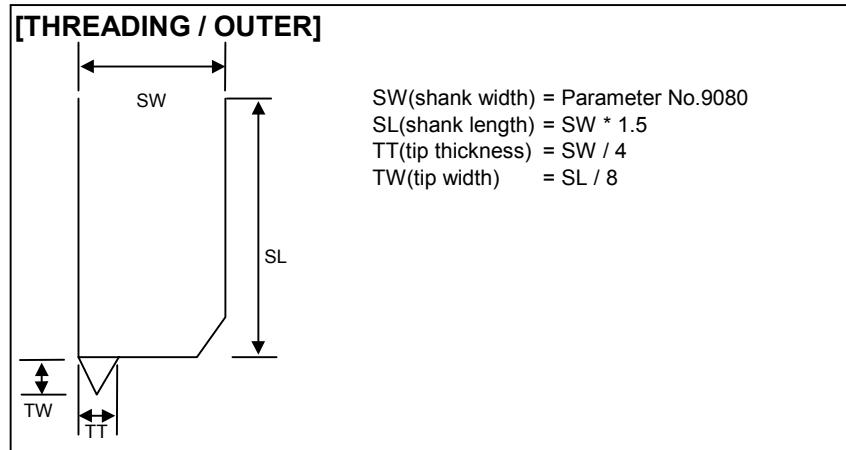
# 9.3    TOOL FORM FOR ANIMATED DRAWING

The tool form for animated drawing is automatically specified as follows.
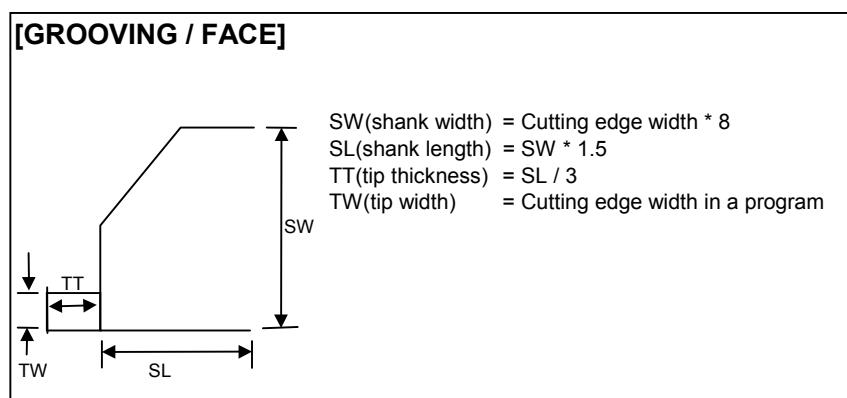
## 9.3.1    General-Purpose Cutting Tools
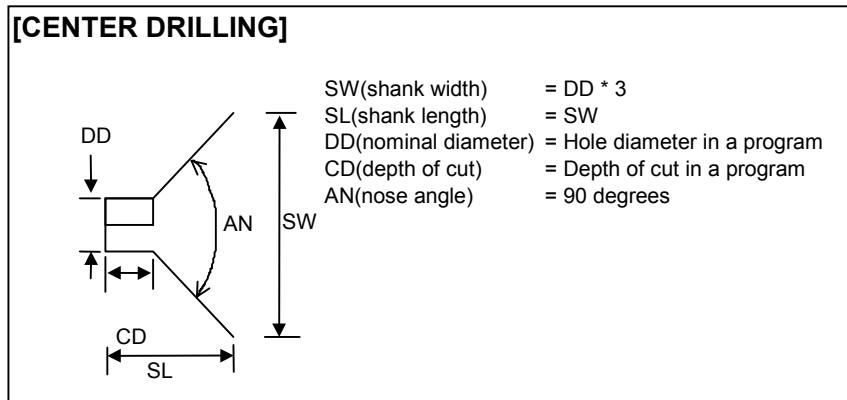
**[BAR MACINING / OUTER]**

SW(shank width)    = Parameter No.9080
SL(shank length)   = SW * 1.5
TT(tip thickness)  = SW / 3
TW(tip width)      = SL / 3
Cutting edge angle = Data in a program
Tool nose angle    = Data in a program

Note)
 A tool for [OUTER REVERSE] is symmetrical with the above tool.
   TIP color is changed by spindle rotating direction.

**[BAR MACINING / INNER] [BAR MACHINING / INNER REVERS]**

SW(shank width)    = Parameter No.9081
TT(tip thickness)  = SW/4
TW(tip length)     = SL/2
Cutting edge angle = Data in a program
Tool nose angle    = Data in a program

**[BAR MACINING / FACE]**

SW(shank width)    = Parameter No.9082
SL(shank length)   = SW * 1.5
TT(tip thickness)  = SW / 4
TW(tip width)      = SL / 2
Cutting edge angle = Data in a program
Tool nose angle    = Data in a program

Note)
 A tool for [FACE REVERSE] is symmetrical with the above tool.
 TIP color is changed by spindle rotating direction.

## *9.3.2*　　**Threading Tools**

**[THREADING / OUTER]**

SW(shank width) = Parameter No.9080
SL(shank length) = SW * 1.5
TT(tip thickness) = SW / 4
TW(tip width)　　 = SL / 8

**[THREADING / INNER]**

SW(shank width) = Parameter No.9081
SL(shank length) = SW * 1.5
TT(tip thickness) = SL / 8
TW(tip width)　　 = SW / 8

## *9.3.3*    Grooving Tools

**[GROOVING / OUTER]**

SW(shank width) = Cutting edge width * 8
SL(shank length) = SW * 1.5
TT(tip thickness) = SL / 3
TW(tip width)      = Cutting edge width in a program

**[GROOVING / INNER]**

SW(shank width) = Cutting edge width * 8
SL(shank length) = SW * 1.5
TT(tip thickness) = SW / 3
TW(tip width)      = Cutting edge width   in a program

**[GROOVING / FACE]**

SW(shank width) = Cutting edge width * 8
SL(shank length) = SW * 1.5
TT(tip thickness) = SL / 3
TW(tip width)      = Cutting edge width in a program

## *9.3.4*    **Center Drills**

**[CENTER DRILLING]**

SW(shank width)        = DD * 3
SL(shank length)       = SW
DD(nominal diameter) = Hole diameter in a program
CD(depth of cut)        = Depth of cut in a program
AN(nose angle)          = 90 degrees

## *9.3.5*    **Drills**

**[DRILLING]**

DD(nominal diameter) = Hole diameter in a program
SL(depth of cut)        = Depth of cut in a program
AN(nose angle)          = Parameter No.9083

## *9.3.6*    **Taps**

**[TAPPING]**

DD(nominal diameter) = Hole diameter in a program
SL(depth of cut)        = Depth of cut in a program

## *9.3.7*    **Reamers**

**[REAMING]**

DD(nominal diameter) = Hole diameter in a program
SL(depth of cut)        = Depth of cut in a program

DD

SL

## *9.3.8*    **Boring Tools**

**[BORING]**

TT

TW

SW(shank width) = Hole diameter - Shift
                            (both in a program)
SL(shank length) = SW * 1.5
TT(tip length)      = SL/4
TW(tip width)      = SW/2

SW

SL

## *9.3.9*    **Round Nose Cutting Tools**

**[GAUIDANCE / SINGLE CUTTING]**

NR

NR(nose radius)    = R geometry offset in a tool file
Cutting edge angle = 0
Tool nose angle     = 0

# 10 CONVERTING MACHINING PROGRAM INTO NC PROGRAM

A machining program saved in the manual guide mode can be converted into an NC program with G-code, M-code and so on.

> **NOTE**
> 1  This function is available when an optional function "Background editing" is added and the parameter No.9760#6(NCO) is set to 1.
> 2  While the program is being converted, no other operation such as background editing, manual movement or data setting can be performed.
> 3  When this function is available, editing B is validated. Two or more address (for example, X100. Z100.) can be edited at a time.
> 4  The program number of the converted NC program is always set to the number of the original program in manual guide format.

# *10.1*    OPERATIONS FOR CONVERSION

1)    Push **RESET** key on a LCD/MDI keyboard.
2)    Push [PROGRM] on an initial screen, then program list window appears.
3)    Select a program to be edited by moving a cursor on a program list window, and push [CNV NC], then the following soft-keys appears.

| SPD UP | SPD DW | PLOT | | | | HEAD | PROCES | EXEC | SINGLE |
|---|---|---|---|---|---|---|---|---|---|

4)    By pushing [EXEC], conversion to the NC program is started and the animated drawing is executed at the same time.
During the conversion, the "EDITING" message blinks at the bottom of screen, and it disappears when the operation ends.
During conversion, avoid changing the screen to another screen or performing other operations.

## *10.2*   ALARMS DURING CONVERSION OF THE MACHINING PROGRAM INTO NC PROGRAM

While the machining program is being converted into the NC program, the machining program is running for check drawing. A P/S alarm may occur because of the contents of the machining program.

For the P/S alarms, see appendix II. which describes the alarms that may occur when the machining program is running. Then correct the machining program.

If the P/S alarm occurs during the conversion, the converted part of the machining program has already been stored in the tape memory area. To restart the conversion of the machining program, correct the erroneous part, delete the converted part, and perform conversion from the beginning.

The following alarms may occur while the conversion.

| Alarm No. | | Description |
|---|---|---|
| 070 | Cause | Insufficient tape memory area |
| | Action | Take the following action depending on the condition:<br>1) If another NC program is already registered and the remaining memory is insufficient for the program being currently converted.<br>-> Save the registered NC program in the external memory unit via the reader/puncher interface, delete the program, and resume conversion.<br>2) If the entire reserved area is insufficient for the program being currently converted.<br>-> The NC program area can be increased by setting parameters. However, this causes the manual guide program area to be reduced and all data items to be cleared temporarily. |
| 072 | Cause | The number of registered programs (it depends on an optional setting) exceeded. |
| | Action | Save some of the registered NC programs into the external memory unit via the reader/puncher interface, delete the programs, and resume conversion from the beginning. |
| 073 | Cause | An attempt was made to specify the same program number as that of a registered program. |
| | Action | Delete the registered NC program having the same number and resume conversion from the beginning. |

# 11  OUTPUTTING / INPUTTING MACHINING PROGRAMS

A manual guide format program can be outputted to an external memory unit via the reader / puncher interface, and it can be inputted similarly.

> **NOTE**
> These outputting and inputting can be done under only EDIT mode, but depending on a machine or MTB, there is a case that the mode is changed to EDIT mode automatically at these operations. Into details, refer to the machine operating manual made by MTB.

# *11.1*    **OUTPUTTING MACHINING PROGRAMS**

1) Push **RESET** key on a LCD/MDI keyboard.
2) Push [PROGRM] on an initial screen, then program list window appears.
3) By pushing a right end soft-key [+], the following soft-keys appears.

| ALL-PN | PUNCH | READ | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|

4) To output all machining program of manual guide format, push a [ALL-PN].
5) To output only one program, select a program to be outputted by moving a cursor on a program list window, and push a [PUNCH].
6) When the output of the machining program starts, the "OUTPUT" message blinks at the bottom of the screen. It continues to flash until the operation ends.

---

**NOTE**
   If the output equipment is the FANUC Cassette, a
   new file is created immediately after the registered
   file.

---

# *11.2*    INPUTTING MACHINING PROGRAMS

1) Push **RESET** key on a LCD/MDI keyboard.
2) Push [PROGRM] on an initial screen, then program list window appears.
3) By pushing a right end soft-key [+], the following soft-keys appears.

| ALL-PN | PUNCH | READ | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| | | | | | | | | | |

4) Using numeric keys, enter the number of the file containing the machining program to be read and push a [READ]. Reading of the machining program is started.
5) If the input equipment is other than the FANUC Handy File or its equivalent one, input 1 as a file number always.
6) When reading of the machining program starts, the "INPUT" message blinks at the bottom of the screen. It continues to flash until the operation ends.

# III. TYPES OF CYCLE MOTIONS

# 1    BAR MACHINING (ROUGHING)
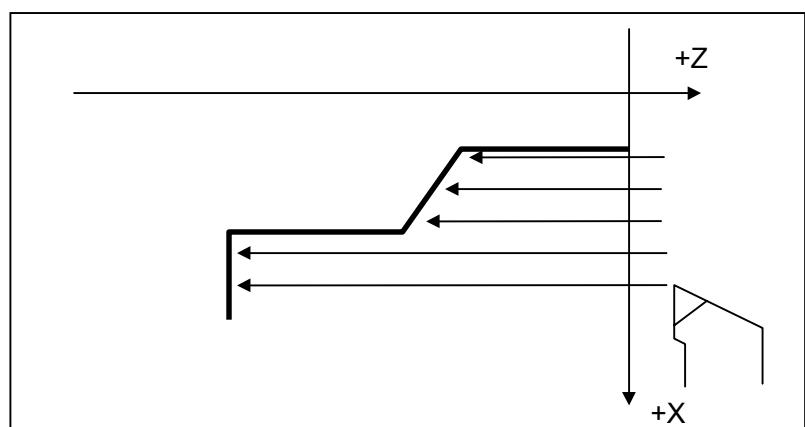
# *1.1*    DETAIL OF PROCESS DATA

## *1.1.1*    Process Data Window

```
BAR MACHINING (ROUGH)
AREA         =        CUT ST Z    =
TOOL NUM.  =        FIN X       =
FEED/REV.   =        FIN Z       =
S-SPD/MIN   =        INFEED AM  =
S-DIRECT    =
CUT ST X     =
```

AREA                Select from [OUTER]/[INNER]/[FACE]/[OUTER
                    REVERS] /[INNER/REVERS]/[FACE REVERS]
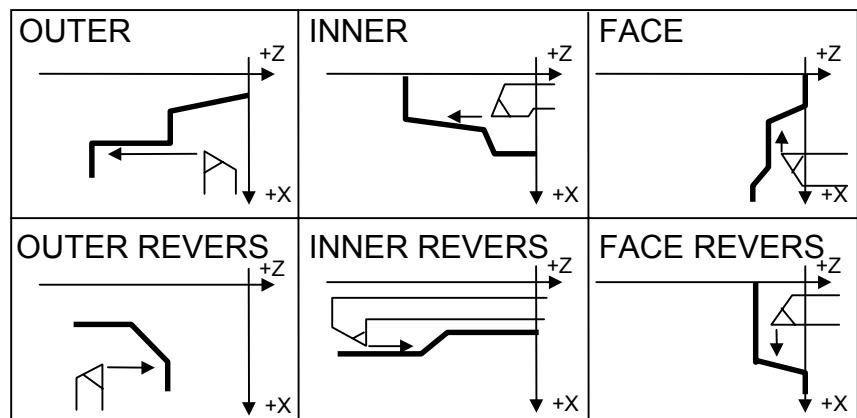


| TOOL NO. | Number of tool used for roughing bar machining |
| --- | --- |
| FEED/REV. | Cutting feedrate, and it can be changed to FEED/MIN. by pushing soft-key [FEED/MIN.]. |
| S-SPD/MIN | Cutting speed, and it can be changed to REV/MIN. by pushing soft-key [REV/MIN.]. |
| S-DIRECT | Spindle rotating direction. Select from [NORMAL] and [REVERS]. |
| CUT ST X/Z | Coordinate of cutting start position. These are automatically set by inputting figure data. |
| FIN X | X direction amount of finishing. |
| FIN Z | Z direction amount of finishing. |
| INFEED AM | Infeed amount of the first cutting motion. |

# *1.1.2*    **Supplemental Process Data in a Pop-up Window**

```
EDGE ANGL  =            COOLANT    =
NOSE ANGL  =            SPDL GEAR=
RESE.AMNT  =
RELESE     =
APROCHX    =
APROCHZ    =
```

These supplemental process data are automatically set always, so in ordinary case, an operator need not to call this pop-up window excepting the case of checking these data.

**EDGE ANGL**      Angle of the cutting edge. In case of that it was set to the value less than 90 degrees, cutting edge compensation will be done automatically. Parameter No.9809 value is automatically set into this data at initially inputting this process.
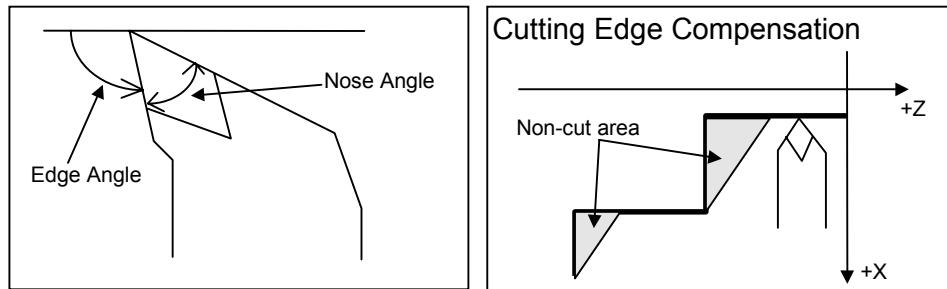




**NOSE ANGLE**      Angle of the tool tip, its default value can be set by parameter. In the case of that pocket figure was inputted as contour figure, cutting back compensation will be done automatically. Parameter No.9810 value is automatically set into this data at initially inputting this process.
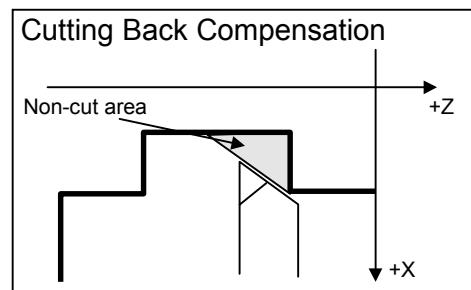
**RESE.AMNT**    Travel amount in X-axis(OUTER/INNER : diameter) or
                 Z-axis(FACE : radius) for retraction after cutting.
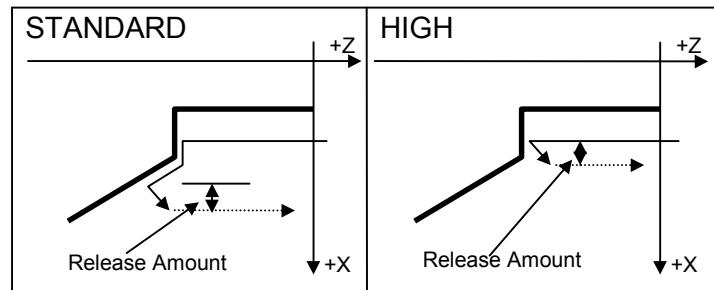


**RELEASE**      Type of tool positioning after cutting. Default value is
                 [STNDRD].

      **[STNDRD]**    The tool is retracted after cutting along the inputted
                      figure.
        **[RAPID]**   The tool is retracted immediately after cutting.



**APROCH X**     X coordinate of approaching intermediate point.
**APROCH Z**     Z coordinate of approaching intermediate point.


**COOLANT**      Coolant specification at playback. Choose from the
                 following soft-key menu. Default value is [ON].
                         [ON]
                         [OFF]
**SPDL GEAR**    Spindle gear at playback. Choose from the following
                 soft-key menu. Default value is [AUTO].
                         [AUTO]
                         [LOW]
                         [MID1]
                         [MID2]
                         [HIGH]

# *1.2*    **DETAIL OF FIGURE DATA**

In Bar machining, up to 30 figures can be specified for one process.

## (1) Inputting Start Point

At the top of contour figure, Start Point figure data is placed always. Figure data window)

```
        X▮▮▮▮▮▮   Z          ;


```

**START X/Z**          Absolute coordinate of figure start point

> **NOTE**
> As to Start Point figure data, different from other figure data such as line or circle whose data are inputted in a pop-up window, X and Z data are inputted in a figure data window directly.

## (2) Selecting a figure type

A menu of figure type appears in soft-keys. Select a figure type by pushing the corresponding soft-key.

| LINE | ARC ↰ | ARC ↴ | ROUND | CHAMF | TANGNT | | | CALC. |
|------|-------|-------|-------|-------|--------|--|--|-------|

When a figure has been selected, a pop-up window for inputting contour figure data appears. Enter the data that is designated on the blue print.

## (3) LINE

```
CONTOUR FORM (LINE)
  END POINT X    X=
  END POINT Z    Z=
  ANGLE          A=
  OVERRIDE       P=
```

**END POINT X/Z**    Absolute coordinate of figure end point
**ANGLE**            Angle of line from +Z-axis. A positive angle is clockwise direction from +Z-axis, and minus angle is counter clockwise direction.
**OVERRIDE**         This data is not used in roughing.

## (4) ARC (CW/CCW)

```
CONTOUR FORM (ARC    CW)

END POINT X     X=
END POINT Z     Z=
RADIUS          R=
CENTER X        I=
CENTER Z        K=
OVERRIDE        P=
```
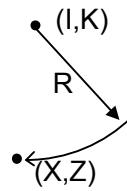
| | |
|---|---|
| **END POINT X/Z** | Absolute coordinate of figure end point |
| **RADIUS** | Radius of arc |
| **CENTER X/Z** | Absolute coordinate of arc center |
| **OVERRIDE** | This data is not used in roughing. |

---

**NOTE**

No major arc can be entered. For bar machining, even a semicircle or major are cannot be entered if that semicircle or minor are extends over three or more quadrants.

---

## (5) ROUND

```
CONTOUR FORM (ROUND)

RADIUS          R=
OVERRIDE        P=
```
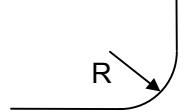
| | |
|---|---|
| **RADIUS** | Radius of Corner-R |
| **OVERRIDE** | This data is not used in roughing. |

## (6) CHAMFER

```
CONTOUR FORM (CHAMF)

CHAMFER         C=
OVERRIDE        P=
```
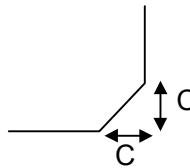
| | |
|---|---|
| **CHAMFER** | Chamfer amount |
| **OVERRIDE** | This data is not used in roughing. |

## (7) TANGNT

Pushing [TANGNT] specifies that the next figure form a tangent to the previous figure (either figure must be a arc). For next figure, arc or line must be entered.

When, [TANGNT] is selected, "TANGNT" is displayed on a screen.

## (8) CALC.

When a new contour figure is added at the end of a series of previously inputted figures, contour calculating is automatically done after pushing **INSERT**.

On the other hand, when some figures are changed after inputted and contour calculated, during data changing, there may be a case that some contradiction between changing figures.

By this reason, contour calculation at figure editing is done only at [CALC.] is pushed by an operator.

---

**NOTE**

In case of specifying contour figures, there are the following limitations according to the selected type of machining area.

---

- In case that machining area is Outer, Outer Reverse, Inner or Inner Reverse,   contour figures must be monotonous change along the Z-axis.
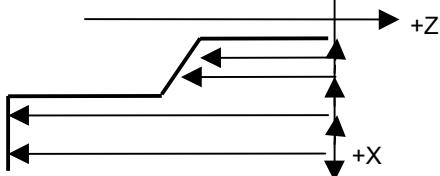
- In case that machining area is Face or Face Reverse, contour figures must be monotonous change along the X-axis.
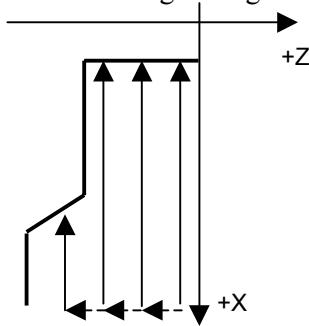
- As the following examples, in case of machining a face area in Bar machining, contour figures must be specified by 2 figures as follows.

Example 1)



Specifying contour figures(*)

Cutting a end face

Example 2)



Specifying contour figures(*)

Cutting a outer surface

Example 3)



Specifying contour figures(*)

Cutting a inner surface
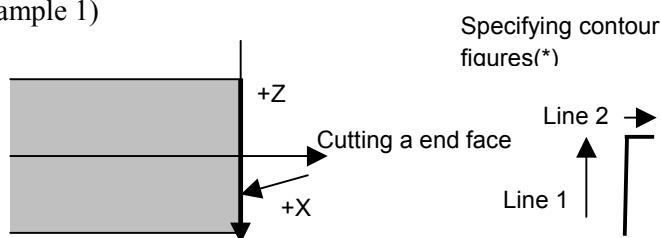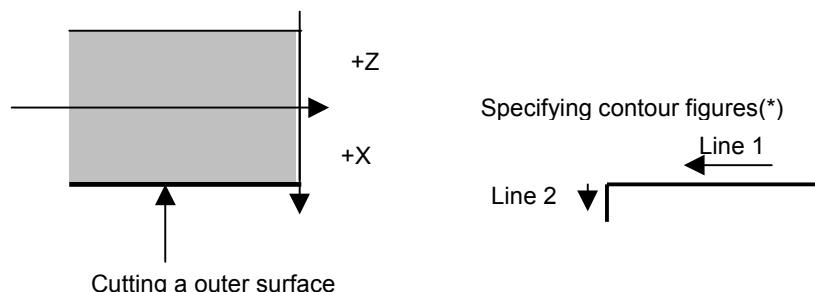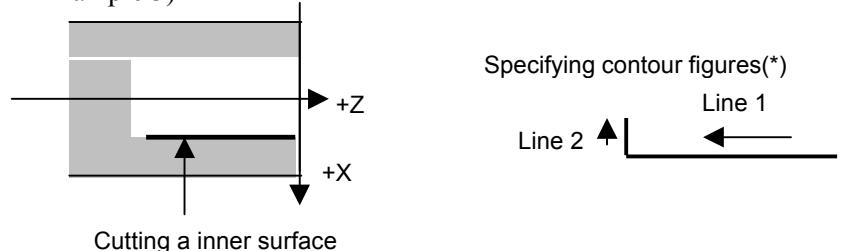
(*)   In these examples, it is not allowed to specify only a Line 1. It is necessary to specify Line 1 and Line 2.

# *1.3*     **DETAIL OF CONTOUR CALCULATION**

A figure block, a part of a contour, with its end point not determined is said to be in the pending state. A pending figure block is drawn by dotted line.

In the pop-up window for inputting contour figure data, more data input items than required are provided. These data items are used to calculate the cross point with the immediately preceding pending figure block, and also calculate the end point. The pending state is then released.

> **NOTE**
> 10 successive figure blocks can be specified as pending blocks.

## (1) CHAMF / ROUND immediately after a start point

Immediately after CHAMF or ROUND, only LINE or ARC can be inputted.
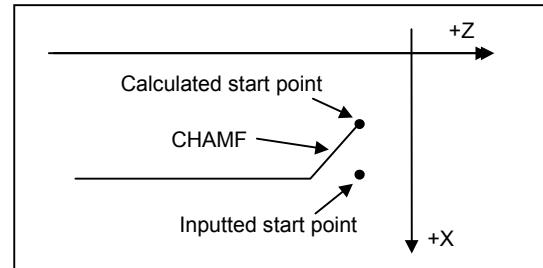


## (2) CHAMF / ROUND at the end of a contour

At the end of a contour, CHAMF or ROUND can be inputted. In this case, input a dummy figure block as a pending block. In this dummy block, input only angle data, with its end point pending.



- 105 -
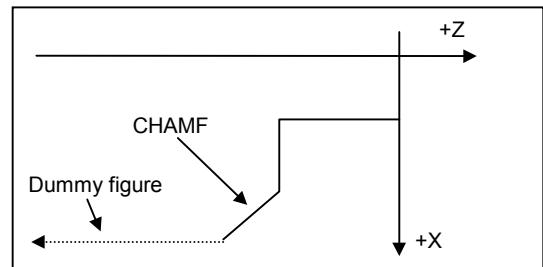
## (3) LINE

    (a)  When the preceding figure block is not pending
        (i)  Only X is inputted
            ->   This line is determined as a vertical line.
        (ii)  Only Z is inputted
            ->   This line is determined as a horizontal line.
        (iii) A and either X or Z   are inputted
            ->   The end point which is not inputted is calculated.
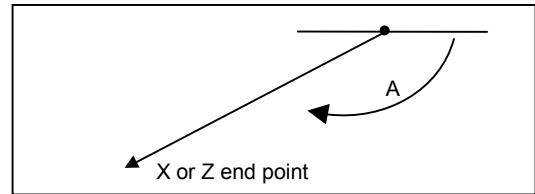


    (b)  When the preceding figure block specifying an arc is not pending, and TANGNT is specified.
        (i)  Either X or Z is inputted
            ->   The angle A is calculated automatically, and an end point is determined. If neither X nor Z is inputted, this line will be pending.
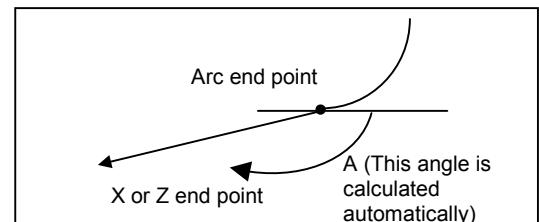


    (c)  When the preceding figure block is pending, and TANGNT is not specified.
        (i)  Both X and Z, and A are inputted
            ->   The cross point between the preceding figure block is calculated.
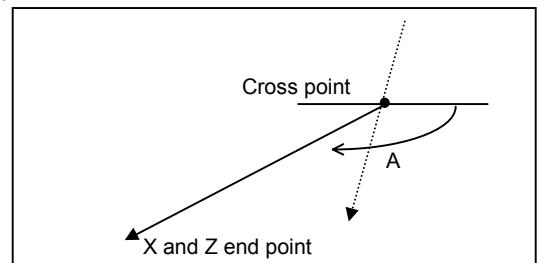


        When the preceding figure is an arc, the cross point selection request window is displayed. Input number data 1 or 2, then push **INSERT**.
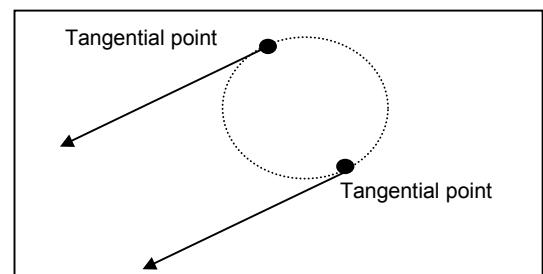
(d) When the preceding figure block is pending arc, and TANGNT is specified.

It is assumed that the radius and the center point coordinate (I,K) of arc have already inputted.

(i) Only A is inputted

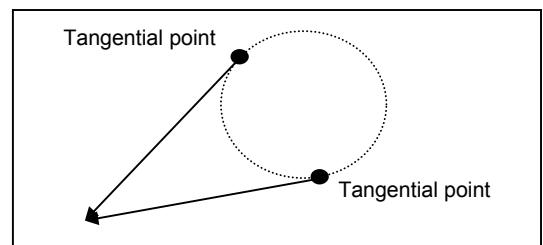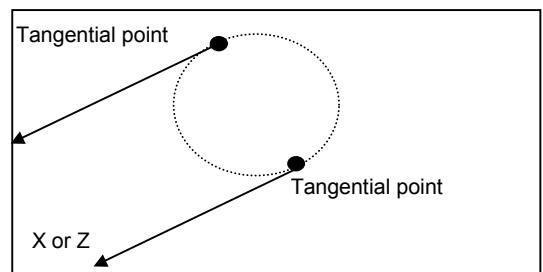-> The tangential point selection request window is displayed. Input number data 1 or 2, then push **INSERT**

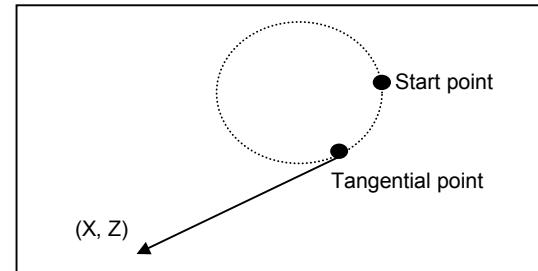This line will be pending.



(ii) Both X and Z are inputted

-> The tangential point selection request window is displayed. Input number data 1 or 2, then push **INSERT**

The end point of this line will be determined.



(iii) A and either X or Z are inputted

-> The tangential point selection request window is displayed. Input number data 1 or 2, then push **INSERT**

The end point of this line will be determined.



If the positional relationship between the tangential point and the line is such that an inputted A conflicts with the inputted X or Z, a warning message is displayed to indicate that invalid data has been inputted.

(iv) A and both X and Z are inputted
   -> The end point of the arc and line will be determined. The tangential point selection request window is not displayed.
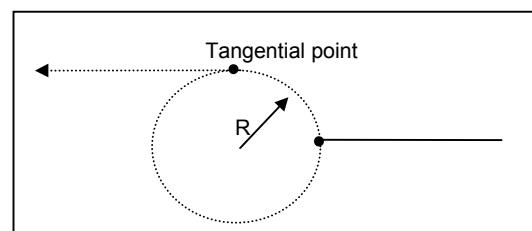


## (4) ARC

(a) When the preceding figure block is not pending, and TANGNT is not specified
   (i) I and K are inputted
      -> This arc will be pending.
   (ii) X, Z and R are inputted, or
   (iii) X, Z, I and K are inputted
      -> A minor arc will be determined.
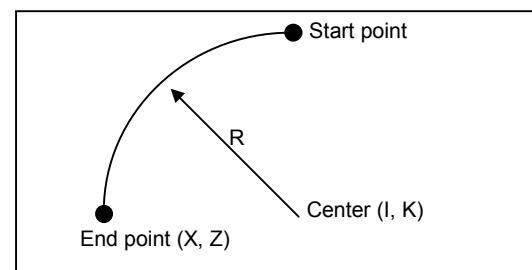


**NOTE**
In the distance (radius) between the start point and center differs from the one between the end point and center, the figure displayed in a graphic window will differ from the actual form, and machining will not be performed correctly.

(iv) Only R is inputted
   -> By specifying TANGNT and inputting a line with A = 180 degree and X coordinate as an immediately after figure, this arc can be determined.
   However, there are two possible tangential points, so one of them must be selected by inputting number data 1 or 2.



- 108 -

(b)  When the preceding figure block is not pending, and TANGNT is specified
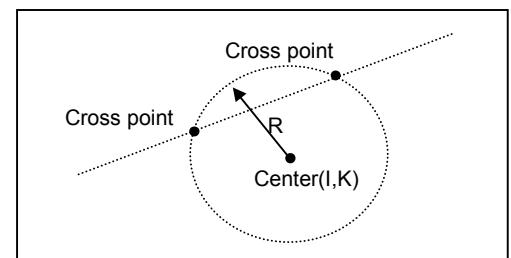(i)   X and Z are inputted
->    The radius is automatically calculated and this arc will be determined.
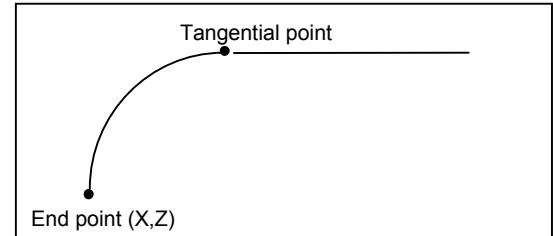


(c)  When the preceding figure block is pending (for which the start point has been determined), and TANGNT is not specified
(i)   R, I and K are inputted
->    The cross point selection request window is displayed. Input number data 1 or 2, then push INSERT. This arc will be pending.



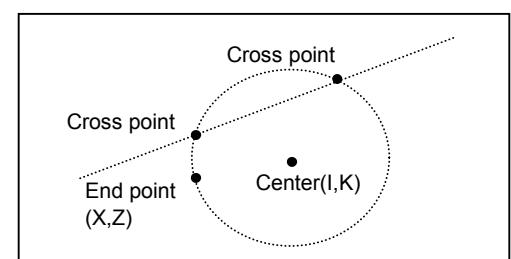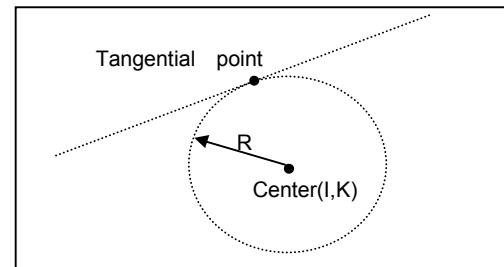(ii)  X, Z,  I and K are inputted
->    The cross point selection request window is displayed. Input number data 1 or 2, then push INSERT. This arc will be determined.



- 109 -

(d) When the preceding figure block is pending (for which the start point has been determined), and TANGNT is specified

    (i)   R, I and K are inputted
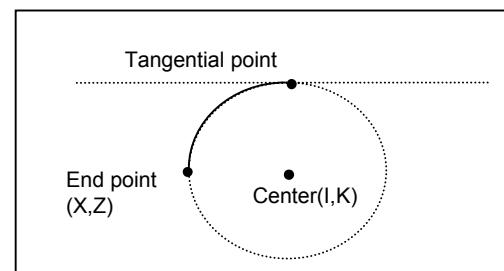
        -> The tangential point is calculated, and this arc will be pending.



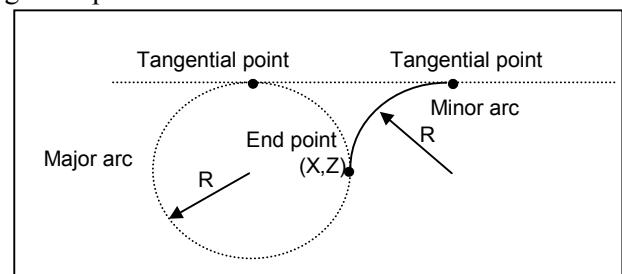    (ii)  X, Z, I and K are inputted

        -> The tangential point is calculated, and this arc will be determined.



    (iii) R and X, Z are inputted

        -> The tangential selection request window is displayed. Input number data 1 or 2, but in this case, select a tangential point to define a minor arc.

(e) When the preceding figure block "arc" is pending (for which the start point has been determined and only the R is to be inputted), and TANGNT is specified
In this case, the directions of two arcs must be the same.
  (i)  R, X and Z are inputted
      -> The tangential selection request window is displayed. Input number data 1 or 2, but in this case, select a tangential point to define a minor arc. This arc will be determined.
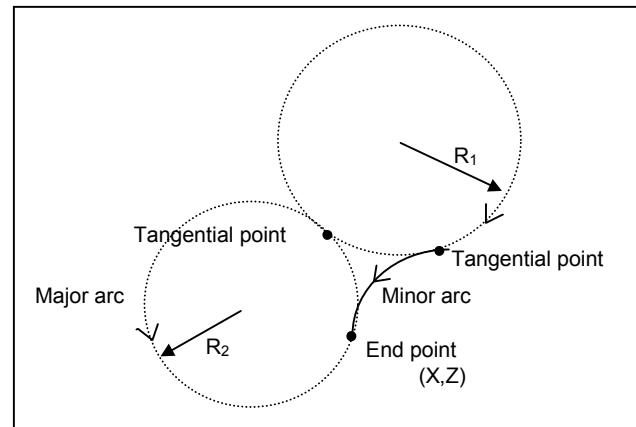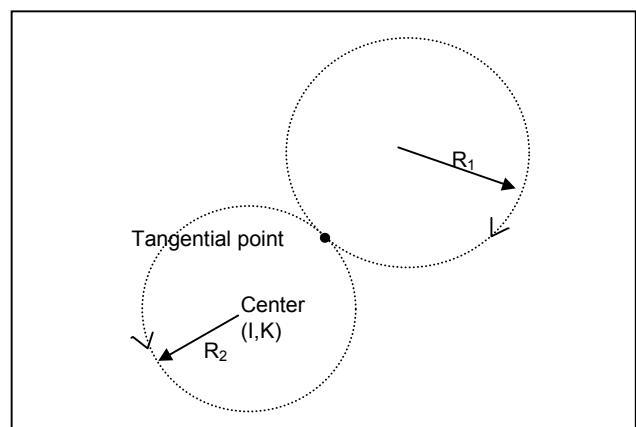


  (ii)  R, I and K are inputted
      -> A tangential point is calculated. This arc will be pending.

## (5) Line tangential to two arcs
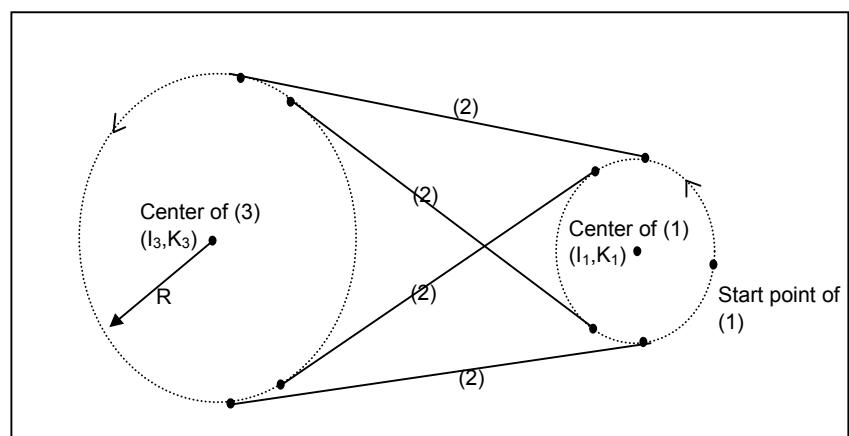


By inputting 3 successive figure as follows, Line (2) tangential to 2 arcs can be specified as the above drawing. The end points of (1) and (2) will be determined, and (3) will be pending.

Among the above 4 lines possibility, depending on the directions of 2 arcs, the line that makes a smooth connection to the arcs will be automatically selected.

    1st figure ARC(1) :

        I and K are inputted. (A start point is determined. This arc is pending.)

    2nd figure LINE(2) :

        No data.

    3rd figure ARC(3) :

        R, I and K are inputted.

## (6) Arc that contacts to both crossing lines and arcs

By inputting 3 successive figure as follows, Arc (2) tangential to 2 lines or arcs can be specified as the above drawing. The end points of (1) and (2) will be determined, and (3) will be pending.
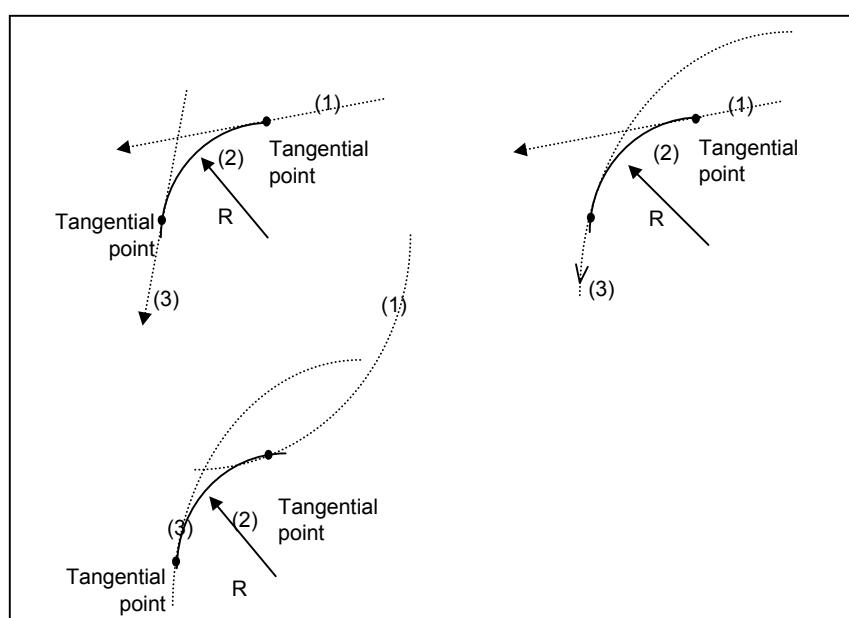
    1st figure LINE(1) or ARC(1) :
        Line that is pending (for which the start point has been determined), or
        Arc that is pending (for which the start point has been determined)
    2nd figure ARC(2) :
        Only R is inputted.
    3rd figure LINE(3) or ARC(3) :
        Line with A, X and Z, or
        Arc with R, I and K

## (7) Arc that contacts to uncrossing line and arc
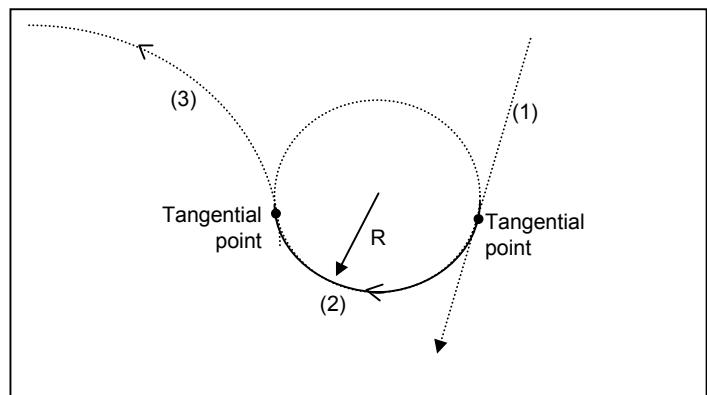


By inputting 3 successive figure as follows, Arc (2) tangential to line and arc , these 2 figures (1) and (3) do not cross, can be specified as the above drawing. The end points of (1) and (2) will be determined, and (3) will be pending.

Among the above 2 arcs possibility, the arc that makes a smooth connection to the line (1) and arc (3) will be automatically selected.

    1st figure LINE(1) :
        Line that is pending (for which the start point has been determined)
    2nd figure ARC(2) :
        Only R is inputted.
    3rd figure   ARC(3) :
        Arc with R, I and K

## (8) Arc that contacts to uncrossing 2 arcs
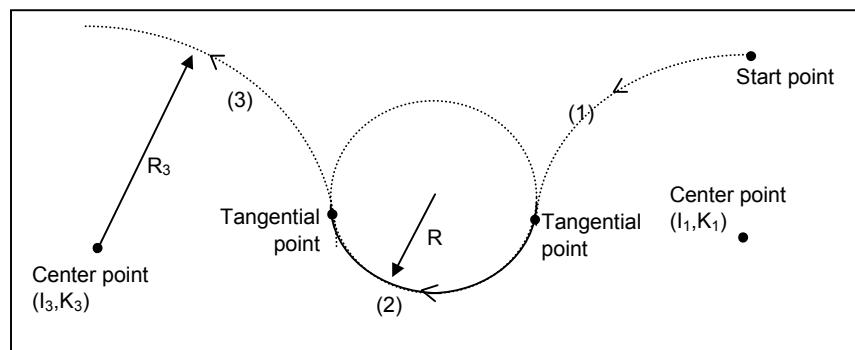


By inputting 3 successive figure as follows, Arc (2) tangential to 2 arcs , these 2 figures (1) and (3) do not cross, can be specified as the above drawing. The end points of (1) and (2) will be determined, and (3) will be pending.

Among 8 arcs possibility, depending on the directions of 2 arcs, the line that makes a smooth connection to the arcs will be automatically selected.

    1st figure ARC(1) :
        Arc with I and K, and it is pending (for which the start point has been determined)
    2nd figure ARC(2) :
        No data.
    3rd figure   ARC(3) :
        Arc with R, I and K

# 2 BAR MACHINING (FINISHING)
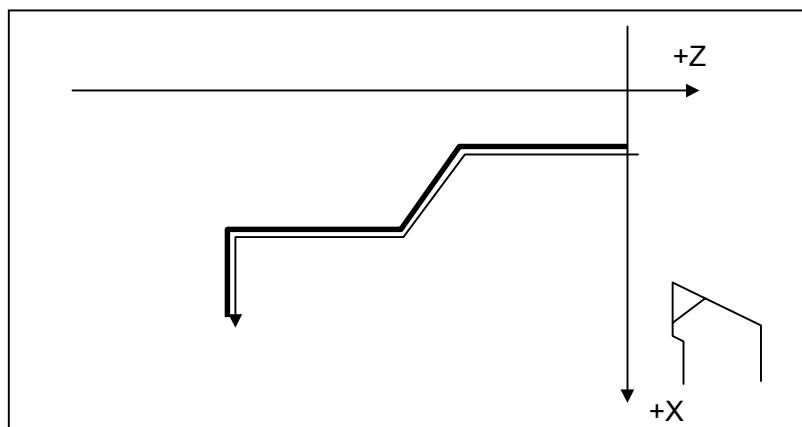
# *2.1*    DETAIL OF PROCESS DATA

## *2.1.1*    Process Data Window

```
┌─────────────────────────────────────────────┐
│ BAR MACHINING (FINISH)                        │
│                                               │
│ AREA        =        CUT ST Z    =            │
│ TOOL NUM.   =                                 │
│ FEED/REV.   =                                 │
│ S-SPD/MIN   =                                 │
│ S-DIRECT.   =                                 │
│ CUT ST X    =                                 │
└─────────────────────────────────────────────┘
```

**AREA**          Select from [OUTER]/[INNER]/[FACE]/[OUTER REVERS] /[INNER/REVERS]/[FACE REVERS]
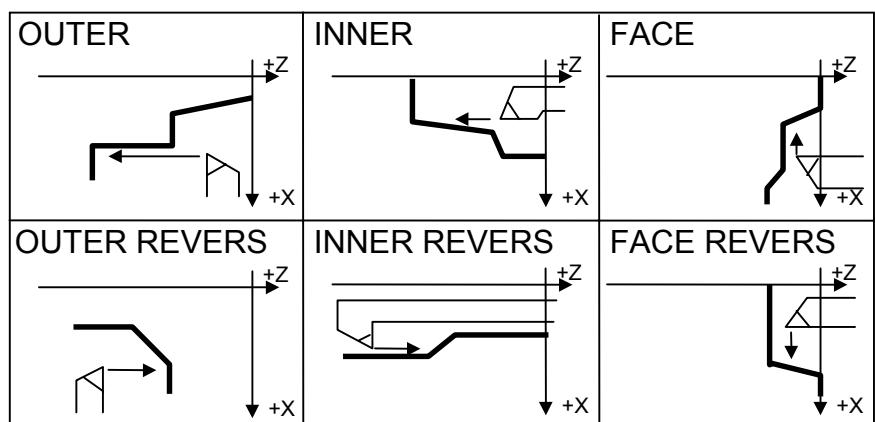


**TOOL NUM**      Number of tool used for finishing bar machining

**FEED/REV.**     Cutting feedrate, and it can be changed to FEED/MIN. by pushing soft-key [FEED/MIN.].

**S-SPD/MIN**     Cutting speed, and it can be changed to REV/MIN. by pushing soft-key [REV/MIN.].

**S-DIRECT**      Spindle rotating direction. Select from [NORMAL] and [REVERS].

**CUT POS X**     X coordinate of cutting start position.

**CUT POS Z**     Z coordinate of cutting start position.

## *2.1.2*    **Supplemental Process Data in a Pop-up Window**

```
EDGE ANGL =            SP.GEAR=
NOSE ANGL =
RESE AMNT =
APPROACHX    =
APPROACHZ    =
COOLANT    =
```

These supplemental process data are automatically set always, so in ordinary case, an operator need not to call this pop-up window excepting the case of checking these data.

**EDGE ANGL**    Angle of the cutting edge, normally it is set automatically to 90 degrees. In the case of that it was set to the value less than 90 degrees, cutting edge compensation will be done automatically. Parameter No.9033 value is automatically set into this data at initially inputting this process.
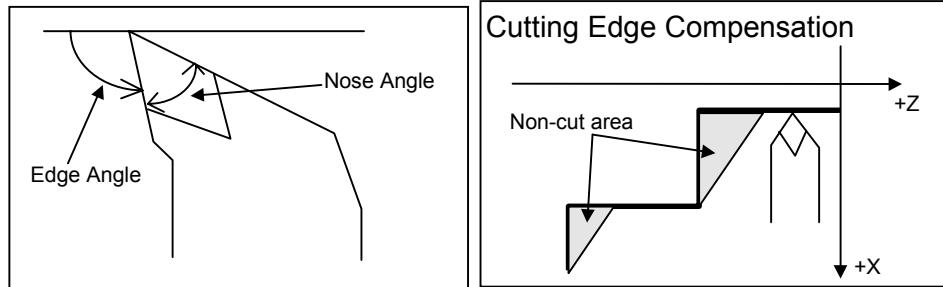


Cutting Edge Compensation

**NOSE ANGLE**    Angle of the tool tip, its default value can be set by parameter. In the case of that pocket figure was inputted as contour figure, cutting back compensation will be done automatically. Parameter No.9034 value is automatically set into this data at initially inputting this process.



Cutting Back Compensation

- 117 -

**RESE.AMNT**    Travel amount in X-axis(OUTER/INNER : diameter) or
                Z-axis(FACE : radius) for retraction after cutting.



Release Amount

**APROCH X**    X coordinate of approaching intermediate point.
**APROCH Z**    Z coordinate of approaching intermediate point.
**COOLANT**     Coolant specification at playback. Choose from the
                following soft-key menu. Default value can be set by
                parameter.
        [ON]          (M8)
        [OFF]         (M9)
**SPDL GEAR**   Spindle gear at playback. Choose from the following
                soft-key menu.
        [AUTO]        default
        [LOW]
        [MEDIUM 1]
        [MEDIUM 2]
        [HIGH]

# *2.2*    DETAIL OF FIGURE DATA

In Bar machining (Finishing), up to 30 figures can be specified for one process, but there are cases that one arc may be counted as 2 figures.

As to contour figure data, they are similar to Bar machining (Roughing), so refer to Chapter 1. But only "OVERRIDE" and "DIRECTION" data are added as follows.

## (1) LINE / ARC / CHAMFER / ROUND

Example of LINE)
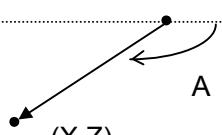
| CONTOUR FORM (LINE) | |
|---|---|
| END POINT X    X= | |
| END POINT Z    Z= | |
| ANGLE          A= | |
| OVERRIDE       P= | |

| | |
|---|---|
| **END POINT X/Z** | Similar to Bar machining (Roughing) |
| **ANGLE** | Similar to Bar machining (Roughing) |
| **OVERRIDE** | This datum is used for only Bar Finishing, an operator can set a feedrate used for finishing this figure only by selecting an override from 10% - 150% by 10% unit. And 100% means that the feedrate inputted in a process data window. |
| | By pushing [+10%] or [-10%], this data can be changed. |

## *2.3* NECKING FIGURE INPUTTING OF DIN509-E, DIN509-F, AND DIN76

### (1) Selecting a necking figure menu

During inputting figures for bar machining, the following menu of figure type appears in soft-keys.

| LINE | ARC ↵ | ARC ↙ | ROUND | CHAMF | TANGNT | | | | CALC. |

By pushing a right end soft-key, menus for necking figure are displayed.

| | | | | | | | DIN509 | 509-F | DIN76 |

### (2) DIN509-E

By pushing a [DIN509], grinding necking of DIN509-E type can be selected.

CONTOUR FORM (DIN509-E)

WIDTH      WT=
DEPTH      DT=
RADIUS     R=
OVERRIDE   P=

WT

R   R   DT
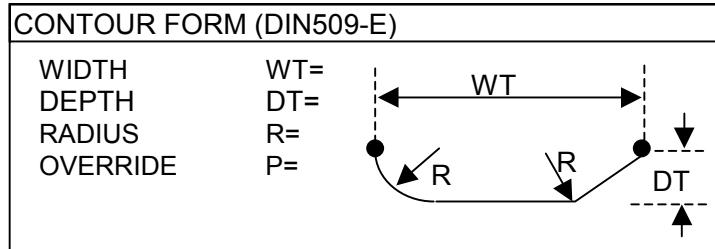
**WIDTH**　　　　　Width of necking figure DIN509-E
**DEPTH**　　　　　Depth of necking figure.
**RADIUS**　　　　　Corner-R radius of necking figure.

By pushing a **INSERT** after inputting the above data according to the DIN509-E standard, this necking figure is converted to the group of necessary figures, lines and corner-R, and these generated figures are displayed in a figure window.

### (3) DIN76

By pushing a [DIN76], threading necking of DIN76 type can be selected.

CONTOUR FORM (DIN76)

WIDTH      WT=
DEPTH      DT=
OVERRIDE   P=

WT

0.6DT   DT

**WIDTH**　　　　　Width of necking figure DIN76
**DEPTH**　　　　　Depth of necking figure.

By pushing a **INSERT** after inputting the above data according to the DIN76 standard, this necking figure is converted to the group of necessary figures, lines and corner-R, and these generated figures are displayed in a figure window.

## (4) DIN509-F

By pushing a [DIN509], neckingfor grinding of DIN509-E type can be selected.



CONTOUR FORM (DIN509-F)

| | |
|---|---|
| WIDTH | WT= |
| DEPTH | DT= |
| RADIUS | R= |
| RELIEF AMNT | W1= |

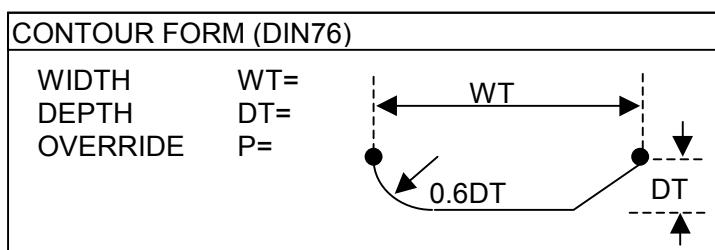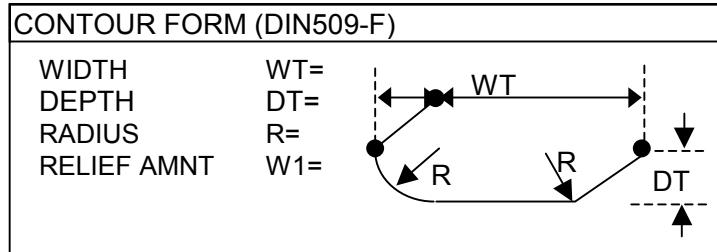**WIDTH**        Width of necking figure DIN509-F
**DEPTH**        Depth of necking figure.
**RADIUS**       Corner-R radius of necking figure.

By pushing a **INSERT** after inputting the above data according to the DIN509-F standard, this necking figure is converted to the group of necessary figures, lines and corner-R, and these generated figures are displayed in a figure window.

---

**NOTE**
1  When inputting this necking figure, the end point coordinate (X, Z) of the figure just prior to it must be decided already. If it remains the pending status, warning message is displayed when inserting this necking figure.
2  After inputting a necking figure and being converted to several lines and corner-R figures, only figure data of them can be modified.

---

# 3 THREADING



Chamfering is possible only at 45degrees
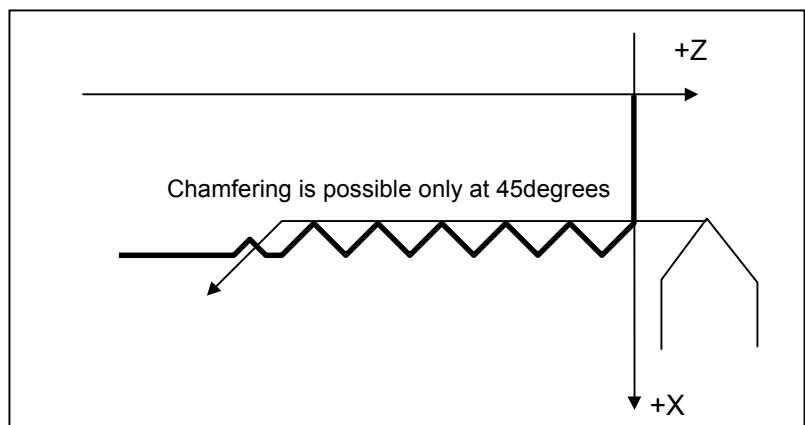
# *3.1*    **DETAIL OF PROCESS DATA**

## *3.1.1*    **Process Data Window**

### (1) GENERAL THREAD

TYPE = [GENERAL]

```
THREADING

AREA        =        CHAMFERNG   =
TOOL NUM.   =        CUT AMUNT   =
S -SPD/MIN  =        TYPE        =GENERL
S -DIRECT.  =        METHOD      =
CUT ST X    =        ANGLE       =
CUT ST Z    =        LEAD        =
```

AREA            Select from [OUTER]/[INNER]



TOOL NUM        Number of tool used for threading

S-SPD/MIN       Cutting speed, and it can be changed to REV/MIN. by
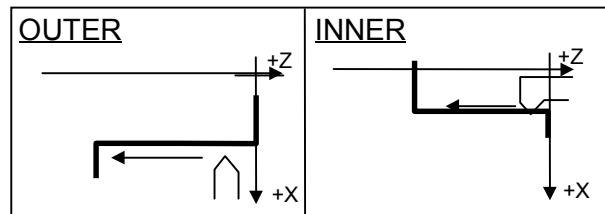                pushing soft-key [REV/MIN.].

S-DIRECT        Spindle rotating direction. Select from [NORMAL] and
                [REVERS].

CHAMFERNG       Select whether to perform chamfering by [ON] / [OFF]
                [ON] :                                          M-
                code for chamfer on (parameter No.9836) is outputted.
                For the chamfering length, parameter No.5130 is used.
                [OFF] :                                         M-
                code for chamfer off (parameter No.9837) is outputted.

CUT AMUNT       Depth of cut for the first pass.
                By pushing [CUT TIMES], the number of cuts can be
                selected.
                [CUT AMUNT] : The number of cuts is determined from
                the depth of cut and the cutting method.
                [CUT TIMES] : When the number of cuts is entered,
                threading is performed according to this value. This
                number of cuts includes the spark-out motions.

---

**NOTE**
1  The maximum number of cuts is 999.
   When zigzag cutting is selected, input an even
   number of cuts, excluding the numbers of spark-out
   motions. If an odd number is inputted, one extra
   cutting is done.

---

**NOTE**
2   Depending on the minimum depth of cut
    (parameter No.9833), inputted number may be
    greater than actually needed. In such a case, the
    actual number is reduced to a value below the
    inputted number.
    Further more, inputted number may differ slightly
    from the actual number because of a degree of
    error in the calculation.

**METHOD**          Select from [SINGLE-AMNT], [BOTH-AMNT], [STRGHT-
                    AMNT], [SINGLE-DPTH], [BOTH-DPTH] and [STRGHT-
                    DPTH].
                      [SINGLE-AMNT]          Single edge cutting, constant
                                             cutting amount.
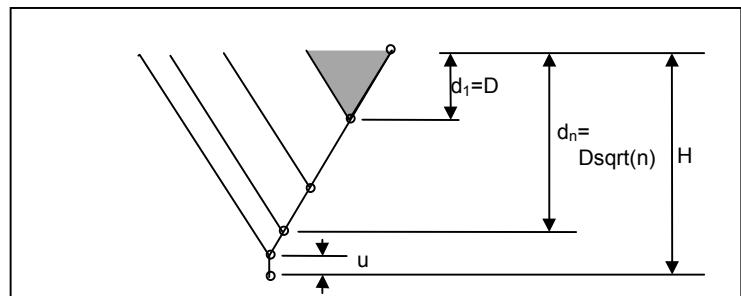


H = Thread height    D = Depth of cut
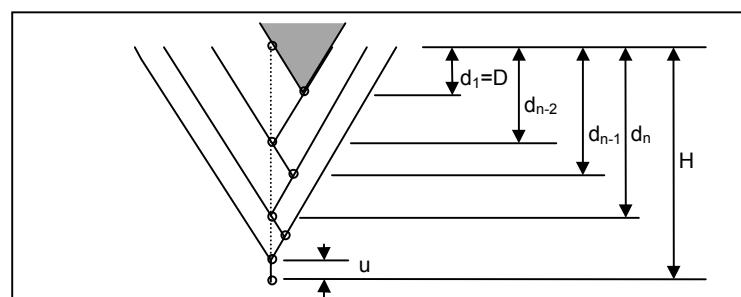u = Finishing amount

                    [BOTH-AMNT]          :Zigzag cutting, constant cutting
                                          amount



$d_{n-2} = D \sqrt{n-2}$
$d_{n-1} = (D(\sqrt{n-2} + \sqrt{n}))/2$
$d_n = D \sqrt{n}$

**NOTE**
    If the depth of cut becomes smaller than the
    minimum depth of cut (parameter No. 9833), the
    depth of cut is fixed to the minimum depth of cut.

[STRGHT-AMNT]          Straight cutting, constant
                       cutting amount.
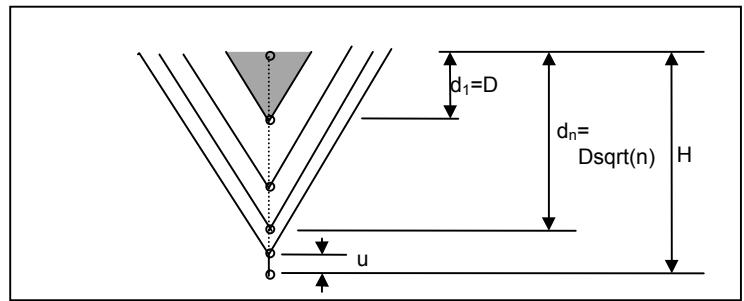


[SINGLE-DPTH]          Single edge cutting, constant
                       cutting depth.
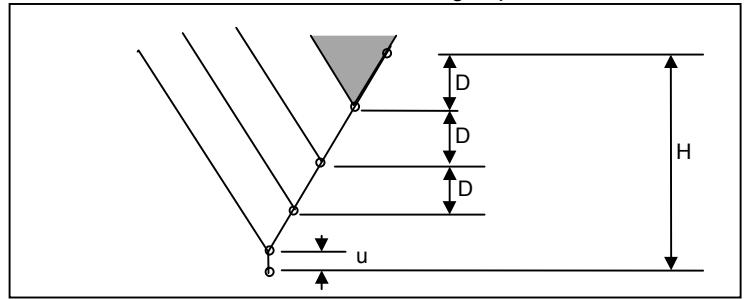


[BOTH-DPTH]            Zigzag cutting, constant cutting
                       depth.
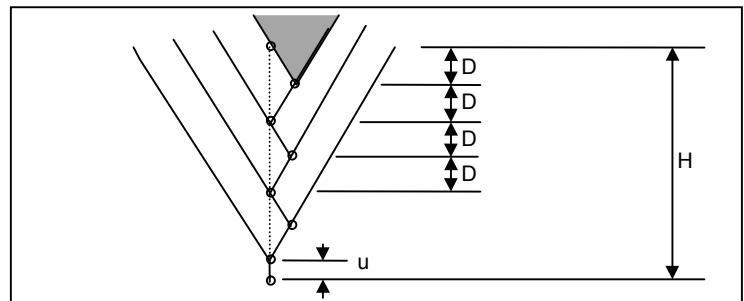


[STRGHT-DPTH]          Straight cutting, constant cutting
                       amount.
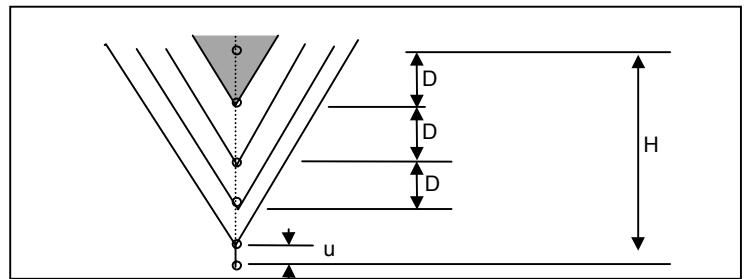


**ANGLE**              Tool tip angle used for only GENERAL thread.
                       Parameter No.9835 value is automatically set. If it is 0,
                       60 is set.
**LEAD**               Thread lead. The least input increment is 0.0001mm or
                       0.000001inch.

## (2) METRIC THREAD

TYPE = [METRIC]

```
THREADING
AREA        =           CHAMFERNG =
TOOL NUM  =           CUT AMUNT  =
S-SPD/MIN  =           TYPE         =METRIC
S-DIRECT   =           METHOD     =
CUT ST X   =           LEAD         =
CUT   ST Z  =
```

Metric thread is performed according to the metric standard. Only straight threads can be made. All the data excepting "ANGLE" are same with the general thread. A thread angle of 60degrees is used for it.

## (3) UNIFIED THREAD

TYPE = [UNIFIED]

```
THREADING
AREA          =           CHAMFERNG   =
TOOL NUM    =           CUT AMUNT    =
S-SPD/MIN    =           TYPE           =UNIFI
S-DIRECT     =           METHOD       =
CUT ST X     =           NUM/INCH      =
CUT ST Z      =
```

Unified thread is performed according to the unified thread standard. Only straight threads can be made. All the data excepting "THRD/INCH" (replaced with "LEAD") are same with the general thread. A thread angle of 60degrees is used for it.

**NUM/INCH**        Number of thread per inch. The least input increment is 0.1.

## (4) PT THREAD

TYPE = [PIPE-T]

```
THREADING
AREA          =           CHAMFERNG =
TOOL NUM    =           CUT AMUNT  =
S-SPD/MIN    =           TYPE         =PIPE-T
S-DIRECT     =           METHOD     =
CUT ST X     =           NUM/INCH   =
CUT ST Z      =
```

PT thread is performed according to the pipe-taper thread standard. Only tapered threads with a slant angle of 1.7899 degrees can be made. All the data are same with the unified type thread. A thread angle of 55degrees is used for it.

When an external tapered (male) thread figure, the end face of the workpiece represents the minimum diameter section. With an internal tapered (female) thread figure, the end face of the workpiece represents the maximum diameter section.

## (5) PF THREAD

TYPE = [PIPE-F]

```
THREADING
AREA          =          CHAMFERNG     =
TOOL NUM      =          CUT AMUNT     =
S-SPD/MIN     =          TYPE          =PIPE-F
S-DIRECT      =          METHOD        =
CUT ST X      =          NUM/INCH      =
CUT ST Z      =
```

PF thread is performed according to the PF thread standard. Only straight threads can be made. All the data are same with the PT type thread. A thread angle of 55degrees is used for it.

> **NOTE**
> All the thread of metric and unified are shaped differently depending on whether they are external threads or internal threads, however, the above functions regard figure data as being identical.

## *3.1.2*    **Supplemental Process Data in a Pop-up Window**

```
MLTI          =        APPROARCHX=
HIGHT         =        APPROARCHZ=
SPARK OUT  =        COOLANT     =
FINISHING   =        SP.GEAR      =
CLEAR   X  =
CLEAR   Z  =
```

These supplemental process data are commonly used for the above 5 types of threads, and automatically set always, so in ordinary case, an operator need not to call this pop-up window excepting the case of checking these data.

| | |
|---|---|
| **MULTI.** | Number of thread at multiple thread screw. Initial value is 1. |



Multiple thread screw
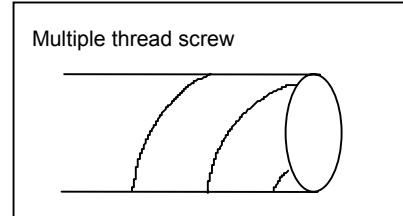
| | |
|---|---|
| **HIGHT** | Thread height. It is automatically calculated from the parameter below when the LEAD(THRD/INCH) is inputted.<br>1.General thread : Parameter No.9832<br>2.Metri/unified thread : Parameter No.9838<br>3.PT/PF thread : Parameter No.9839 |
| **SPARK OUT** | Number of finishing operations to be done. Initial value is 1. |
| **FINISHING** | Finishing amount. It is automatically preset by the value of parameter No.9834. |
| **CLEAR X** | Clearance along the X-axis in threading. It is automatically preset by the value of parameter No.9830. |
| **CLEAR Z** | Clearance along the Z-axis in threading, and it means the distance required for the spindle to reach a stable speed. It is automatically preset by the value of parameter No.9831. |
| **APPROACH X** | X coordinate of approaching intermediate point. |
| **APPROACH Z** | Z coordinate of approaching intermediate point. |
| **COOLANT** | Coolant specification at playback. Choose from the following soft-key menu. Default value is [ON].<br>[ON]      (M8)<br>[OFF]     (M9) |
| **SPDL GEAR** | Spindle gear at playback. Choose from the following soft-key menu. Default value is [AUTO].<br>[AUTO]<br>[LOW]<br>[MID1]<br>[MID2]<br>[HIGH] |

# *3.2*  **DETAIL OF FIGURE DATA**

## (1) GENERAL THREAD

```
START X     =
START Z     =
END X       =
END Z       =
```
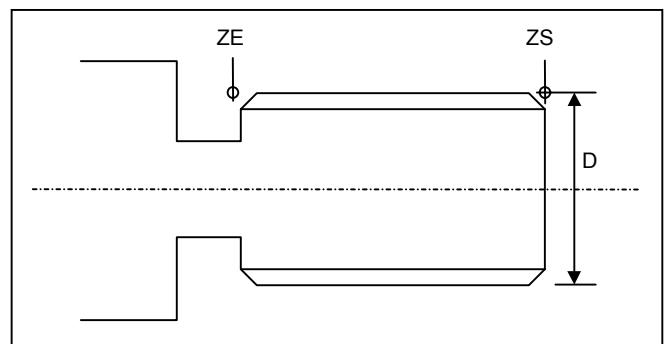
**START X / Z**      X and Z Coordinate of a threading start point.
**END X / Z**        X and Z Coordinate of a threading end point.
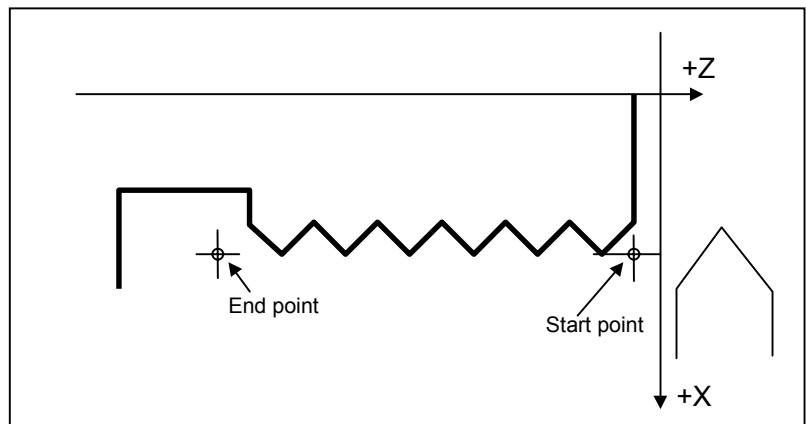


## (2) METRIC THREAD

```
THRED-DIA  =
START Z      =
START Z      =
```

**THRED-DIA**       Thread diameter(D).
**START Z**         Z Coordinate of a threading start point(ZS).
**END Z**           Z Coordinate of a threading end point(ZE).

## (3) UNIFIED THREAD

```
THRED-DIA  =
START Z    =
START Z    =
```

| | |
|---|---|
| **THRED-DIA** | Thread diameter(D). |
| **START Z** | Coordinate of a threading start point(ZS). |
| **END Z** | Coordinate of a threading end point(ZE). |



## (4) PT THREAD

```
START X    =
START Z    =
END Z      =
```

| | |
|---|---|
| **START X / Z** | X and Z Coordinate of a threading start point(ZS). |
| **END Z** | Z Coordinate of a threading end point(ZE). |
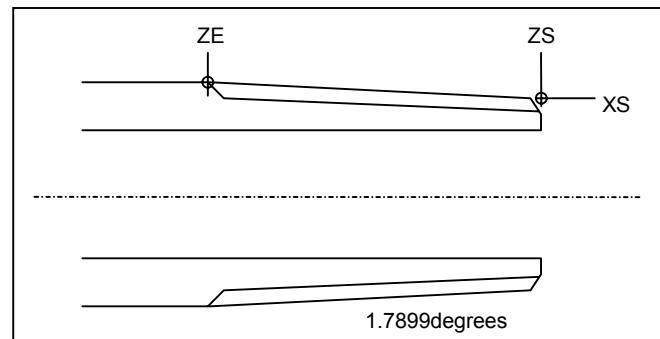
## (5) PF THREAD

```
THRED-DIA  =
START Z    =
END Z      =
```

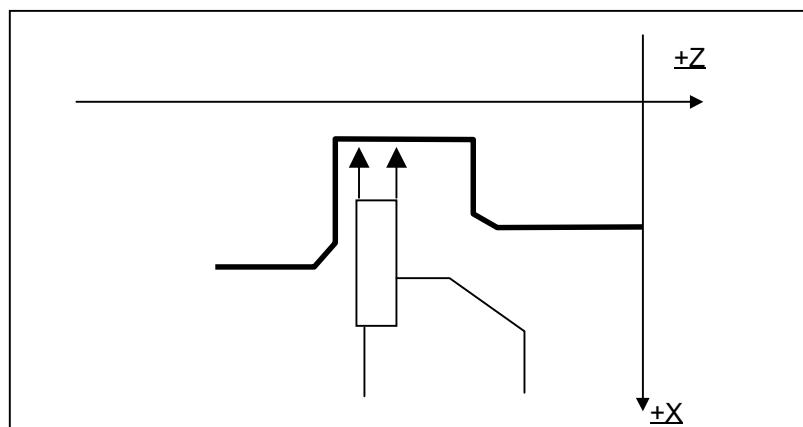| | |
|---|---|
| **THREAD-DIA** | Thread diameter(D). |
| **START Z** | Coordinate of a threading start point(ZS). |
| **END Z** | Coordinate of a threading end point(ZE). |
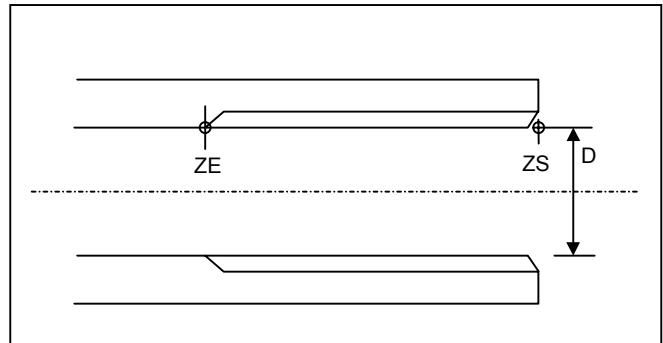
# 4 GROOVING

# *4.1*    **DETAIL OF PROCESS DATA**

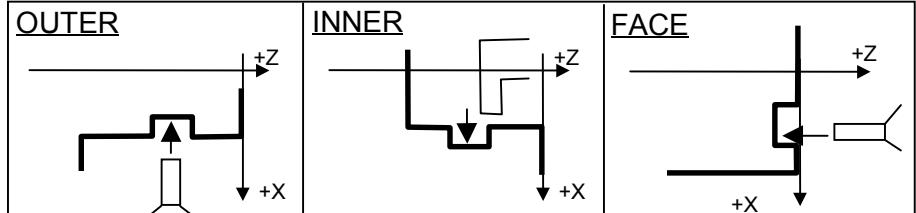## *4.1.1*    **Process Data Window**

```
GROOVING
AREA          =          CUT ST Z      =
TOOL NUM.     =          TOOL WIDT     =
FEED/REV      =          STEP AMNT     =
S-SPD/MIN     =          DWELL         =
S -DIRECT.    =
CUT ST X      =
```

| | |
|---|---|
| **AREA** | Select from [OUTER]/[INNER]/[FACE] |



| | |
|---|---|
| **TOOL NUM** | Number of tool used for grooving |
| **FEED/REV.** | Cutting feedrate, and it can be changed to FEED/MIN. by pushing soft-key [FEED/MIN.]. |
| **S-SPD/MIN** | Cutting speed, and it can be changed to REV/MIN. by pushing soft-key [REV/MIN.]. |
| **S-DIRECT** | Spindle rotating direction. Select from [NORMAL] and [REVERS]. |
| **CUT ST X** | X coordinate of an point before groove part machining is started. |
| **CUT ST Z** | Z coordinate of an point before groove part machining is started. |
| **TOOL WIDTH** | Width of grooving tool at cutting edge. |
| **STEP AMNT** | Depth of each cutting motion. |
| **DWELL** | Dwell at the bottom of a groove in roughing. |

In normal grooving, semi-finish machining is carried out. But, by setting the parameter 9772<REN> to 1, it is not carried out. About the details of this parameter, see Appendix A "Parameters."

## *4.1.2* **Supplemental Process Data in a Pop-up Window**

```
APPROARCHX   =
APPROARCHZ   =
COOLANT      =
SP.GEAR      =
```

These supplemental process data are automatically set always, so in ordinary case, an operator need not to call this pop-up window excepting the case of checking these data.

| | |
|---|---|
| **APPROACH X** | X coordinate of approaching intermediate point. |
| **APPROACH Z** | Z coordinate of approaching intermediate point. |
| **COOLANT** | Coolant specification at playback. Choose from the following soft-key menu. Default value is [ON]. |
| |     [ON]      (M8) |
| |     [OFF]    (M9) |
| **SPDL GEAR** | Spindle gear at playback. Choose from the following soft-key menu. Default value is [AUTO]. |
| |     [AUTO] |
| |     [LOW] |
| |     [MID1] |
| |     [MID2] |
| |     [HIGH] |

# *4.2*    **DETAIL OF FIGURE DATA**

```
GRV.WIDTH  =
GRV.DEPTH  =
PITCH      =
GRV NUM    =
GRV ST X   =
GRV ST Z   =
GRV END X  =
CHAMFER    =
```

| | |
|---|---|
| **GRV.WIDTH** | Width of a groove to be machined. |
| **GRV.DEPTH** | Depth (radius value) of a groove to be machined. By pushing a [GRV.DIA], absolute X( Z in FACE) coordinate of the groove bottom can be specified. |
| **GRV ST X / Z** | X or Z coordinate of a point where grooving starts. |
| **GRV END X** | X coordinate of the end point of a groove. This data is necessary only at the depth of a groove is different. If no value is inputted into this data item, depth of start point end point becomes equal. At end face grooving, "GRV END Z" is displayed and Z coordinate must be inputted. |
| **CHAMFER** | Chamfer amount of a groove. |

(1)  OUTER
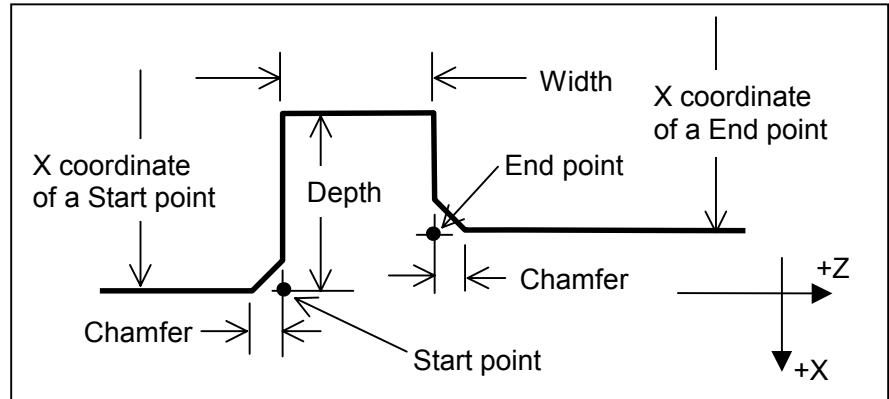


(2)  INNER
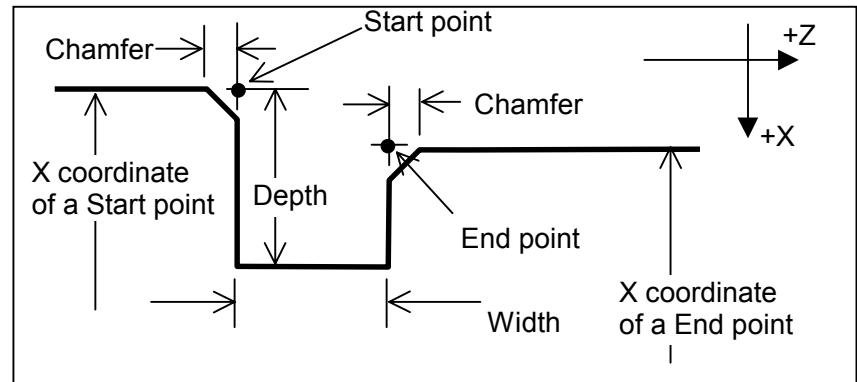
(3)  FACE



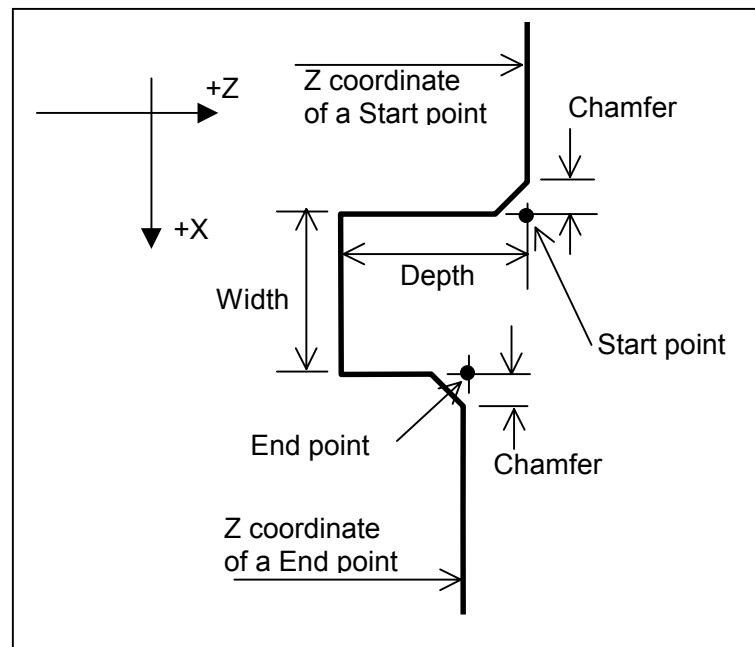**PITCH**          Distance between grooves when grooves of the same figure are machined at regular intervals.
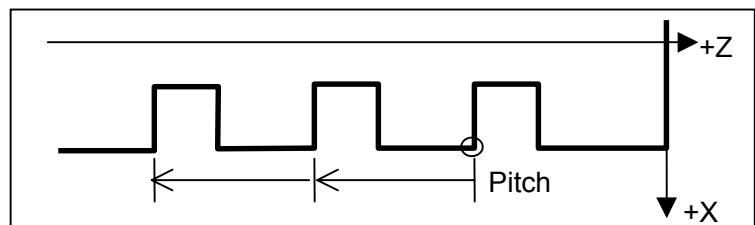
1)  PITCH < 0



2)  PITCH > 0
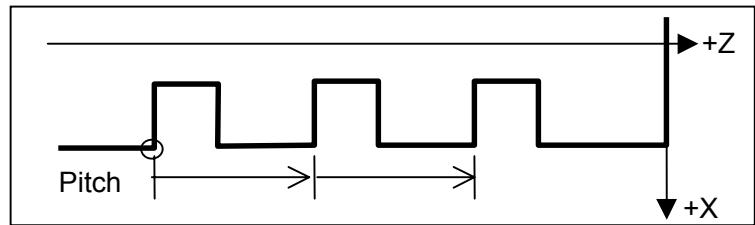


**NUMBER**        Number of grooves when more than 0ne groove are to be machined at regular intervals. 1 is set as the default value.

# 5    TRAPEZOIDAL GROOVING (ROUGHING AND FINISHING)

# *5.1*    DETAIL OF PROCESS DATA

## *5.1.1*    Process Data Window for Roughing

```
GROOV TRAPEZOID ROUGH
AREA        =      STEP AMNT  =
TOOL NUM.   =      DWELL      =
FEED/REV    =      FINISH X   =
S-SPD/MIN   =      FINISH Z   =
S -DIRECT.  =      PITCH      =
TOOL WIDT   =      GRV NUM    =
```
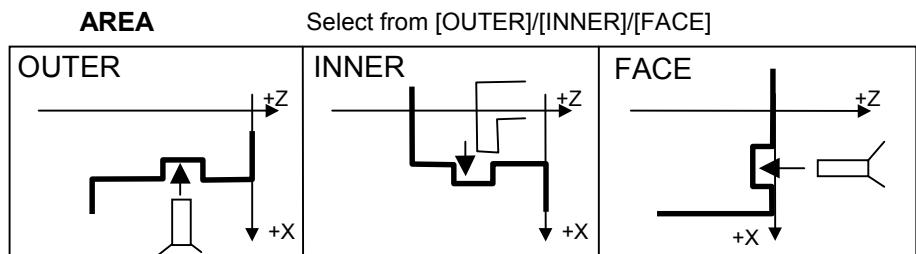
**AREA**                    Select from [OUTER]/[INNER]/[FACE]



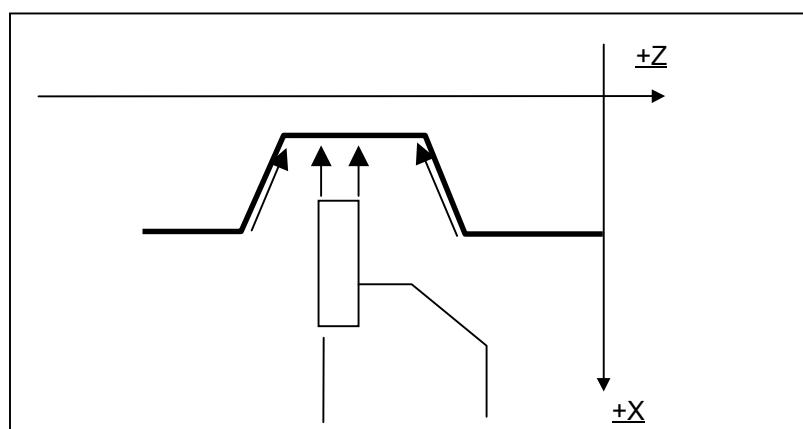| **TOOL NUM** | Number of tool used for grooving |
| **FEED/REV.** | Cutting feedrate, and it can be changed to FEED/MIN. by pushing soft-key [FEED/MIN.]. |
| **S-SPD/MIN** | Cutting speed, and it can be changed to REV/MIN. by pushing soft-key [REV/MIN.]. |
| **TOOL WIDTH** | Width of grooving tool at cutting edge. |
| **STEP AMNT** | Depth of each cutting motion. |
| **DWELL** | Dwell at the bottom of a groove in roughing. |
| **FINISH X** | X direction amount of finishing. |
| **FINISH Z** | Z direction amount of finishing. |
| **PITCH** | Distance between grooves when grooves of the same figure are machined at regular intervals. |
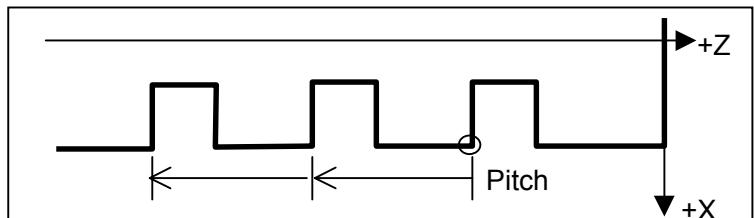
1)    PITCH < 0



2)    PITCH > 0
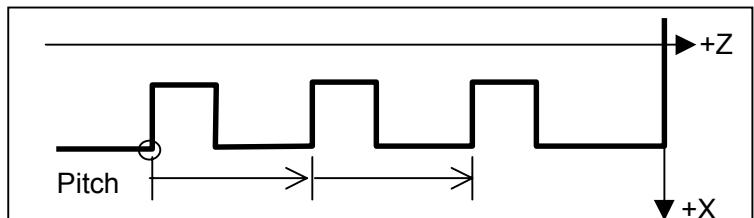


**NUMBER**                  Number of grooves when more than 0ne groove are to be machined at regular intervals.

## *5.1.2*    **Process Data Window for Finishing**

```
GROOV TRAPEZOID FINISH
AREA          =          CUT ST Z    =
TOOL NUM.  =          TOOL WIDT  =
FEED/REV    =          PITCH        =
S-SPD/MIN  =          GRV NUM    =
S -DIRECT.   =
CUT ST X     =
```

**CUT ST X**         X coordinate of an point before groove part machining is started. In roughing process, this data item is displayed in a detailed data window.

**CUT ST Z**         Z coordinate of an point before groove part machining is started. In roughing process, this data item is displayed in a detailed data window.

As to the data item other than the above, they are similar to roughing.

# *5.1.3*  **Supplemental Process Data in a Pop-up Window for Roughing**

```
CUT ST X      =
CUT ST Z      =
APPROARCHX=
APPROARCHZ=
COOLANT     =
SP.GEAR      =
```

These supplemental process data are automatically set always, so in ordinary case, an operator need not to call this pop-up window excepting the case of checking these data.

| | |
|---|---|
| **CUT ST X** | X coordinate of a point before groove part machining is started. |
| **CUT ST Z** | Z coordinate of a point before groove part machining is started. |
| **APPROACH X** | X coordinate of approaching intermediate point. |
| **APPROACH Z** | Z coordinate of approaching intermediate point. |
| **COOLANT** | Coolant specification at playback. Choose from the following soft-key menu. Default value is [ON]. |
| |   [ON]  (M8) |
| |   [OFF]  (M9) |
| **SPDL GEAR** | Spindle gear at playback. Choose from the following soft-key menu. Default value is [AUTO]. |
| |   [AUTO] |
| |   [LOW] |
| |   [MID1] |
| |   [MID2] |
| |   [HIGH] |

## *5.1.4*    **Supplemental Process Data in a Pop-up Window for Finishing**

```
APPROARCHX=
APPROARCHZ=
COOLANT    =
SP.GEAR    =
```

Excepting that cutting start point data are moved to the process window, other data item are similar to roughing.

# *5.2*     **DETAIL OF FIGURE DATA**

```
START X      =        POINT 3 X     =
START Z      =        POINT 3 Z     =
POINT 1 X    =        CORNER R      =
POINT 1 Z    =        POINT 4 X     =
CORNER R     =        POINT 4 Z     =
POINT 2 X    =        CORNER R      =
POINT 2 Z    =        END X         =
CORNER R     =        END Z         =
```

**START X / Z**          Coordinate of a point where grooving starts.
**POINT 1 to 4 X / Z**   X and Z coordinates of the corners of a trapezoidal groove shown below.

> **NOTE**
> The bottom figure of a trapezoidal groove must be parallel to Z-axis (OUTER and INNER groove) or X-axis (FACE groove). In other word, when OUTER or INNER groove is specified, the X coordinate of point 2 and 3 must be same, and when FACE groove is specified, the Z coordinate of them must be same.

**CORNER R 1 to 4**      Corner-R radius of each point 1 to 4. These data items can be changed to chamfering figure by pushing a [CHAMF].
**CHAMFER 1 to 4**       Chamfer amount for each point 1 to 4.
**END X / Z**            X and Z coordinates of the end point of a trapezoidal groove shown below.

(1)  OUTER
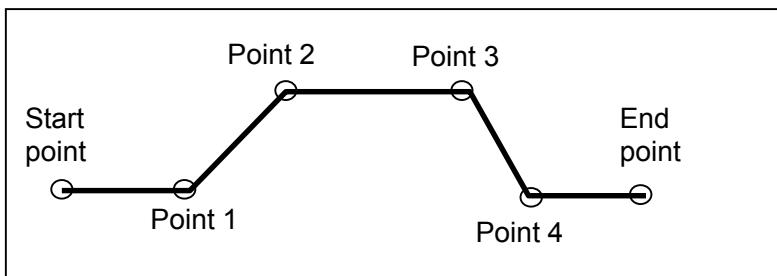


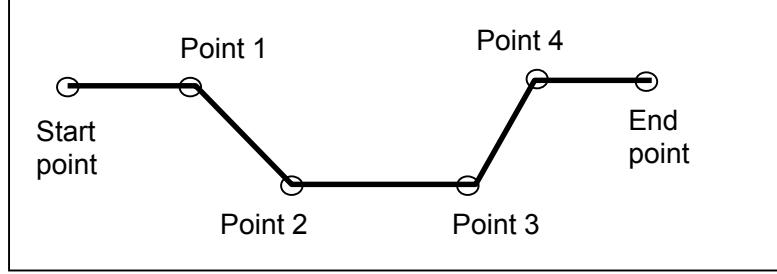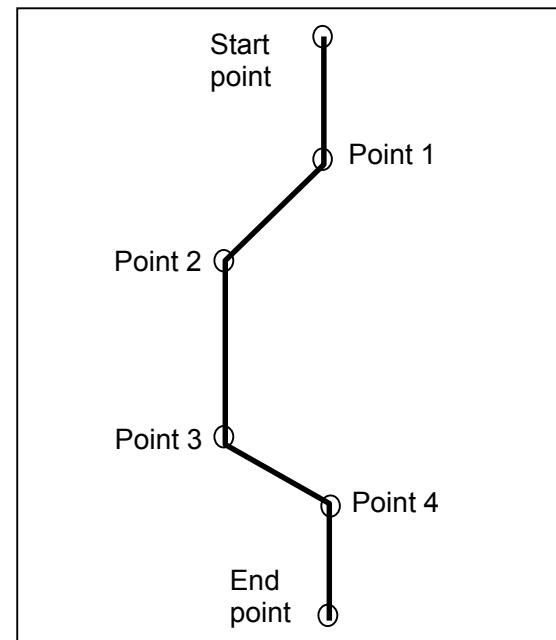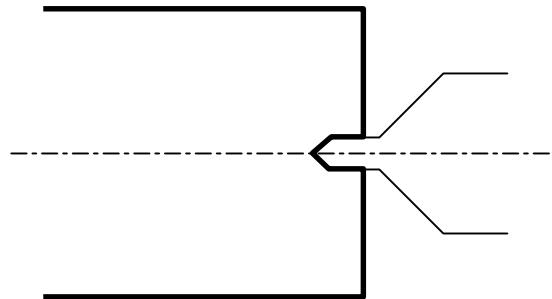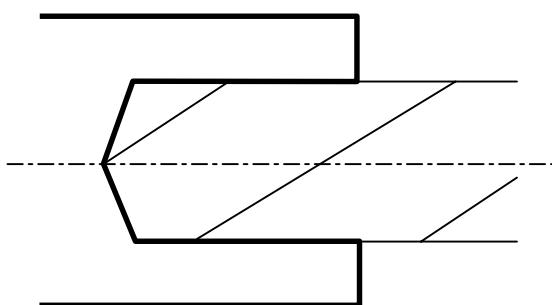(2)  INNER

(3)  FACE

# 6

# CENTER DRILLING / DRILLING / REAMING / BORING / TAPPING
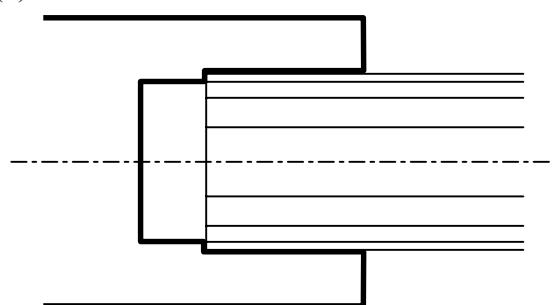
(1)   CENTER DRILLING

(2)   DRILLING

(3)   REAMING

(4)   BORING
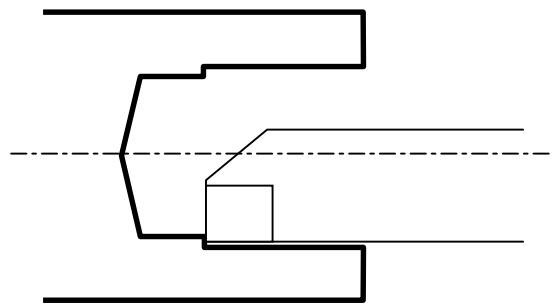
(5)   TAPPING
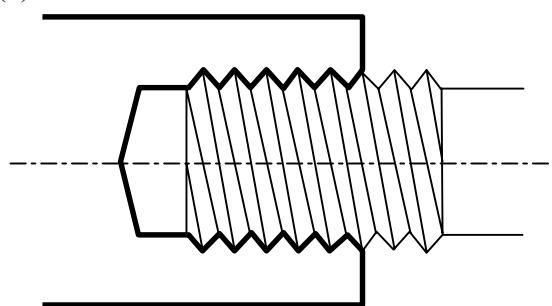
# *6.1* DETAIL OF PROCESS DATA

## *6.1.1* Process Data Window

### (1) CENTER DRILLING

```
CENTER DRILLING
TOOL NUM   =        HOLE DIA    =
FEED/MIN   =        START Z     =
REV/MIN    =        DEPTH       =
S-DIRECT   =        DWELL       =
CUT ST X   =
CUT ST Z   =
```

| | |
|---|---|
| **TOOL NUM** | Number of tool used for center drilling |
| **FEED/MIN** | Feed amount in center drilling, and it can be changed to FEED/REV by pushing a [FEED/REV]. |
| **REV/MIN** | Spindle rotating speed. |
| **S-DIRECT** | Spindle rotating direction. Select from [NORMAL] and [REVERS]. |
| **CUT ST X** | X-coordinate of a point where machining starts |
| **CUT ST Z** | Z-coordinate of a point where machining starts |
| **HOLE DIA** | Hole diameter or tool nominal diameter. This datum is used for only animated drawing. |
| **START Z** | Absolute Z-coordinate of hole start point |
| **DEPTH** | Depth of hole. By pushing [END Z], absolute Z-coordinate of hole end point can be selected. |
| **DWELL** | Dwell time at hole end point |

### (2) DRILLING

```
DRILLING
TOOL NUM   =        HOLE DIA    =
FEED/MIN   =        START Z     =
REV/MIN    =        DEPTH       =
S-DIRECT   =        CUT DEPTH   =
CUT ST X   =        DWELL       =
CUT ST Z   =        PATTERN     =
```

| | |
|---|---|
| **TOOL NUM** | Number of tool used for drilling |
| **FEED/MIN** | Feed amount in drilling, and it can be changed to FEED/REV by pushing a [FEED/REV]. |
| **REV/MIN** | Spindle rotating speed. |
| **S-DIRECT** | Spindle rotating direction. Select from [NORMAL] and [REVERS]. |
| **CUT ST X** | X-coordinate of a point where machining starts |
| **CUT ST Z** | Z-coordinate of a point where machining starts |
| **HOLE DIA** | Hole diameter or tool nominal diameter. This datum is used for only animated drawing. |

**START Z**        Absolute Z-coordinate of hole start point

**DEPTH**        Depth of hole.

By pushing [END Z], absolute Z-coordinate of hole end point can be selected.

**CUT-DPTH**        Depth of first cut in peck drilling [DEEP HO;E] or high-speed peck drilling [RAPID DEEP] (parameter 9853 : DRILCF)

**DWELL**        Dwell time at hole end point

**PATTERN**        Drilling type

Select from [DRILL], [DEEP HOLE] and [RAPID DEEP].

1.[DRILL]



Hole end point      Hole start point      Cutting start point

2.[DEEP HOLE]



Hole end point

Return amount amount      Cutting start point

Hole start point

3.[RAPID DEEP]



Hole end point

Return amount      Cutting start point

Hole start point
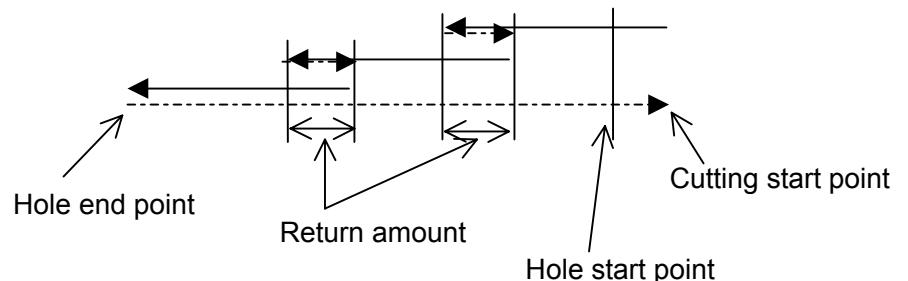
> **NOTE**
> The above return amount can be set in a detail data window.

- 147 -

## (3) REAMING

```
REAMING
TOOL NUM   =        HOLE DIA    =
FEED/MIN   =        START Z     =
REV/MIN    =        DEPTH       =
S-DIRECT   =        CHAMF-LNG   =
CUT ST X   =        DWELL       =
CUT ST Z   =
```

| | |
|---|---|
| **TOOL NO** | Number of tool used for reaming |
| **FEED/MIN** | Feed amount in reaming, and it can be changed to FEED/REV by pushing a [FEED/REV]. |
| **REV/MIN** | Spindle rotating speed. |
| **S-DIRECT** | Spindle rotating direction. Select from [NORMAL] and [REVERS]. |
| **HOLE DIA** | Hole diameter or tool nominal diameter. |
| | This datum is used for only animated drawing. |
| **START Z** | Absolute Z-coordinate of hole start point |
| **DEPTH** | Depth of hole. |
| | By pushing [END Z], absolute Z-coordinate of hole end point can be selected. |
| **CHAMF-LNG** | Cutting-lip length of a reamer at cutting start (parameter 9054 : RIMKLN) |



CHAMF-LNG
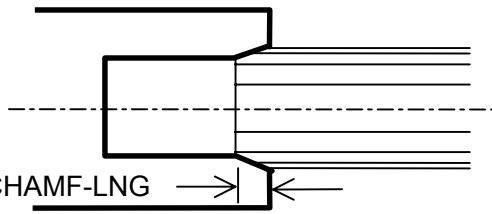
| | |
|---|---|
| **DWELL** | Dwell time at hole end point |
| **CUT ST X** | X-coordinate of a point where machining starts |
| **CUT ST Z** | Z-coordinate of a point where machining starts |

## (4) BORING

```
BORING
TOOL NUM   =        HOLE DIA    =
FEED/MIN   =        START Z     =
REV/MIN    =        DEPTH       =
S-DIRECT   =        SHIFT       =
CUT ST X   =        DWELL       =
CUT ST Z   =
```

| | |
|---|---|
| **TOOL NUM** | Number of tool used for boring |
| **FEED/MIN** | Feed amount in boring, and it can be changed to FEED/REV by pushing a [FEED/REV]. |
| **REV/MIN** | Spindle rotating speed. |
| **S-DIRECT** | Spindle rotating direction. Select from [NORMAL] and [REVERS]. |
| **HOLE DIA** | Hole diameter of boring cutting |

- 148 -

**START Z**      Absolute Z-coordinate of hole start point

**DEPTH**        Depth of hole.

By pushing [END Z], absolute Z-coordinate of hole end point can be selected.
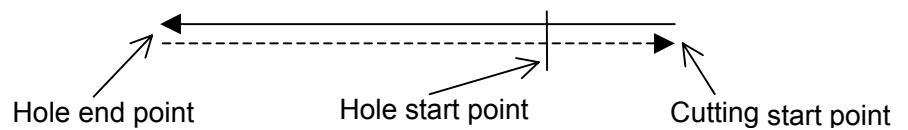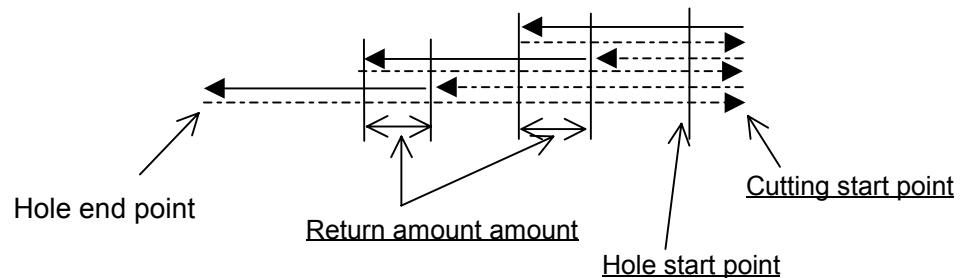
**DWELL**        Dwell time at hole end point

**SHIFT**        Shift amount for relief motion in boring (parameter 9858 : BRSHFT)



**CUT ST X**     X-coordinate of a point where machining starts

**CUT ST Z**     Z-coordinate of a point where machining starts

## (5) TAPPING

```
TAPPING
TOOL NO.    =       HOLE DIA    =
FEED/MIN    =       START Z     =
REV/MIN     =       DEPTH       =
S-DIRECT    =       PITCH       =
CUT ST X    =       DWELL       =
CUT ST Z    =       TAP TYPE    =
```
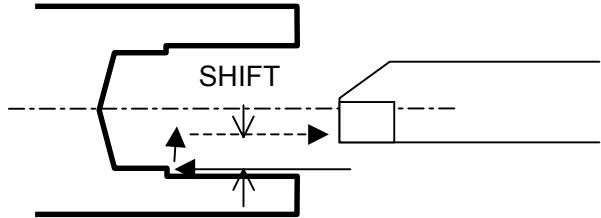
**TOOL NUM**     Number of tool used for tapping

**FEED/MIN**     Feed amount in tapping, and it can be changed to FEED/REV by pushing a [FEED/REV].

This datum is automatically calculated when a pitch datum mentioned later is inputted.

**REV/MIN**      Spindle rotating speed.

**S-DIRECT**     Spindle rotating direction. Select from [NORMAL] and [REVERS].

**HOLE DIA**     Hole diameter or tool nominal diameter.

This datum is used for only animated drawing.

**START Z**      Absolute Z-coordinate of hole start point

**DEPTH**        Depth of hole.

By pushing [END Z], absolute Z-coordinate of hole end point can be selected.

**PITCH**        Pitch of a tapping tool

**DWELL**        Dwell time at hole end point

**TAP TYPE**     Kind of tapping motion, rigid tapping or conventional tapping. Select from [RIGID] or [NORMAL].

**CUT ST X**     X-coordinate of a point where machining starts

**CUT ST Z**     Z-coordinate of a point where machining starts

- 149 -

## *6.1.2*     Supplemental Process Data in a Pop-up Window

These supplemental process data are automatically set always, so in ordinary case, an operator need not to call this pop-up window excepting the case of checking these data.

## (1) CENTER DRILLING

```
APROCH X   =
APROCH Z   =
COOLANT    =
SPDL GEAR  =
```

| | |
|---|---|
| **APPROACH X** | X coordinate of approaching intermediate point. |
| **APPROACH Z** | Z coordinate of approaching intermediate point. |
| **COOLANT** | Coolant specification at playback. Choose from the following soft-key menu. Default value is [ON]. |
| |     [ON]          (M8) |
| |     [OFF]        (M9) |
| **SPDL GEAR** | Spindle gear at playback. Choose from the following soft-key menu. Default value is [AUTO].<br>[AUTO]<br>[LOW]<br>[MID1]<br>[MID2]<br>[HIGH] |

## (2) DRILLING [DRILL]

```
ST FEED    =
ST CLEAR   =
APROCH X   =
APROCH Z   =
COOLANT    =
SPDL GEAR  =
```

| | |
|---|---|
| **ST FEED** | Cutting feedrate override at the start of cutting (parameter 9050 : STFECF) The cutting feedrate is calculated as follows.<br>Feedrate = (FEED/MIN or FEED/REV) X (STFECF) / 100 |
| **ST CLEAR** | Clearance (distance between the tool tip and tool shoulder) at the start of cutting. The tool cuts the workpiece by the above feedrate from a point distanced from the cutting start point by an amount equal to this clearance. This clearance is calculated as follows.<br>Clearance = (HOLE DIA) / (2 X Tan (nose angle/2)<br>Nose angle = parameter 9083 : DRLANGL |
| **APPROCH X** | Similar to center drilling |
| **APPROCH Z** | Similar to center drilling. |
| **COOLANT** | Similar to center drilling. |
| **SPDL GEAR** | Similar to center drilling |

# (3) DRILLING [DEEP HOLE] / [RAPID DEEP]

```
DPTH DCRS =          APROCH Z   =
RET AMNT   =          COOLANT    =
MIN DEPTH  =          SPDL GEAR  =
ST FEED      =
ST CLEAR   =
APROCH X   =
```

| | |
|---|---|
| **DPTH DCRS** | Decrement of the depth of cut for peck or high-speed peck drilling. The depth of cut is reduced by this for each cut. (parameter 9850 : DRLDEC) |
| **RET AMNT** | Amount of retraction for peck or high-speed peck drilling. (parameter 9851 : DRLRET) |
| **MIN DEPTH** | Minimum depth of cut. (parameter 9852 : DRLMIN) |
| **ST FEED** | Similar to drilling [DRILL]. |
| **ST CLEAR** | Similar to drilling [DRILL]. |
| **APPROCH X** | Similar to drilling [DRILL]. |
| **APPROCH Z** | Similar to drilling [DRILL]. |
| **COOLANT** | Similar to drilling [DRILL]. |
| **SPDL GEAR** | Similar to drilling [DRILL]. |

# (4) REAMING

```
RET SPEED =
ST FEED      =
APROCH X   =
APROCH Z   =
COOLANT    =
SPDL GEAR =
```

| | |
|---|---|
| **RET SPEED** | Feedrate for retraction to the cutting start point upon the completion of reaming motion. This feedrate is calculated as follows. (parameter 9860 : RMBROV) Feedrate = (FEED/MIN or FEED/REV) X (RTRN FD) / 100 |
| **ST FEED** | Similar to drilling [DRILL] |
| **APPROCH X** | Similar to center drilling. |
| **APPROCH Z** | Similar to center drilling. |
| **COOLANT** | Similar to center drilling. |
| **SPDL GEAR** | Similar to center drilling. |

## (5) BORING

```
APROCH X   =
APROCH Z   =
COOLANT    =
SPDL GEAR  =
```

**APPROCH X**      Similar to center drilling.
**APPROCH Z**      Similar to center drilling.
**COOLANT**        Similar to center drilling.
**SPDL GEAR**      Similar to center drilling.

## (6) TAPPING

```
APROCH X   =
APROCH Z   =
COOLANT    =
SPDL GEAR  =
```

**APPROCH X**      Similar to center drilling.
**APPROCH Z**      Similar to center drilling.
**COOLANT**        Similar to center drilling.
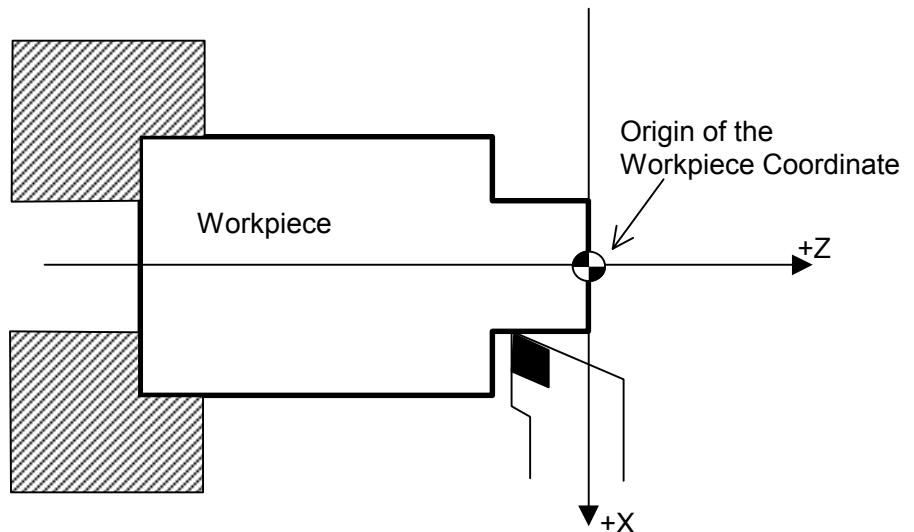**SPDL GEAR**      Similar to center drilling.

# 7 -X AREA CUTTING FUNCTION

In the cycle motions of MANUAL GUIDE, as shown in the following sample drawing, an operator enter cutting figures in the +X area, and its actual cutting motions are performed in the +X area too.

On the other hand, on the machine which has opposite double tool posts, there is a case that -X area cutting must be done as follows.
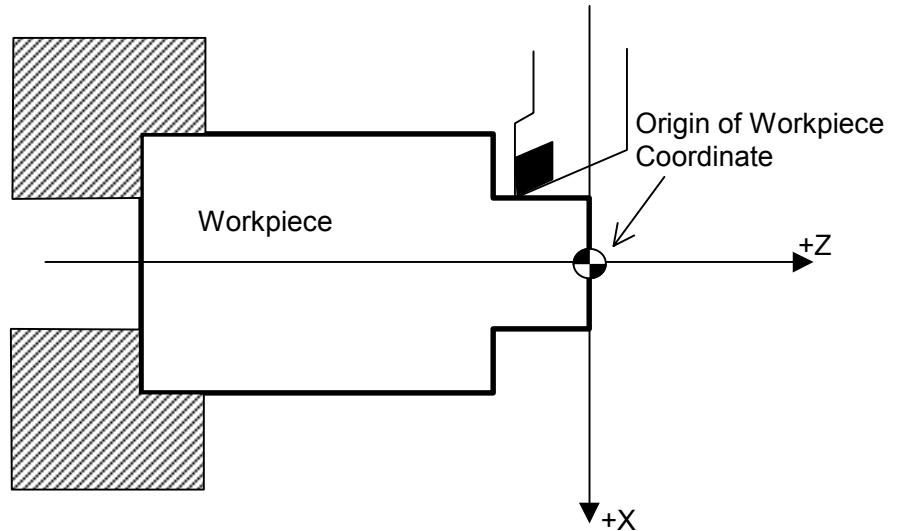
When the optional function "-X Area Cutting Function" is available, it is possible to select the cutting in +X area or -X area in the cycle motions excepting hole machining processes. Whether this optional function is available or not is depending on the machine construction, so please refer to the manual from the machine tool builder.

# *7.1*    **SELECTING OF -X AREA CUTTING**

When the "-X Area Cutting Function" is available, in the detail data window of cycle motions excepting hole machining, as shown in the following, a data item "X AREA" is displayed, and it is used for selecting -X area cutting.

Example) Detail data window of Bar machining (Roughing)

```
EDGE ANGL   =           APPROACHZ =
NOSE ANGL   =           COOLANT    =
RESE.AMNT   =           SPDL GEAR  =
RELEASE     =
X AREA      =
APPROACHX =
```

**X AREA**      Select from [+X] / [-X]. Default value is [+X].

In case of selecting the -X area cutting, an operator must enter a cutting figure of cycle motions in the +X area, and it is quite similar to the +X area cutting.
On case of performing the -X area cutting, an operator must select a tool suitable for its cutting motions in the -X area. This tool must have an opposite figure to the normal +X area cutting tool. Its tool data such as Wear/Geometry offset value and imaginary tool tip position must be set corresponding to an actual tool. For example, when the imaginary tool tip position is set to 3 on a normal +X area cutting tool, 2 must be set in its data for the -X area cutting tool.

| NOTE |
|---|
| -X area cutting cannot be selected in a Guidance Cutting, Single Cutting and hole machining process. |

# 7.2    DETAIL OF -X AREA CUTTING

When the -X area cutting is selected, the following transformation are done against the normal +X area cutting motions, and the result motions are performed.

(1) X-axis motion command

The signs of all motion commands of X-axis are reversed.

(2) G-codes

G-codes performed in cutting motions are reversed as follows.

G02 -> G03

G03 -> G02

G41 -> G42

G42 -> G41

(3) Machining simulation

According to the -X area cutting motions, a tool used for animated drawing is displayed in opposite form. And, drawing for cutting a workpiece which is accompanied with tool motion is performed in the -X area.

---

**NOTE**

When this optional function "-X Area Cutting Function" is available, animated drawing with a solid model cannot be used. As the soft-key menu for machining simulation, only [PATH] and [ANIMAT+PATH] are displayed, and [ANIMAT] is never displayed.

---

# IV. SAMPLE OF PROGRAMMING

# 1 EXAMPLE OF CONTOUR FIGURE INPUTTING
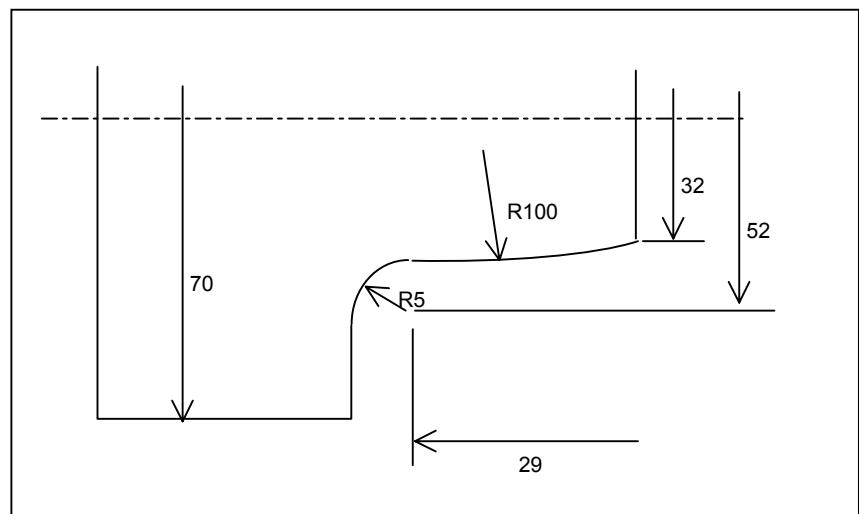
> ⚠ **WARNING**
> The following example of entering contour data is intend only to illustrate what automatic calculation of cross points is like when contour data is entered. The contour data given in the example may not be able to used in actual machining.
> If the machine is forced to run according to a program like the one shown above, the tool may bump against the workpiece, and the machine may be forced to perform unnatural machining, possibly causing damage to the tool and/or machine, and even injuries.

> **NOTE**
> The following examples show the input procedure for contours, assuming that the process data for bar machining has already been inputted.
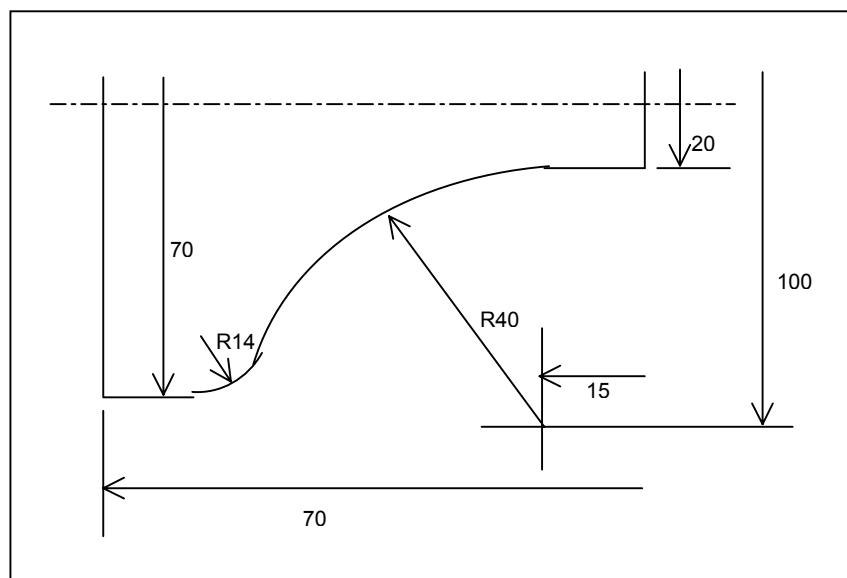
## Sample 1



| | | | |
|---|---|---|---|
| 32 | **INPUT** | (Start Point X) |
| 0 | **INPUT** | (Start Point Z) |
| [CIRCLE CW] | | |
| 100 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CCW] | | |
| 5 | **INPUT** | (Radius) |
| 52 | **INPUT** | (Center X) |
| -29 | **INPUT** | (Center Z) |
| **INSERT** | | |
| 2 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 70 | **INPUT** | (End Point X) |
| 90 | **INPUT** | (Angle) |
| **INSERT** | | |
| [LINE] | | |
| -70 | **INPUT** | (End Point Z) |
| **INSERT** | | |
| [LINE] | | |
| 80 | **INPUT** | (End Point X) |
| **INSERT** | | |

## Sample 2



| | | |
|---|---|---|
| 20 | **INPUT** | (Start Point X) |
| 0 | **INPUT** | (Start Point Z) |
| [LINE] | | |
| -15 | **INPUT** | (End Point Z) |
| **INSERT** | | |
| [CIRCLE CCW] | | |
| 40 | **INPUT** | (Radius) |
| 100 | **INPUT** | (Center X) |
| -15 | **INPUT** | (Center Z) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CW] | | |
| 14 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 70 | **INPUT** | (End Point X) |
| -70 | **INPUT** | (End Point Z) |
| 180 | **INPUT** | (Angle) |
| **INSERT** | | |
| 2 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [LINE] | | |
| 80 | **INPUT** | (End Point X) |
| **INSERT** | | |

- 161 -

## Sample 3
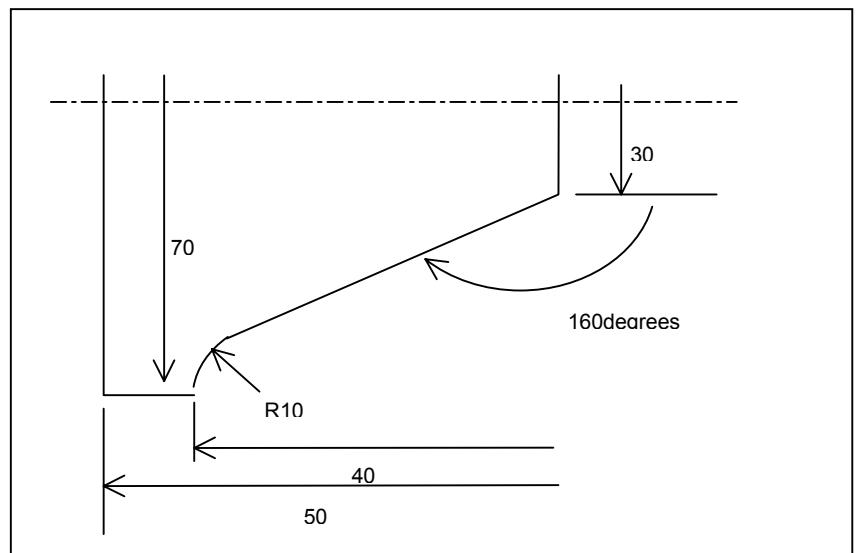


| | | |
|---|---|---|
| 30 | **INPUT** | (Start Point X) |
| 0 | **INPUT** | (Start Point Z) |
| [LINE] | | |
| 160 | **INPUT** | (Angle) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CCW] | | |
| 70 | **INPUT** | (End Point X) |
| -40 | **INPUT** | (End Point Z) |
| 10 | **INPUT** | (Radius) |
| **INSERT** | | |
| 1 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [LINE] | | |
| -50 | **INPUT** | (End Point Z) |
| **INSERT** | | |
| [LINE] | | |
| 80 | **INPUT** | (End Point X) |
| **INSERT** | | |

## Sample 4



| | | |
|---|---|---|
| 30 | **INPUT** | (Start Point X) |
| 0 | **INPUT** | (Start Point Z) |
| | | |
| [CIRCLE CW] | | |
| 10 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 70 | **INPUT** | (End Point X) |
| -40 | **INPUT** | (End Point Z) |
| 160 | **INPUT** | (Angle) |
| **INSERT** | | |
| 2 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [LINE] | | |
| -50 | **INPUT** | (End Point Z) |
| **INSERT** | | |
| [LINE] | | |
| 80 | **INPUT** | (End Point X) |
| **INSERT** | | |

## Sample 5
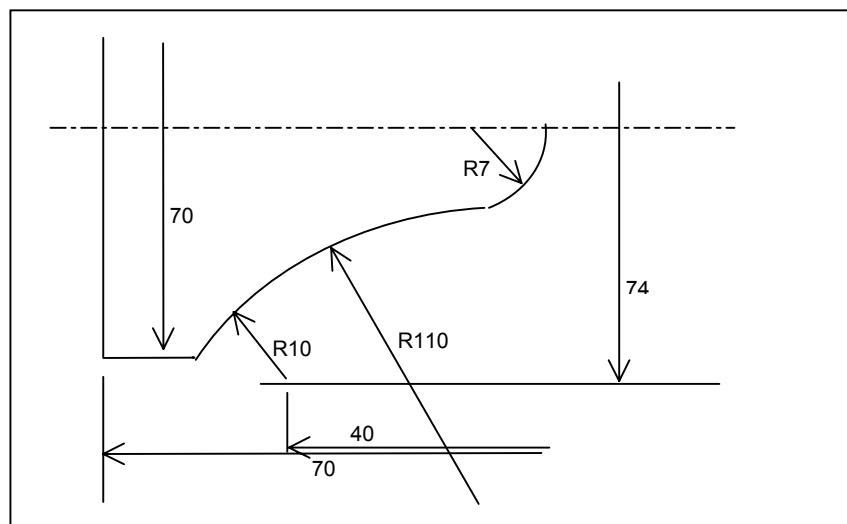


| 0 | **INPUT** | (Start Point X) |
|---|-----------|-----------------|
| 0 | **INPUT** | (Start Point Z) |
| [CIRCLE CW] | | |
| 7 | **INPUT** | (Radius) |
| 0 | **INPUT** | (Center X) |
| -7 | **INPUT** | (Center Z) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CCW] | | |
| 110 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CCW] | | |
| 10 | **INPUT** | (Radius) |
| 74 | **INPUT** | (Center X) |
| -40 | **INPUT** | (Center Z) |
| **INSERT** | | |
| 1 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [LINE] | | |
| 70 | **INPUT** | (End Point X) |
| -70 | **INPUT** | (End Point Z) |
| 180 | **INPUT** | (Angle) |
| **INSERT** | | |
| 2 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [LINE] | | |
| 80 | **INPUT** | (End Point X) |

## Sample 6
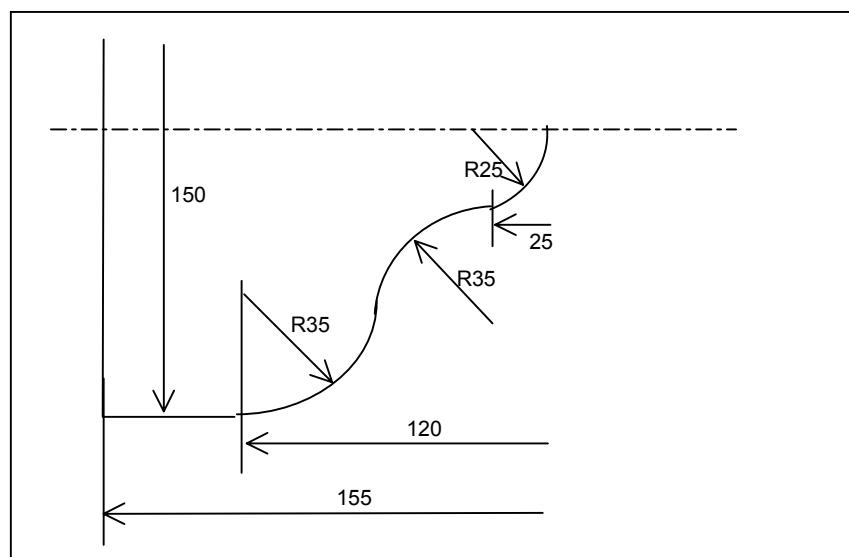


| | | |
|---|---|---|
| 0 | **INPUT** | (Start Point X) |
| 0 | **INPUT** | (Start Point Z) |
| [CIRCLE CW] | | |
| 25 | **INPUT** | (Radius) |
| 0 | **INPUT** | (Center X) |
| -25 | **INPUT** | (Center Z) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CCW] | | |
| 35 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CW] | | |
| 35 | **INPUT** | (Radius) |
| 80 | **INPUT** | (Center X) |
| -120 | **INPUT** | (Center Z) |
| **INSERT** | | |
| 1 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 150 | **INPUT** | (End Point X) |
| -150 | **INPUT** | (End Point Z) |
| 180 | **INPUT** | (Angle) |
| **INSERT** | | |
| [LINE] | | |
| 160 | **INPUT** | (End Point X) |

- 165 -

**Sample 7**
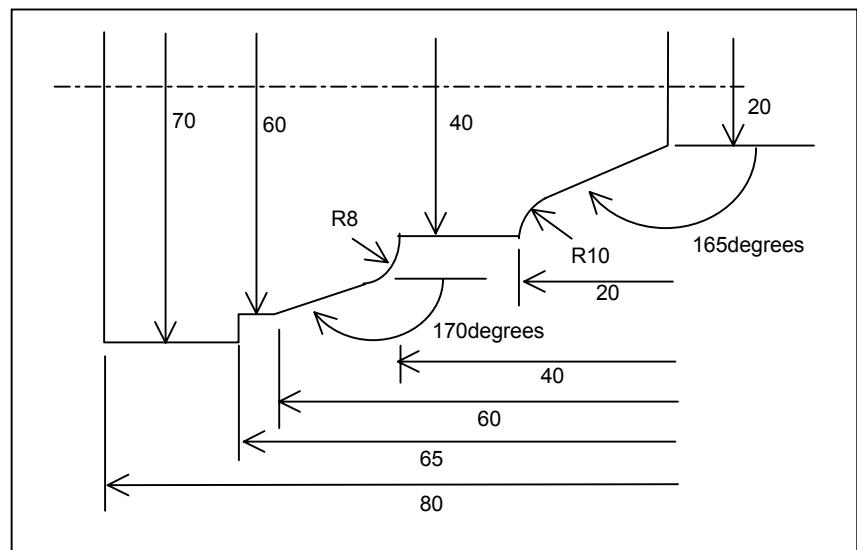


| | | |
|---|---|---|
| 20 | **INPUT** | (Start Point X) |
| 0 | **INPUT** | (Start Point Z) |
| [LINE] | | |
| 165 | **INPUT** | (Angle) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CCW] | | |
| 40 | **INPUT** | (End Point X) |
| -20 | **INPUT** | (End Point Z) |
| 10 | **INPUT** | (Radius) |
| **INSERT** | | |
| 1 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [LINE] | | |
| -40 | **INPUT** | (End Point Z) |
| **INSERT** | | |
| [CIRCLE CW] | | |
| 8 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 60 | **INPUT** | (End Point X) |
| -60 | **INPUT** | (End Point Z) |
| 170 | **INPUT** | (Angle) |
| **INSERT** | | |
| 2 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [LINE] | | |
| -65 | **INPUT** | (End Point Z) |
| **INSERT** | | |

- 166 -

[LINE]
70          **INPUT**                    (End Point X)
**INSERT**
[LINE]
-80         **INPUT**                    (End Point Z)
**INSERT**
[LINE]
80          **INPUT**                    (End Point X)
**INSERT**

## Sample 8
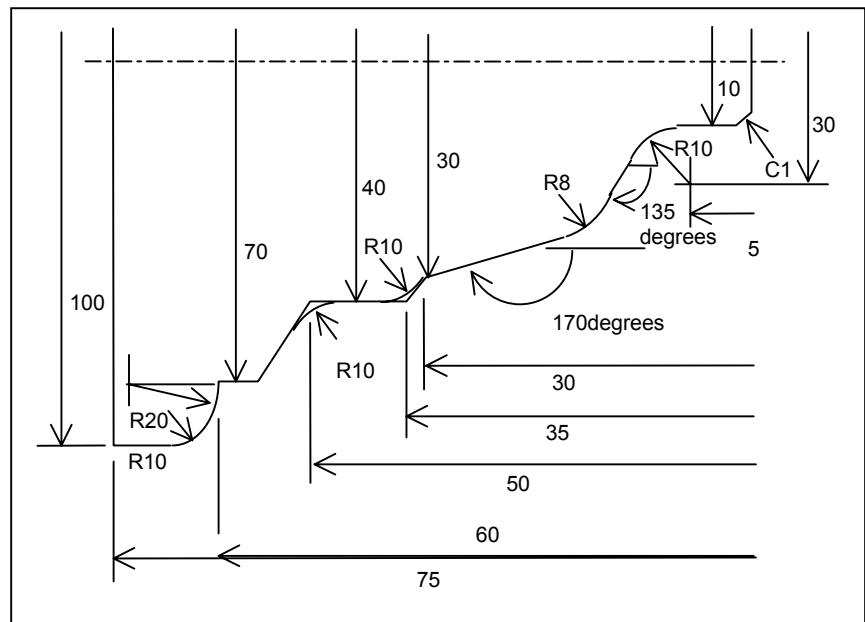


| | | |
|---|---|---|
| 10 | **INPUT** | (Start Point X) |
| 0 | **INPUT** | (Start Point Z) |
| [CHAMF] | | |
| 1 | **INPUT** | (Chamfer) |
| **INSERT** | | |
| [LINE] | | |
| -5 | **INPUT** | (End Point Z) |
| **INSERT** | | |
| [CIRCLE CCW] | | |
| 10 | **INPUT** | (Radius) |
| 30 | **INPUT** | (Center X) |
| -5 | **INPUT** | (Center Z) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 135 | **INPUT** | (Angle) |
| **INSERT** | | |
| 2 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CW] | | |
| 8 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 30 | **INPUT** | (End Point X) |
| -30 | **INPUT** | (End Point Z) |

| | | |
|---|---|---|
| 170 | **INPUT** | (Angle) |
| **INSERT** | | |
| [CIRCLE CW] | | |
| 10 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 40 | **INPUT** | (End Point X) |
| -50 | **INPUT** | (End Point Z) |
| 180 | **INPUT** | (Angle) |
| **INSERT** | | |
| 2 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [ROUND] | | |
| 10 | **INPUT** | (Radius) |
| **INSERT** | | |
| [LINE] | | |
| 70 | **INPUT** | (End Point X) |
| 110 | **INPUT** | (Angle) |
| **INSERT** | | |
| [LINE] | | |
| -60 | **INPUT** | (End Point Z) |
| **INSERT** | | |
| [CIRCLE CW] | | |
| 20 | **INPUT** | (Radius) |
| 70 | **INPUT** | (Center X) |
| -80 | **INPUT** | (Center Z) |
| **INSERT** | | |
| [TANGNT] | | |
| [CIRCLE CW] | | |
| 10 | **INPUT** | (Radius) |
| **INSERT** | | |
| [TANGNT] | | |
| [LINE] | | |
| 100 | **INPUT** | (End Point X) |
| -75 | **INPUT** | (End Point Z) |
| 180 | **INPUT** | (Angle) |
| **INSERT** | | |
| 1 | **INPUT** | (Cross Point Number) |
| **INSERT** | | |
| [LINE] | | |
| 110 | **INPUT** | (End Point X) |
| **INSERT** | | |

# V. SUPPLEMENT

# 1 CALCULATING INPUT DATA

# *1.1*    **OVERVIEW**

Data can be input for each numeric data item on the MANUAL GUIDE screen with calculator-type arithmetic operations.

One of four dyadic fundamental operations (addition, subtraction, multiplication, and division) can be performed as an operation at a time. The result of an operation can be used as the first term of the next operation to perform as many operations as required for one numeric data input item.

The inputtable numeric data depends on the data format of the target numeric data item. If data smaller than the least input increment is input, processing such as round off is not performed and the data is ignored.

# *1.2*     **OPERATION**

Use the following key operations for the arithmetic operations :
(1)   Addition          :          100.+200.[INPUT]
(2)   Subtraction       :          100.-200.[INPUT]
(3)   Multiplication    :          100.*20.[INPUT]
(4)   Division          :          100./10.[INPUT]
The displayed result of an operation is not directly entered in the numeric data item. The operator must check the displayed result of an operation, then press [INPUT] to enter the result in the input data item.
Example)
      210. - 65.3 + 1.25
      Key input operation and displayed results of pressing the INPUT key for the above operation

- Key operation

| 2 | 1 | 0 | . | - | 6 | 5 | . | 3 | **INPUT** | + | 1 | . | 2 | 5 | **INPUT** | **INPUT** |

- Key-in buffer display

| 210.-65.3 | START X |
|---|---|

(INPUT key)

| 144.7 | START X |
|---|---|

- (Key operation for the second operation)
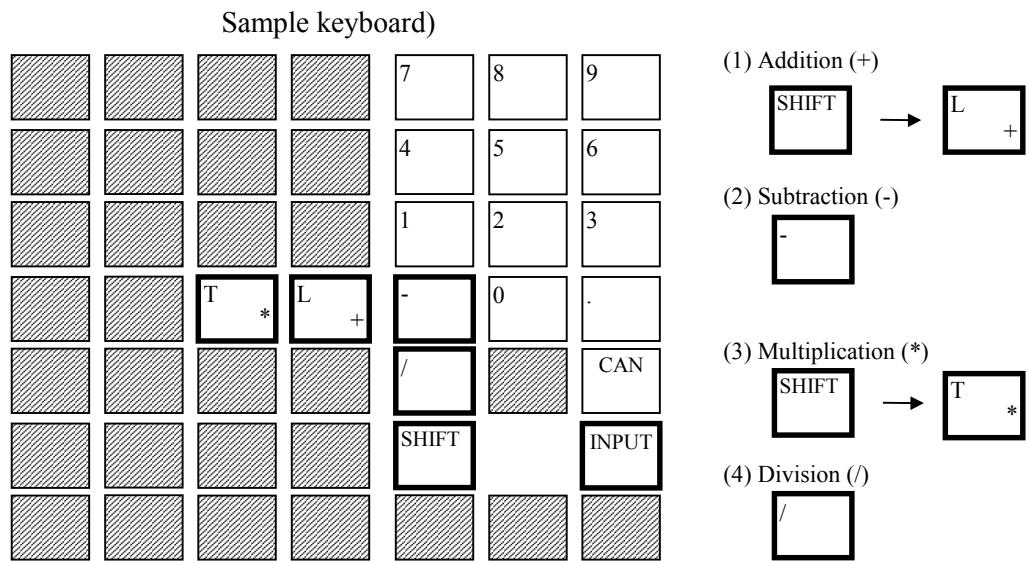
| 144.7+1.25 | START X |
|---|---|

- (INPUT key)

| 145.95 | START X |
|---|---|

- (INPUT key)

The operation result (149.95) is input to the START X data item.

# *1.3*    KEYS FOR ARITHMETIC OPERATIONS

Sample keyboard)



(1) Addition (+)

(2) Subtraction (-)

(3) Multiplication (*)

(4) Division (/)

# 2          USEFUL DISPLAY FUNCTIONS

# *2.1*  BLANK FIGURE DATA DISPLAY IN THE PROGRAM LIST WINDOW

Blank figure data items can also be displayed in the program list window displayed by pressing soft-key [PRGRM], by parameter setting.

| PROGRAM LIST | | | | BLANK SIZE |
|---|---|---|---|---|
| PROGRAM (NUM.) USED/FREE | | 12/13 | | LENGTH |
| MEMORY (BLOCK) USED/FREE | | 112/255 | | 32.000 |
| 01 : CIRCLE MACHINING | 5M12S | 1998/10/06 | 10:12 | OUTER DIA |
| 02 : LINEAR MACHINING | 4M15S | 1998/10/05 | 11:15 | 25.000 |
| 03 : DRILLING | 2M10S | 1998/10/01 | 10:20 | INNER DIA |
| 04 : OUTER ROUGH | 6M15S | 1998/10/02 | 11:15 | 0.000 |
| 05 : GROOVING | 3M22S | 1998/10/06 | 10:12 | REMOVAL |
| 10 : TAPPING | 1M35S | 1998/10/07 | 11:15 | 2.000 |
| 11 : THREADING | 3M12S | 1998/10/06 | 10:12 | |
| 15 : TEST-1 | 7M45S | 1998/10/05 | 11:15 | |
| 18 : TEST-2 | 9M52S | 1998/10/05 | 10:12 | |
| 20 : INNER MACHINING | 6M35S | 1998/10/05 | 11:15 | |

Blank figure data can be set for each machining program.
When the cursor moves to the next program, the blank figure data for the new program is also displayed. Pressing soft-key [WORK] moves the cursor to the blank figure data field to allow the operator to input required data. Pressing soft-key [PRGRM] moves the cursor back to the program list.

---

**NOTE**
1  To display the above blank figure data in the program list window, set bit 2 (BSZ) of parameter No. 9761 to 1.
2  To perform simultaneous bar machining and end facing with a value set for REMOVAL, be sure to specify a vertical line drawn from the start point (X0, Z0) as the first figure.

---

# *2.2* ALARM MESSAGE WINDOW

An alarm message window is displayed when soft-key [ALARM] is pressed on the MANUAL GUIDE screen or an alarm is issued during actual machining or animated drawing.

| ALARM MESSAGE |
|---|
| 3011   CHECK PROCESS DATA |
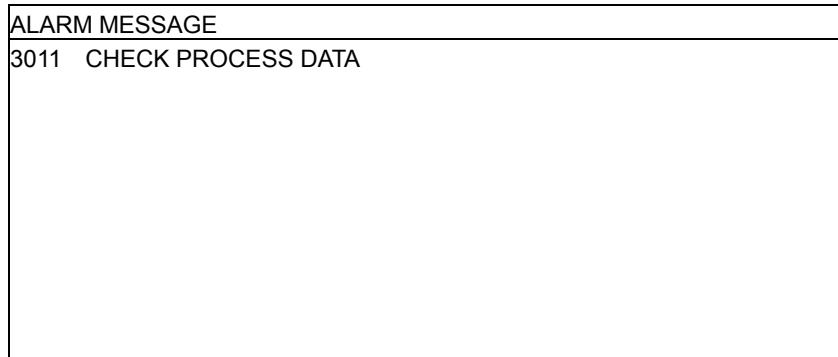
**NOTE**
> To enable the above function of displaying the alarm message window, set bit 4 (AWD) of parameter No. 9761 to 1.

If an alarm is issued, the nature of the alarm can be checked on the MANUAL GUIDE screen and the alarm screen need not be displayed when this window is used.

# *2.3*   VARIOUS TYPES OF CURRENT POSITION DISPLAY AND COORDINATE SYSTEM PRESET

Pressing soft-key [POS] can step the current position data displayed in the status indicator window from the absolute coordinates to the relative coordinates and machine coordinates.

When the relative coordinates are displayed, the X, Z, or C coordinate can be preset by the following operation:

Press the rightmost soft-key [+]. R [PRESET] is displayed.

Press address key X, Z, or C.

Use numeric keys to input a value to be preset.

Press [PRESET].

> **NOTE**
> To enable the current position display switch function and relative coordinate preset function above, set bit 1 (POS) of parameter No. 9846.

## *2.4*     SOFT-KEY FOR ANIMATED DRAWING SCALING OPERATION

Soft-key [ZOOM] used for animated drawing scaling operation can be displayed on the first page containing various soft-keys for animated drawing, by parameter setting.

[<][SPD UP] [SPD DW] [PLOT] [ROTATE] [CIRC] [ZOOM] [HEAD] [PROCES] [EXEC] [SINGLE][+]

Pressing [ZOOM] displays the following soft-keys to allow the operator to scale a drawing up. Pressing [DESIDE] executes scaleup operation, then the previous soft-keys are displayed again.

[<][SPD UP][SPD DW][PLOT][EXEC][          ][NORMAL][SCL UP][SCL DW][CANCEL][DECIDE][+]

> **NOTE**
>    To display the above soft-key for scaling operation, set bit 3 (ZOM) of parameter No. 9764 to 1.

# *2.5*     DIRECT INPUT OF THE JOG FEEDRATE

Data item "JOG FED X/Z" can be displayed on the basic menu screen and guidance/single cutting screen to directly input the jog feedrate, by parameter setting.

The input data is temporarily stored as common parameter No. 9097 and is converted according to the increment system and data format, then is written as parameter No. 1423 for the jog feedrate.

When data is input directly as common parameter No. 9097, the corresponding data is written as parameter No. 1423 for the jog feedrate when the basic menu screen is displayed.

The valid data format and increment system are as follows:

When the parameter No.1402#4(JRV) is set to 0.
    1mm/min. (at mm input)
    0.1inch/min. (at inch input)

When the parameter No.1402#4(JRV) is set to 1.
    0.01mm/rev. (at mm input)
    0.001inch/rev. (at inch input)

---

**NOTE**
To enable the above jog feedrate input function, set bit 0 (JOG) of parameter No. 9846 to 1.

---

⚠ **WARNING**
1  After jog feedrate data is input, the feedrates related to all jog feed operations on the CNC are changed to the new input feedrate.
For this reason, the jog feedrate indicated on the machine operator's panel or in the machine operator's manual becomes unreliable. If you want to prevent this situation, be sure to disable this direct jog feedrate input function.
If you do not have enough skill and knowledge for machine operation, never perform this direct jog feedrate input operation. Otherwise the tool may strike the workpiece or machine, possibly breaking the tool or machine or injuring you.
2  If the input jog feedrate is very fast, the tool may strike the workpiece or machine, possibly breaking the tool or machine or injuring the operator.
For this reason, be sure to confirm, on the screen, that the input data is correct before moving the tool. Moreover, before actual move operation, move the tool by a very short distance at a time gradually with sufficient care and confirm that the tool moves at the correct feedrate.
If you do not have enough skill and knowledge for machine operation, never perform this direct jog feedrate input operation.

# *2.6*    NC STATEMENT DISPLAY ON THE MACHINING SIMULATION SCREEN

The specified NC statements can be displayed on the machining simulation screen by parameter setting. This allows the operator to check the NC statements specified during conversion of NC statements or execution of a program.

> **NOTE**
> To enable the above NC statement display function, set bit 2 (NCD) of parameter No. 9847 to 1.
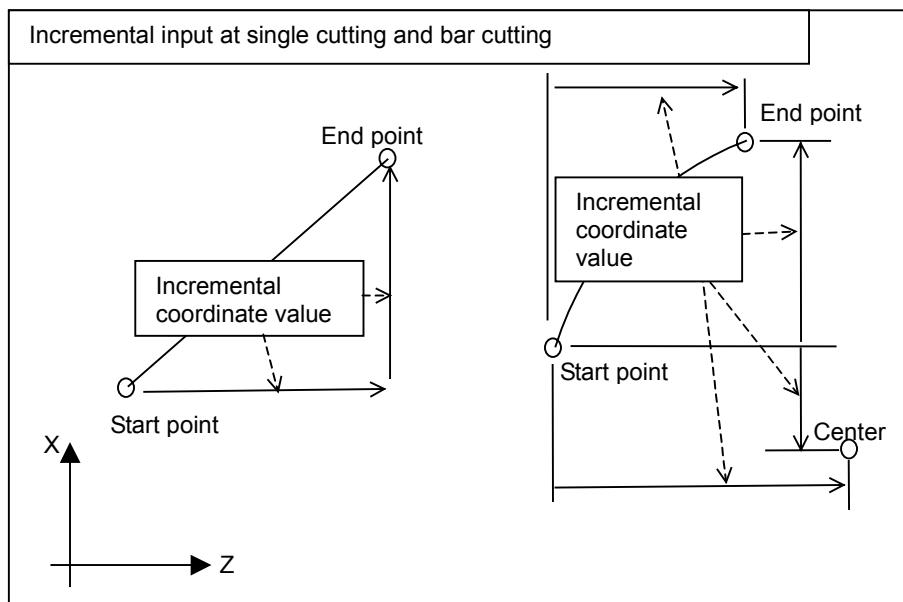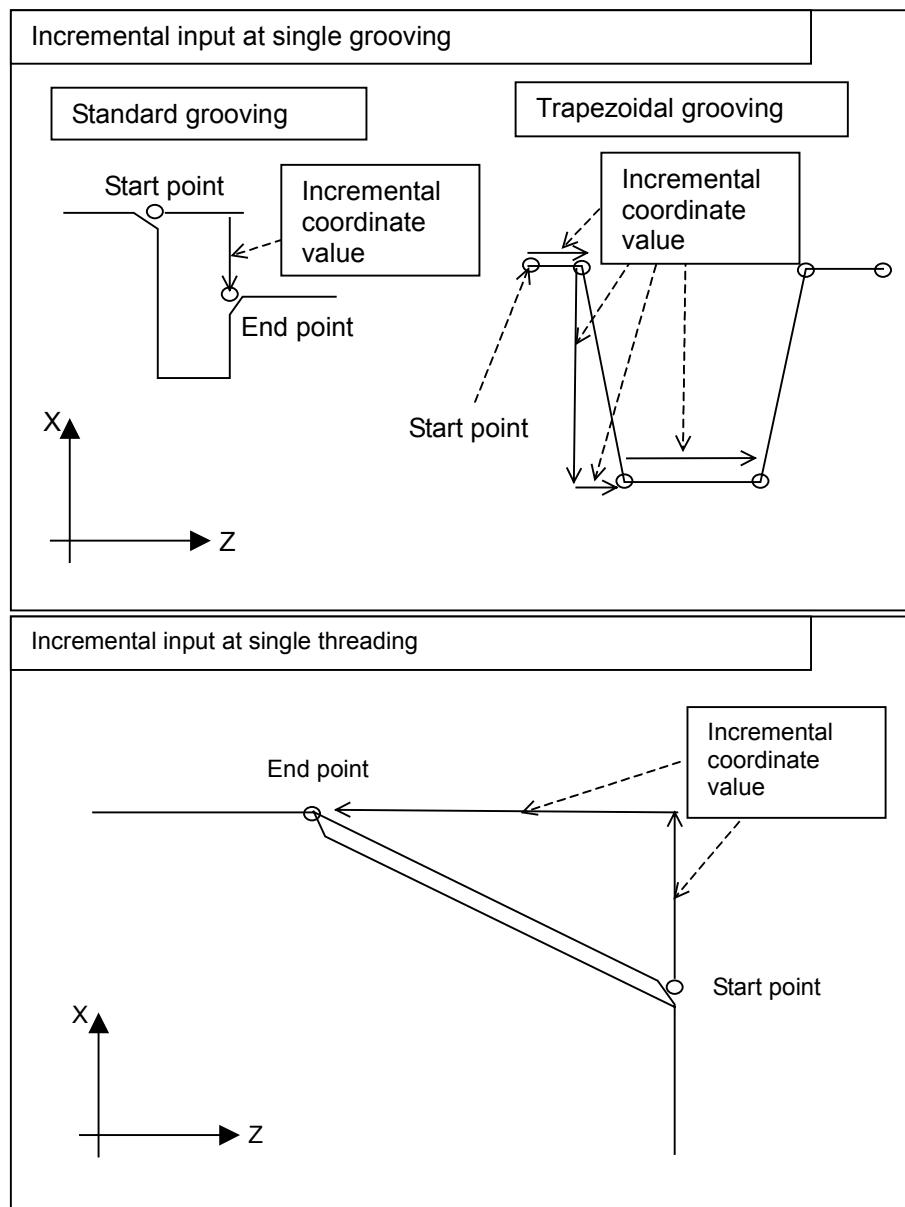
# 3    USEFUL FUNCTIONS FOR MACHINING PROCESSES

# *3.1*    **INCREMENTAL COORDINATE INPUT**

Incremental values can be input for the following data in each machining below by parameter setting:

- Single Cutting
  - Coordinates of the end point in each block
  - Coordinates of the center of an arc
- Bar machining (Roughing, Finishing)
  - Coordinates of the end point of each figure
  - Coordinates of the center of an arc
- Threading
  - Coordinates of the end point of a thread
- Standard grooving
  - X coordinate (outer or inner surface) or Z coordinate (end face) of the end point of a groove
- Trapezoidal grooving (Roughing, Finishing)
  - Coordinates of each point other than the start point of a trapezoidal groove

Each incremental coordinate indicates the distance from the start point to the end point of a line or arc. Each incremental coordinate of the center of an arc indicates the distance from the start point of the arc.



Incremental input at single cutting and bar cutting

Incremental input at single grooving

Standard grooving

Trapezoidal grooving



Incremental input at single threading



Pressing soft-key [ABS/IN] can switch between absolute input (standard specification of the MANUAL GUIDE) and incremental input. The selected input format (absolute or incremental) is displayed on the title bar of the figure data window.

> **NOTE**
> Absolute input and incremental input cannot be specified together in one machining process.
> If absolute input is switched to incremental input or vice versa in one process, the format of all figure data in the process is changed to the newly selected input format with the input numeric values. In this case, because the figure data cannot be used as is, delete all input figures, then input them again.

The address of each coordinate in each block input in the incremental format in a single cutting process is indicated with "U" (X-axis) or "W" (Z-axis).

When coordinates are input in the incremental format in a bar machining process, the figure code of each figure is displayed in reverse video as shown in the example below:

<- X30.0 Z –20. P100;

---

**NOTE**
To enable incremental input, set bit 4 (INC) of parameter No. 9764 to 1.

---

# *3.2*      **SUPPORT OF BACK TOOL POST LATHES**

The MANUAL GUIDE supports a flatbed manual lathe, of which tool post is located between the workpiece and operator, in principle.
By using the "back tool post lathe support function" (option function), the MANUAL GUIDE can also be used for a slant-bed lathe, of which tool post is located beyond the workpiece viewed from the operator.
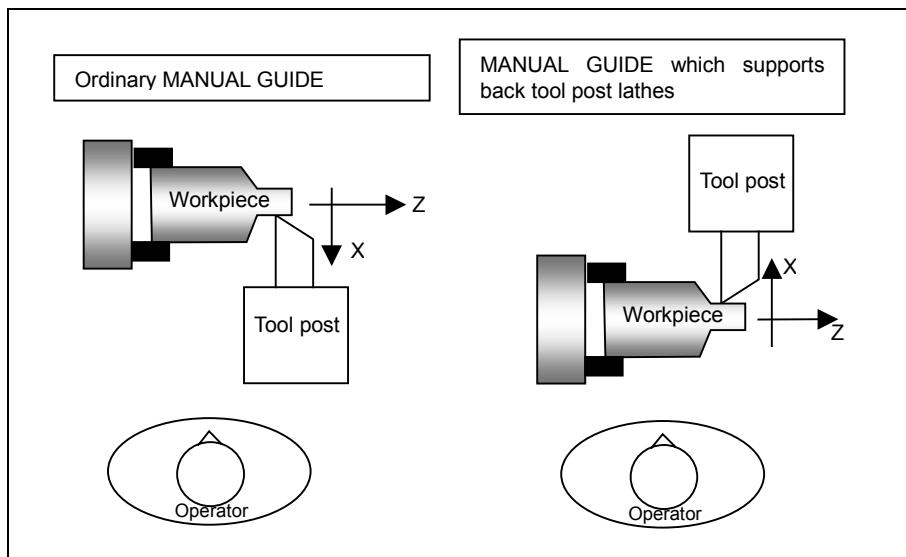


> **NOTE**
> When this option function is installed, set the following parameters because the upward direction on the screen indicates the positive X direction in the coordinate system for the lathe:
> Bit 5 (SRV) of parameter No. 9761 = 0 (The soft-key icon points up.)
> Parameter No. 6510 = 14 (The upward direction on the screen indicates the positive X direction in the coordinate system of animated drawing.)

When this option is installed, all guidance drawings on the MANUAL GUIDE screen and input figures in the graphic window are displayed top side down to match the direction of the X-axis of the actual machine.

> **NOTE**
> For a lathe for which the MANUAL GUIDE is installed together with this option function, manual cutting such as guidance or single cutting may not be needed.
> In this case, the following parameter can be set so that the related soft-keys for cutting are not displayed on the menus:
> Bit 2 (NGS) of parameter No. 9762 = 1 (Soft-keys [GUIDNC] and [SINGLE] are not displayed.)

# *3.3*     **UNIDIRECTIONAL GROOVE CUTTING**

In ordinary grooving, for both normal and trapezoidal grooves, the center of a groove is first cut with the tool, then the tool is moved right and left to cut the groove.

Unidirectional cutting can also be selected by parameter setting. In this case, an end face side of the workpiece to be grooved is first cut with the tool, then the tool is moved toward the chuck to cut the groove.

This cutting method can be used for suppressing vibrations which may be caused by the tool during heavy cutting.

> **NOTE**
> To use this unidirectional groove cutting, set bit 5 (G1W) of parameter No. 9764 to 1.
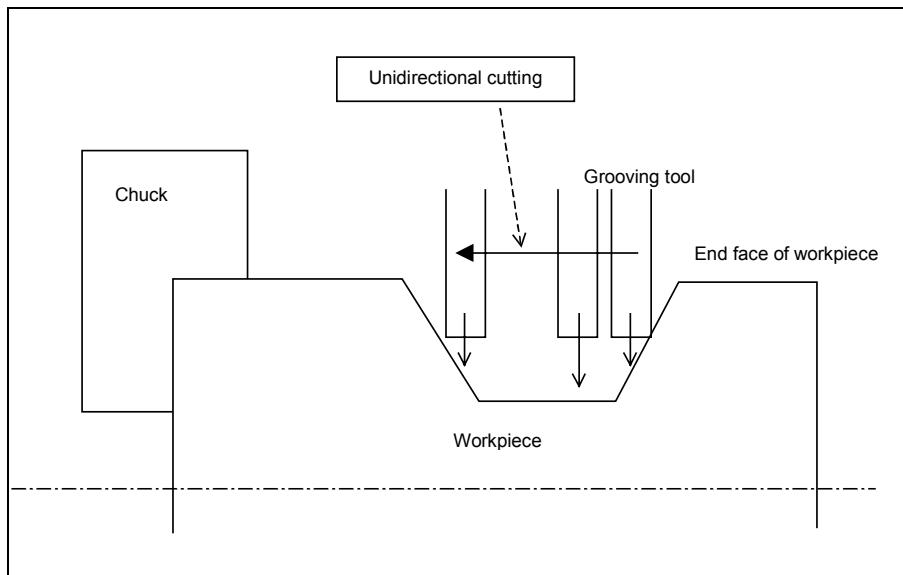
# *3.4*   AUTOMATIC DIVISION OF TAUGHT-IN BLOCKS IN GUIDANCE CUTTING

In guidance cutting, a cutting motion whose end point is the tool position when a teach-in soft-key ([CUT]/[RAPID]) is pressed is stored as a motion block.

Each motion can also be stored as a block according to the following conditions by parameter setting:
  - Switching between approach cutting and along cutting
  - Direction of cutting in the handle or jog feed mode
  - Axis along which the tool moves in the handle or jog feed mode

When soft-key [CUT] is pressed, all cutting motions stored as separate blocks are taught as cutting feed (G01, G02, or G03).

When soft-key [RAPID] is pressed, they are taught as rapid traverse (G00).

Up to 30 blocks of motions can be stored until the teach-in soft-key is pressed.

When this function is enabled, if the tool is moved simultaneously along two axes in the handle or jog feed mode, large blocks are stored at a time. Always move the tool along only one axis.

---

**NOTE**

1  To use automatic division of taught-in cutting motions, set bit 1 (TCA) of parameter No. 9762 to 1.

2  For auxiliary function teach-in, up to eight blocks of motions can continuously be stored, regardless of whether the above parameter is set.

3  The same block may be taught as multiple continuous blocks by handwheel operation. In this case, delete unnecessary blocks.

---

# *3.5*   SPECIFICATION OF SEMIFINISHING FOR BAR MACHINING

Semifinishing only can be executed for bar machining by parameter setting.

The "ROUGH/SEMI" data item is displayed in the detail data window for bar machining (roughing) to allow the operator to select a cutting motion from the soft-key menu.

- [ROUGH]      →   Executes only roughing.
- [R+SEMI]     →   Executes roughing and semifinishing.
- [SEMI]       →   Executes only semifinishing.

> **NOTE**
>    To use this semifinishing specification function, set bit 6 (SEM) of parameter No. 9761 to 1.

# *3.6*    SAFE OPERATION FOR PROGRAM SELECTION

To prevent machining operation with a program erroneously selected by an operator error, only the specified program can be used for machining operation.

Move the cursor to a program to be used for machining in the program list window, then press soft-key [SELPR].

When soft-key [SELPR] is pressed, the screen changes to the machining simulation screen determined by the setting of bit 3 (SGR) of parameter No. 9847. When the Playback Start button on the operator's panel is pressed, playback machining starts.

If the Playback Start button is pressed without pressing soft-key [SELPR] after another program is selected in the program list window using the cursor, an alarm is issued and playback machining cannot be started.

> **NOTE**
> To enable the above function, set bit 7 (SPR) of
> parameter No. 9847 to 1.

## *3.7*   **REVERSE TAPPING**

Reverse tapping can be performed by parameter setting. When this function is enabled, the M code indicating forward or reverse rotation is issued before tapping.

> **NOTE**
> The above function is enabled by setting bits 5 (MTM) and 4 (TTM) of parameter No. 9846. For details of the parameter setting, see Appendix A, "PARAMETERS." When bit 6 (G32) of parameter No. 9779 is set to 1, this function is disabled for tapping by turning.

When this function is enabled, the motion of each machining pattern is as follows.

**(1) Forward tapping by turning :**
**When the parameter No.9779#6(G32) is set to 0.**

```
                    :
(M8/M9);
G50 S_ ;
                    :
M(Forward spindle rotation tapping pre-command P9292); . . . (A)*
G97G99G40S_ T_ M(Forward spindle rotation command P9295);
G80;
(Movement to the cutting start point)
G99G84Z_ R0.F_ P_ ;
G80;
                    :
```
\*    Block (A) is output when a value is set for parameter No. 9292.

**(2) Reverse tapping by turning :**
**When the parameter No.9779#6(G32) is set to 0.**

```
                    :
(M8/M9);
G50 S_ ;
                    :
M(Reverse spindle rotation tapping pre-command P9293); . . . (A)*
G97G99G40S_ T_ M(Forward spindle rotation command P9295);
G80;
(Movement to the cutting start point)
G99G84Z_ R0.F_ P_ ;
G80;
                    :
```

> **NOTE**
> 1  When bit 4 (TTM) of parameter No. 9846 is set to 0 or when the bit is set to 0 and parameter No. 9293 is not set, reverse rotation may be specified as the direction of rotation for tapping (in process data). In this case, P/S alarm, "3008 A WRONG TOOL SELECTED," is issued.
> 2  In reverse tapping, the reverse spindle rotation command is also issued at the bottom of the hole in the same way as in forward tapping. To prevent this command from being issued, the M code for the reverse spindle rotation tapping pre-command is output before a tapping command (G84 or G88) is issued to issue the reverse rotation command at the bottom of the hole. For this reason, set a signal to reverse the spindle rotation commands following this M code in the PMC ladder to make the system reverse the spindle rotation commands according to the signal. Make the PMC ladder cancel the signal at the time such as at the falling edge of the TAP signal (bit 5 (TAP) of F0001). Use the M code for the forward spindle rotation tapping pre-command as required.

## (3) Forward rigid tapping by turning : When the parameter No.9779#6(G32) is set to 0.

```
                    :
(M8/M9);
G50 S_ ;
G99G40T_ ;
G80;
(Movement to the cutting start point)
┌────────────────────────────────────────────────┐
│M(Spindle stop command P9294); . . . .. . . (B1) │
│     or  . . . (B)*                               │
│M(Forward spindle rotation tapping pre-commandP9292);│
│     . . . (B2)                                   │
└────────────────────────────────────────────────┘
S_ M(Rigid tapping mode command P5210);
G99G84Z_ R0.F_ P_ ;
G80;
                    :
```

*(For block (B),
- Block (B1) is output when bit 4 (TTM) of parameter No. 9846 is set to 0.
- Block (B2) is output when bit 4 (TTM) of parameter No. 9846 is set to 1 and a value is set for parameter No. 9292. When no value is set for parameter No. 9292, no command is output.

## (4) Reverse rigid tapping by turning :
## When the parameter No.9779#6(G32) is set to 0.

```
                 :
(M8/M9);
G50 S_ ;
G99G40T_ ;
G80;
(Movement to the cutting start point)
M(Reverse spindle rotation tapping pre-command P9293);
S_ M(Rigid tapping mode command P5210);
G99G84Z_ R0.F_ P_ ;
G80;
                 :
```

> **NOTE**
> When bit 4 (TTM) of parameter No. 9846 is set to 0
> or when the bit is set to 0 and parameter No. 9293
> is not set, reverse rotation may be specified as the
> direction of rotation for tapping. In this case, P/S
> alarm, "3008 A WRONG TOOL SELECTED," is
> issued.

## (5) C-axis forward tapping by milling, even spacing

```
                 :
M(Milling axis stop command P9876);
(M8/M9);
M(Milling forward tapping pre-command P9088); . . . (C)*
G97G98G40S_ T_ M(Forward milling axis rotation command P9877)
G80;
(Movement to the cutting start point)
G98(G84/G88)_ ;
(Second figure specification)
                 :
(nth figure specification)
                 :
G80M(Milling axis stop command P9876);
                 :
```

*    Block (C) is output when bit 5 (MTM) of parameter No. 9846 is
     set to 1 and a value is set for parameter No. 9088.

## (6) C-axis reverse tapping by milling, even spacing

```
                    :
M(Milling axis stop command P9876);
(M8/M9);
M(Milling reverse tapping pre-command P9089);
G97G98G40S_ T_ M(Forward milling axis rotation command P9877)
G80;
(Movement to the cutting start point)
G98(G84/G88)_ ;
(Second figure specification)
                    :
(nth figure specification)
                    :
G80M(Milling axis stop command P9876);
                    :
```

> **NOTE**
> When bit 4 (TTM) of parameter No. 9846 is set to 0
> or when the bit is set to 0 and parameter No. 9089
> is not set, reverse rotation may be specified as the
> direction of rotation for tapping. In this case, P/S
> alarm, "3008 A WRONG TOOL SELECTED," is
> issued.

## (7) C-axis forward tapping by milling, uneven spacing

```
                    :
M(Milling axis stop command P9876);
(M8/M9);
M(Milling forward tapping pre-command P9088); . . . (D)*
G97G98G40S_ T_ M(Forward milling axis rotation command P9877)
G80;
(Movement to the cutting start point)
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
(Second figure specification)
M(Milling forward tapping pre-command P9088); . . . (D)*
M(Milling forward command P9877);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
                    :
(nth figure specification)
M(Milling forward tapping pre-command P9088); . . . (D)*
M(Milling forward command P9877);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
                    :
```

\*     Block (D) is output when bit 5 (MTM) of parameter No. 9846 is
      set to 1 and a value is set for parameter No. 9088.

## (8) C-axis reverse tapping by milling, uneven spacing

```
                    :
M(Milling axis stop command P9876);
(M8/M9);
M(Milling reverse tapping pre-command P9089);
G97G98G40S_ T_ M(Forward milling axis rotation command P9877)
G80;
(Movement to the cutting start point)
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
(Second figure specification)
M(Milling reverse tapping pre-command P9089);
M(Milling forward command P9877);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
                    :
(nth figure specification)
M(Milling reverse tapping pre-command P9089);
M(Milling forward command P9877);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
                    :
```

> **NOTE**
> When bit 4 (TTM) of parameter No. 9846 is set to 0
> or when the bit is set to 0 and parameter No. 9089
> is not set, reverse rotation may be specified as the
> direction of rotation for tapping. In this case, P/S
> alarm, "3008 A WRONG TOOL SELECTED," is
> issued.

## (9) C-axis forward tapping by rigid tapping, even spacing

:
:
:
:

M(Milling axis stop command P9876);
(M8/M9);
G97G98G40T_ ;
G80;
(Movement to the cutting start point)

M(Milling axis stop command P9876);. .. .. .. . . (E1)
    or  . . . (E)*
M(Milling forward tapping pre-command P9088);. . . (E2)

S_ M(Rigid tapping mode command P5210);
G98(G84/G88)_ ;
(Second figure specification)
:
(nth figure specification)
:
G80M(Milling axis stop command P9876);
:

*   For block (E),
  - Block (E1) is output when bit 5 (MTM) of parameter No. 9846 is set to 0.
  - Block (E2) is output when bit 5 (MTM) of parameter No. 9846 is set to 1 and a value is set for parameter No. 9088. When no value is set for parameter No. 9088, no command is output.

## (10) C-axis reverse tapping by rigid tapping, even spacing

:
M(Milling axis stop command P9876);
(M8/M9);
G97G98G40T_ ;
G80;
(Movement to the cutting start point)
M(Milling reverse tapping pre-command P9089);
S_ M(Rigid tapping mode command P5210);
G98(G84/G88)_ ;
(Second figure specification)
:
(nth figure specification)
:
G80M(Milling axis stop command P9876);
:

> **NOTE**
> When bit 4 (TTM) of parameter No. 9846 is set to 0 or when the bit is set to 0 and parameter No. 9089 is not set, reverse rotation may be specified as the direction of rotation for tapping. In this case, P/S alarm, "3008 A WRONG TOOL SELECTED," is issued.

## (11) C-axis forward tapping by rigid tapping, uneven spacing

```
                    :
M(Milling axis stop command P9876);
(M8/M9);
G97G98G40T_ ;
G80;
(Movement to the cutting start point)
M(Milling axis stop command P9876); . . .. .. .. (F1)
     or   . . . (F)*
M(Milling forward tapping pre-command P9088); . . . (F2)
S_ M(Rigid tapping mode command P5210);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
(Second figure specification)
M(Milling forward tapping pre-command P9088); . . . (G)*
S_ M(Rigid tapping mode command P5210);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
                    :
(nth figure specification)
M(Milling forward tapping pre-command P9088); . . . (G)*
S_ M(Rigid tapping mode command P5210);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
                    :
```

*    For block (F),
   •    Block (F1) is output when bit 5 (MTM) of parameter No. 9846 is set to 0.
   •    Block (F2) is output when bit 5 (MTM) of parameter No. 9846 is set to 1 and a value is set for parameter No. 9088. When no value is set for parameter No. 9088, no command is output.
*    For block (G),
   Block (G) is output when bit 5 (MTM) of parameter No. 9846 is set to 1 and a value is set for parameter No. 9088. When no value is set for parameter No. 9088, no command is output.

## (12) C-axis reverse tapping by rigid tapping, uneven spacing

```
               :
M(Milling axis stop command P9876);
(M8/M9);
G97G98G40T_ ;
G80;
(Movement to the cutting start point)
M(Milling reverse tapping pre-command P9089);
S_ M(Rigid tapping mode command P5210);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
(Second figure specification)
M(Milling reverse tapping pre-command P9089);
S_ M(Rigid tapping mode command P5210);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
               :
(nth figure specification)
M(Milling reverse tapping pre-command P9089);
S_ M(Rigid tapping mode command P5210);
G98(G84/G88)_ ;
G80M(Milling axis stop command P9876);
               :
```

> **NOTE**
> When bit 4 (TTM) of parameter No. 9846 is set to 0
> or when the bit is set to 0 and parameter No. 9089
> is not set, reverse rotation may be specified as the
> direction of rotation for tapping. In this case, P/S
> alarm, "3008 A WRONG TOOL SELECTED," is
> issued.

If the reverse spindle rotation tapping pre-command is used to reverse the specified direction of spindle rotation, the icon indicating the spindle rotation status points opposite to the actual direction of spindle rotation. For this reason, to make the icon points the correct direction, set the parameter for "using an external signal to display the spindle rotation status (bit 7 (SPE) of parameter No. 9845)" to 1. Also set forward and reverse spindle rotation signals for external signal R987 as follows. The MANUAL GUIDE checks these signals and display the icon indicating the spindle rotation status.

Bit 0 of R987 = 1: Set to 1 when M03 (forward rotation) is specified.
Bit 1 of R987 = 1: Set to 1 when M03 (reverse rotation) is specified.
When both signals are set to 0, the spindle is assumed to stop.

# 4

# C-AXIS CUTTING A

# *4.1* **C-AXIS END FACE CENTER DRILLING, C-AXIS END FACE DRILLING, C-AXIS END FACE REAMING, C-AXIS END FACE BORING, C-AXIS END FACE TAPPING**



Blank
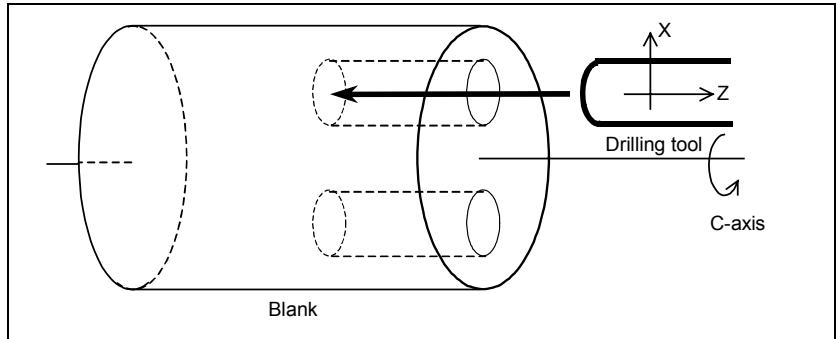
In C-axis end face drilling, center drilling, drilling, reaming, boring, and tapping can be used in the same way as for drilling by turning.

> **NOTE**
> To use C-axis boring, set bit 1 (CBR) of parameter
> No. 9765 to 1.

## *4.1.1* **Selecting the Type of Machining**

Perform the following operation to select each type of machining.

C-axis center drilling :
    Press soft-key [C-CDRL], then press [FACE].
C-axis drilling :
    Press soft-key [C-DRIL], then press [FACE].
C-axis reaming :
    Press soft-key [C-REAM], then press [FACE].
C-axis boring :
    Press soft-key [C-BORE], then press [FACE].
C-axis tapping :
    Press soft-key [C-TAP], then press [FACE].

## *4.1.2*     **Detail of Process Data**

### (1) C-axis center drilling

```
CYCLE CUTTING   * C-AXIS CENTER DRILL*

AREA          =          CUT ST Z    =

TOOL NUM      =          HOLE DIA    =

FEED/MIN      =          DWELL       =

REV/MIN       =          HOLE POS    =

S-DIRECT      =

CUT ST X      =
```

TOOL NUM
　　Number of tool used for center drilling
FEED/MIN
　　Feed amount in center drilling
REV/MIN
　　Spindle rotating speed

> **NOTE**
> 　In C-axis drilling, only FEED/MIN is available while
> 　FEED/MIN or REV/MIN can be selected for drilling
> 　by turning.

S-DIRECT
Spindle rotating direction. Select from [NORMAL] and [REVERS].
CUT ST X
　　X coordinate of the retract position for drilling multiple holes
CUT ST Z
　　Z coordinate of the retract position for drilling multiple holes
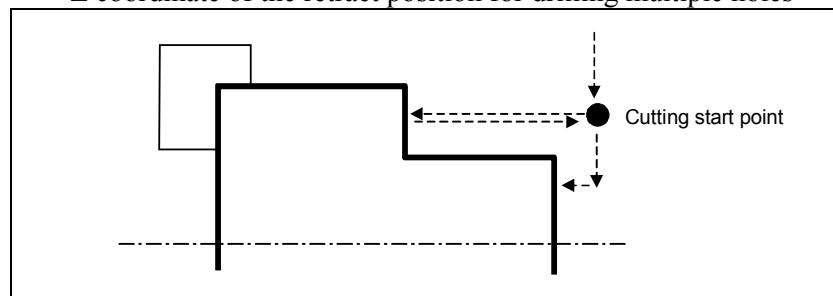


HOLE DIA
　　Nominal diameter of the tool. Used only in animated drawing.
DWELL
　　Dwell time at hole end point
HOLE POS
　　Hole position pattern. Select from [EQ INT] and [NE INT].

1.    [EQ INT]
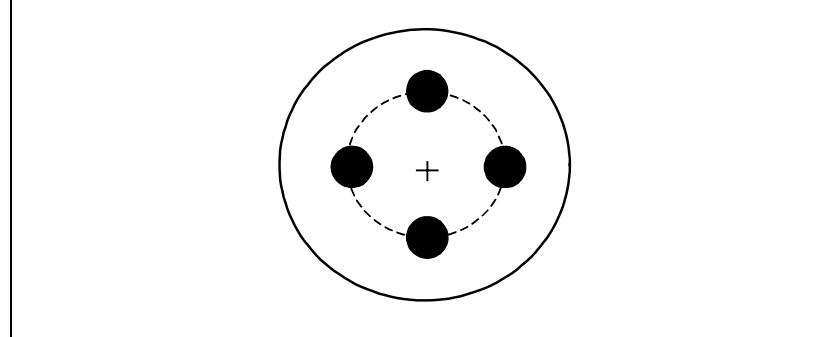


2.    [NE INT]
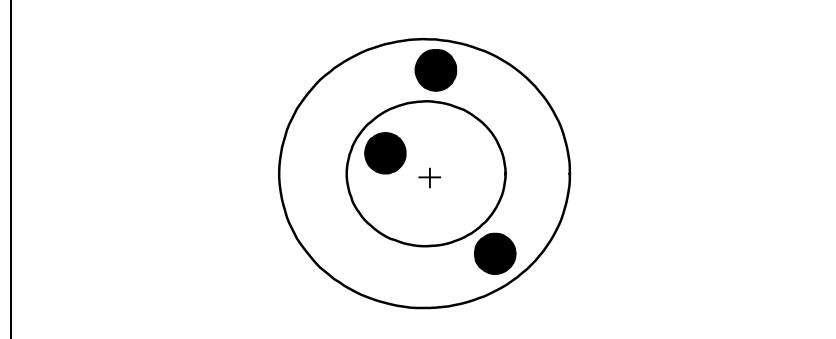


## (2) C-axis drilling

```
CYCLE CUTTING    ***C-AXIS DRILL***
AREA        =        CUT ST Z    =
TOOL NUM    =        HOLE DIA    =
FEED/MIN    =        CUT-DPTH    =
REV/MIN     =        DWELL       =
S-DIRECT    =        HOLE POS    =
CUT ST X    =        PATTERN     =
```

TOOL NUM
    Number of tool used for drilling
FEED/MIN
    Feed amount in drilling
REV/MIN
    Spindle rotating speed
S-DIRECT
    Spindle rotating direction. Select from [NORMAL] and
    [REVERS].
CUT ST X
    X coordinate of the retract position for drilling multiple holes
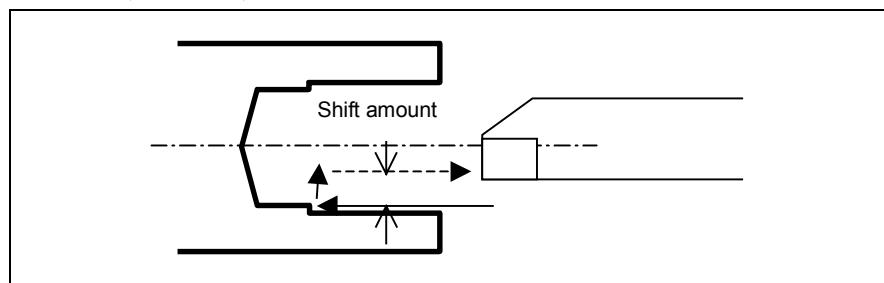CUT ST Z
    Z coordinate of the retract position for drilling multiple holes
HOLE DIA
    Nominal diameter of the tool. Used only in animated drawing.
CUT-DPTH
    Depth of the first cut in peck drilling or high-speed peck drilling.
    Automatically determined according to the hole diameter and
    value of parameter No. 9853 (DRILCF).

- 205 -

DWELL
    Dwell time at hole end point
HOLE POS
    Hole position pattern. Select from [EQ INT] and [NE INT].
PATTERN
    Select from [DRILL], [DEEP HOLE], and [RAPID DEEP].

1.    [DRILL]
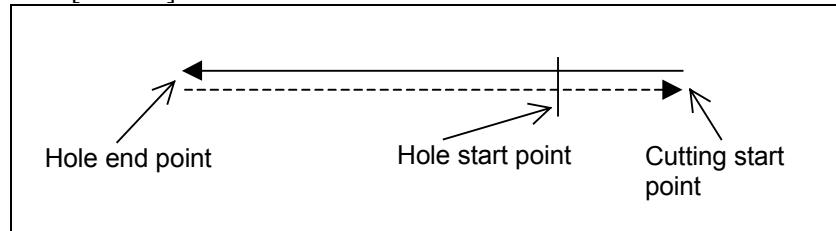


2.    [DEEP HOLE]
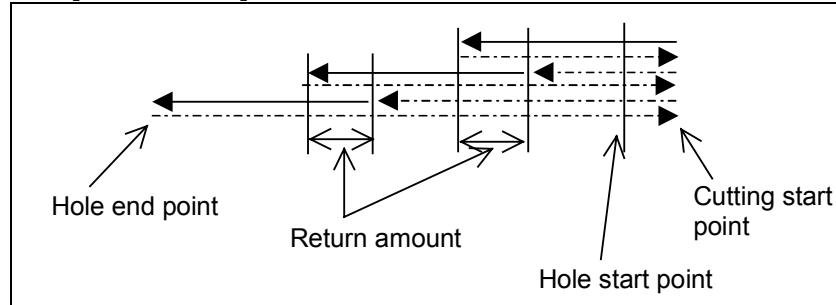


3.    [RAPID DEEP]
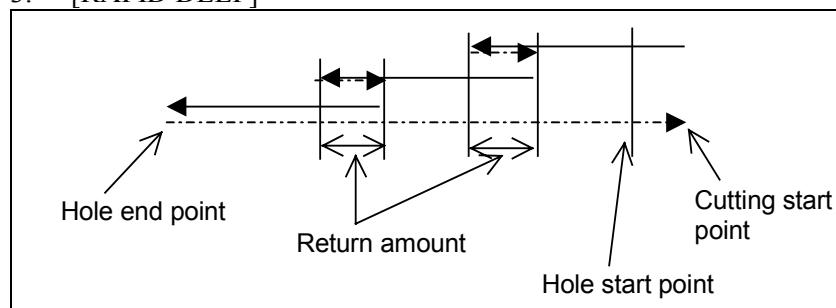


**NOTE**
    The return amount shown in the above figure is
    displayed in the detail data window.

## (3) C-axis reaming

```
CYCLE CUTTING    ***C-AXIS REAMING***
AREA        =           CUT ST Z    =
TOOL NUM    =           HOLE DIA    =
FEED/MIN    =           CHAMF-LNG   =
REV/MIN     =           DWELL       =
S-DIRECT    =           HOLE POS    =
CUT ST X    =
```

TOOL NUM
>     Number of tool used for reaming
FEED/MIN
>     Feed amount in reaming
REV/MIN
>     Spindle rotating speed
S-DIRECT
>     Spindle rotating direction. Select from [NORMAL] and [REVERS].
CUT ST X
>     X coordinate of the retract position for drilling multiple holes
CUT ST Z
>     Z coordinate of the retract position for drilling multiple holes
HOLE DIA
>     Nominal diameter of the tool. Used only in animated drawing.
CHAMF-LNG
>     Cutting-lip length of a reamer at the start of cutting. Automatically determined according to the value of parameter No. 9054 (RIMKLN).
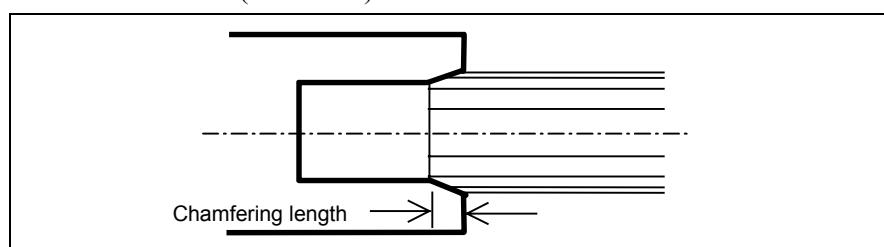


Chamfering length

DWELL
>     Dwell time at hole end point
HOLE POS
>     Hole position pattern. Select from [EQ INT] and [NE INT].

## (4) C-axis boring

```
CYCLE CUTTING    ***C-AXIS BORING***
AREA        =           CUT ST Z    =
TOOL NUM    =           HOLE DIA    =
FEED/MIN    =           SHIFT       =
REV/MIN     =           DWELL       =
S-DIRECT    =           HOLE POS    =
CUT ST X    =
```

TOOL NUM
>     Number of tool used for boring

FEED/MIN

Feed amount in boring

REV/MIN

Spindle rotating speed

S-DIRECT

Spindle rotating direction. Select from [NORMAL] and [REVERS].

CUT ST X

X coordinate of the retract position for drilling multiple holes

CUT ST Z

Z coordinate of the retract position for drilling multiple holes

HOLE DIA

Nominal diameter of the tool. Used only in animated drawing.

SHIFT

Shift amount for relief motion in boring. Automatically determined according to the value of parameter No. 9858 (BRSHFT).



DWELL

Dwell time at hole end point

HOLE POS

Hole position pattern. Select from [EQ INT] and [NE INT].

## (5) C-axis tapping

```
CYCLE CUTTING    ***C-AXIS TAPPING***
AREA        =           CUT ST Z    =
TOOL NUM    =           HOLE DIA    =
FEED/MIN    =           PITCH       =
REV/MIN     =           DWELL       =
S-DIRECT    =           HOLE POS    =
CUT ST X    =           TAP TYPE    =
```

TOOL NUM

Number of tool used for tapping

FEED/MIN

Feed amount in tapping. Automatically determined according to the pitch and speed.

REV/MIN

Spindle rotating speed

S-DIRECT

Spindle rotating direction. Select from [NORMAL] and [REVERS].

CUT ST X

X coordinate of the retract position for drilling multiple holes

CUT ST Z
> Z coordinate of the retract position for drilling multiple holes

HOLE DIA
> Nominal diameter of the tool. Used only in animated drawing.

PITCH
> Pitch of a tapping tool

DWELL
> Dwell time at hole end point

HOLE POS
> Hole position pattern. Select from [EQ INT] and [NE INT].

TAP TYPE
> Kind of tapping motion, rigid tapping or conventional tapping. Select from [RIGID] or [NORMAL].

## *4.1.3*    **Supplemental Process Data in a Pop-up Window**

### (1) C-axis center drilling

```
┌─────────────────────────┐
│                         │
├─────────────────────────┤
│ APROCH X  =             │
│ APROCH Z  =             │
│ COOLANT   =             │
│ SPDL GEAR  =            │
│                         │
└─────────────────────────┘
```

APPROACH X
  X coordinate of approaching intermediate point.
APPROACH Z
  Z coordinate of approaching intermediate point.
COOLANT
  COOLANT command at playback. Choose from the following
  soft-key menu. Default value is [ON].
  [ON]      : (M8)
  [OFF]     : (M9)
SPDL GEAR
  Select the milling gear. Directly input an M code (only when
  required).

### (2) C-axis drilling [DRILL]

```
┌─────────────────────────┐
│                         │
├─────────────────────────┤
│ ST FEED    =            │
│ ST CLEAR   =            │
│ APROCH X   =            │
│ APROCH Z   =            │
│ COOLANT    =            │
│ SPDL GEAR  =            │
└─────────────────────────┘
```

ST FEED
  Cutting feedrate override at the start of cutting (parameter 9050 :
  STFECF) The cutting feedrate is calculated as follows.
  ST FEED = (Feedrate) × (STFECF) / 100
ST CLEAR
  Clearance (distance between the tool tip and tool shoulder) at the
  start of cutting. The tool cuts the workpiece by the above
  feedrate from a point distanced from the cutting start point by an
  amount equal to this clearance. This clearance is calculated as
  follows.
  Clearance = (HOLE DIA) / (2 X Tan (nose angle/2)
  Nose angle = parameter 9083 : DRLANGL
APPROCH X
  Similar to center drilling
APPROCH Z
  Similar to center drilling
COOLANT
  Similar to center drilling

SPDL GEAR
Similar to center drilling

## (3) C-axis drilling [DEEP HOLE] / [RAPID DEEP]

```
DPTH DCRS  =           APPROCH Z  =
RET AMNT   =           COOLANT    =
MIN DEPTH  =           SPDL GEAR  =
ST FEED    =
ST CLEAR   =
APPROCH X  =
```

DPTH DCRS
Decrement of the depth of cut for peck or high-speed peck drilling. The depth of cut is reduced by this for each cut. (parameter 9850 : DRLDEC)

RET AMNT
Amount of retraction for peck or high-speed peck drilling. (parameter 9851 : DRLRET)

MIN DEPTH
Minimum depth of cut. (parameter 9852 : DRLMIN)

ST FEED
Similar to drilling [DRILL].

ST CLEAR
Similar to drilling [DRILL].

APPROCH X
Similar to center drilling

APPROCH Z
Similar to center drilling

COOLANT
Similar to center drilling

SPDL GEAR
Similar to center drilling

## (4) C-axis reaming

```
RET SPEED  =
ST FEED    =
APPROCH X  =
APPROCH Z  =
COOLANT    =
SPDL GEAR  =
```

RET SPEED
Feedrate for retraction to the cutting start point upon the completion of reaming motion. This feedrate is calculated as follows. (parameter 9860 : RMBROV)
RET SPEED = (Feed per minute) × (RMBROV) / 10

ST FEED
Similar to drilling [DRILL].

ST FEED
Similar to drilling [DRILL].

- 211 -

APPROCH X
    Similar to center drilling
APPROCH Z
    Similar to center drilling
COOLANT
    Similar to center drilling
SPDL GEAR
    Similar to center drilling

## (5) C-axis boring

```
ORIENT M    =
SFT DIRCT   =
APPROCH X   =
APPROCH Z   =
COOLANT     =
SPDL GEAR   =
```

ORIENT M
    M code for milling spindle orientation. Automatically determined according to the value of parameter No. 9056 (CBRORM).
SFT DIRCT
    Shift direction of retraction after boring. Automatically determined according to the setting of bit 2 (BSH) of parameter No. 9765. (Positive direction when this parameter is set to 0; negative direction when it is set to 1)
    To change the direction, select a desired direction by pressing soft-key [+SHIFT] or [-SHIFT].

1.    [+SHIFT]



+X

Shift amount
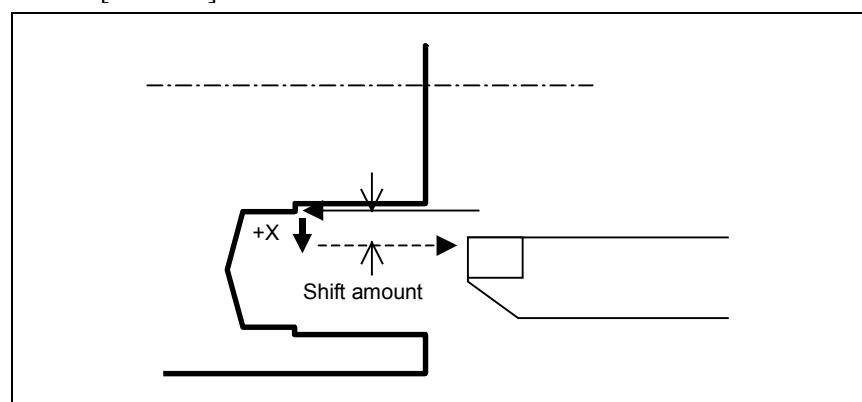
2.   [-SHIFT]
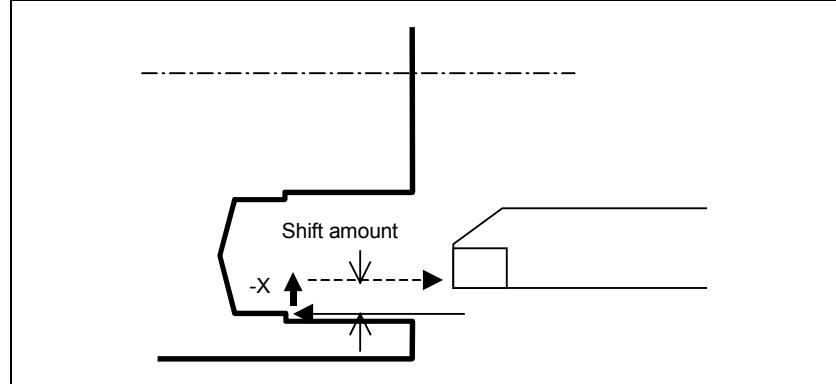


APPROCH X
      Similar to center drilling
APPROCH Z
      Similar to center drilling
COOLANT
      Similar to center drilling
SPDL GEAR
      Similar to center drilling

## (6) C-axis tapping

| |
|---|
| APPROCH X   = |
| APPROCH Z   = |
| COOLANT      = |
| SPDL GEAR   = |

APPROCH X
      Similar to center drilling
APPROCH Z
      Similar to center drilling
COOLANT
      Similar to center drilling
SPDL GEAR
      Similar to center drilling

# *4.1.4*    **Detail of Figure Data**

## (1) "Evenly spaced" figure data

START X

    X coordinate of the hole position (common)

START Z

    Z coordinate of the hole start point (common)

START C

    C coordinate of the position of the first hole

---

**NOTE**

    A cutting motion in drilling is started the amount of drilling clearance (parameter No. 9855 (GRDLCL)) before the start point.

---
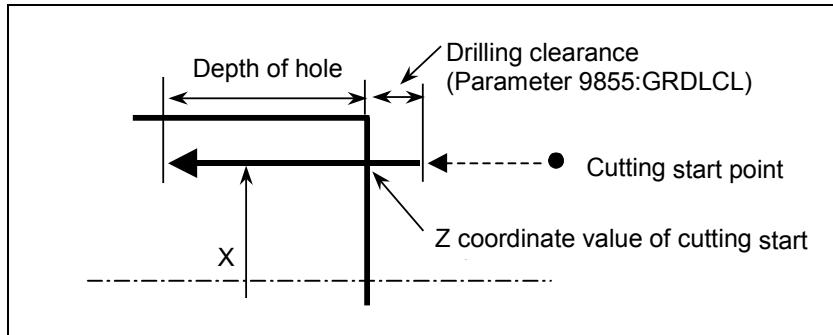


DEPTH D

    Depth of hole (common)

ANGLE A

    Angle indicating the hole position spacing

    The positive or negative direction can be specified.

H.NUM N

    Total number of holes

LST.ANG Q

    C coordinate of the position of the last hole

1.   When no data is input for ANGLE(A), H.NUM(N), and LST.ANG(Q)



- 214 -

2.    When no data is input for ANGLE(A), a value of 2 or more is
      input for H.NUM(N), and data is input for LST.ANG(Q)

Example)  X = 2 r
          C = 90.000
          N = 3
          Q = 270.000

C = 0

r

Input a diameter for START X.

3.    When data is input for ANGLE(A), a value of 2 or more is input
      for H.NUM(N), and no data is input for LST.ANG(Q)

Example)  X = 2r
          C = 90.000
          A = 90.000
          N = 3

C = 0

A

A

r

Input a diameter for START X.
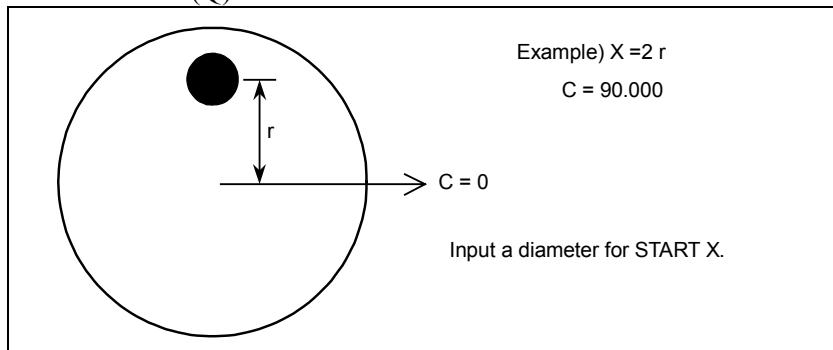
# (2) "Unevenly spaced" figure data

Up to four end face holes can be specified at any positions with any depths.

POINT n X
    X coordinate of the position of the nth hole
POINT n Z
    Z coordinate of the start point of the nth hole
POINT n C
    C coordinate of the position of the nth hole
DEPTH n D
    Depth of the nth hole (Specify this value for each hole.)

> **NOTE**
> A cutting motion in drilling is started the amount of drilling clearance (parameter No. 9855 (GRDLCL)) before the start point.
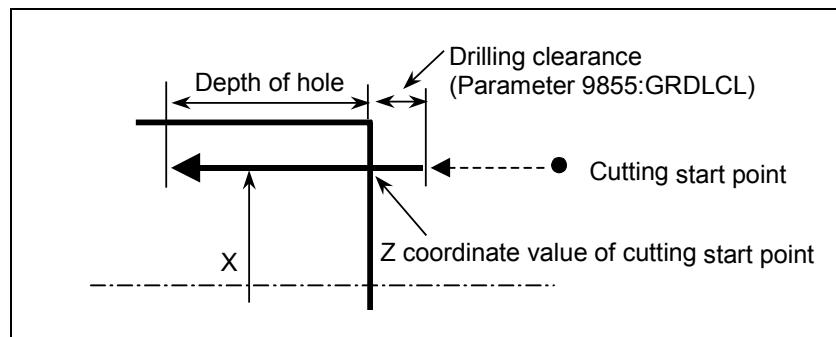


> **NOTE**
> For a hole for which not all the above figure data items are input, no cutting is executed.

# 4.2    C-AXIS SIDE FACE CENTER DRILLING, C-AXIS DRILLING, C-AXIS REAMING, C-AXIS BORING, C-AXIS TAPPING
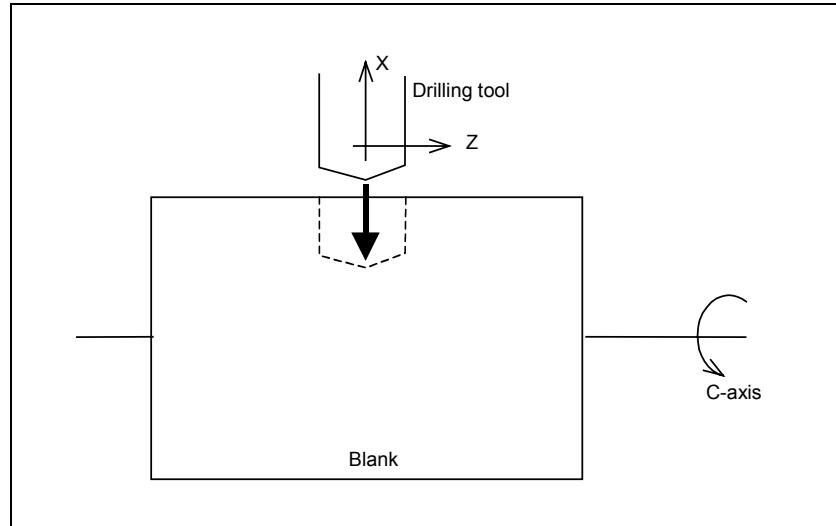


## 4.2.1    Selecting the Type of Machining

Perform the following operation to select each type of machining.
C-axis center drilling :
    Press soft-key [C-CDRL], then press [SIDE].
C-axis drilling :
    Press soft-key [C-DRIL], then press [SIDE].
C-axis reaming :
    Press soft-key [C-REAM], then press [SIDE].
C-axis boring :
    Press soft-key [C-BORE], then press [SIDE].
C-axis tapping :
    Press soft-key [C-TAP], then press [SIDE].

## *4.2.2*   **Detail of Process Data**

### (1) C-axis center drilling, C-axis drilling, C-axis reaming, C-axis boring, and C-axis tapping

CUT ST X
    X coordinate of the retract position for drilling multiple holes
CUT ST Z
    Z coordinate of the retract position for drilling multiple holes
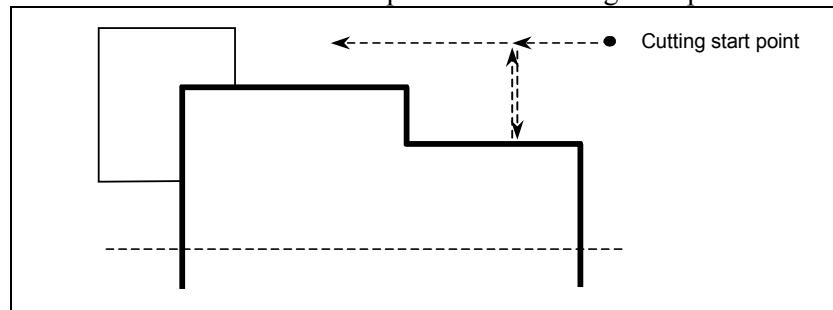


    All data displayed in process data and detail data windows other than the coordinates of the cutting start point above and data for boring below is the same as for C-axis end face drilling.
SFT-DRCT
    Shift direction of retraction after boring. Automatically determined according to the setting of bit 2 (BSH) of parameter No. 9765. (Positive direction when this parameter is set to 0; negative direction when it is set to 1)
    To change the direction, select a desired direction by pressing soft-key [+SHIFT] or [-SHIFT].
1.   [+SHIFT]
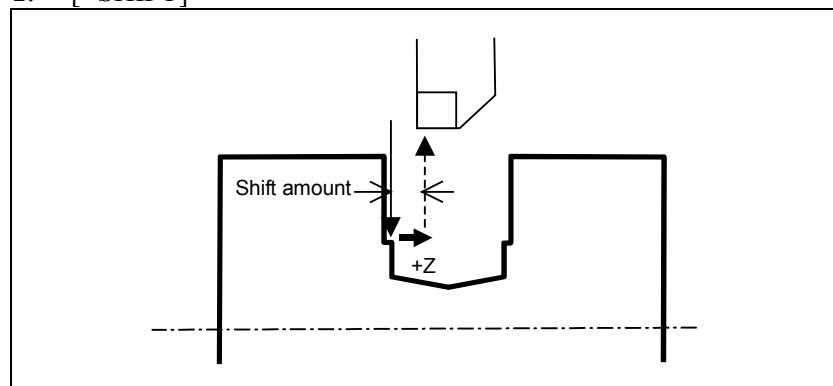
2.    [-SHIFT]
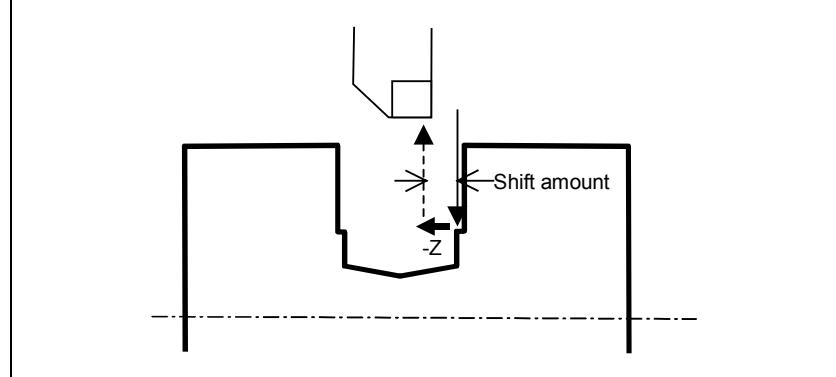
## *4.2.3*    **Detail of Figure Data**

### (1) "Evenly spaced" figure data

START X
> X coordinate of the hole position (common)

START Z
> Z coordinate of the hole start point (common)

START C
> C coordinate of the position of the first hole

---

**NOTE**
> A cutting motion in drilling is started the amount of drilling clearance (parameter No. 9855 (GRDLCL)) before the start point.

---
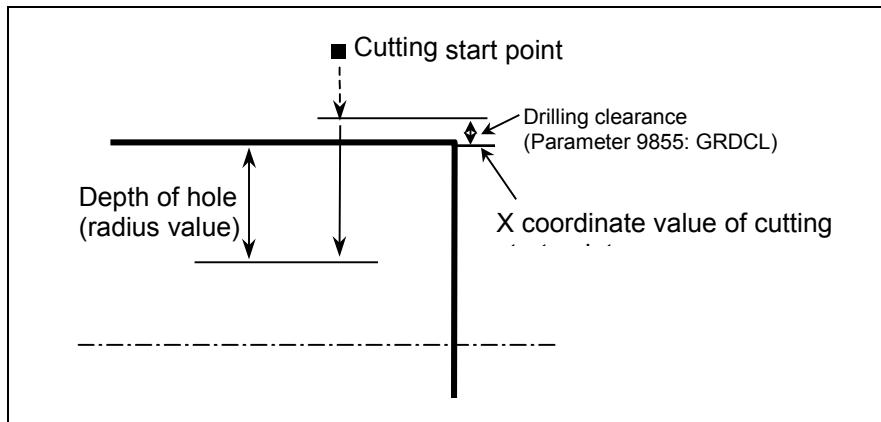


DEPTH D
> Depth of hole (radius: common)

ANGLE A
> Angle indicating the hole position spacing
> The positive or negative direction can be specified.

H.NUM N
> Total number of holes

LST.ANG Q
> C coordinate of the position of the last hole

1. When no data is input for ANGLE(A), H.NUM(N), and LST.ANG(Q)



- 220 -

2.    When no data is input for ANGLE(A), a value of 2 or more is
      input for H.NUM(N), and data is input for LST.ANG(Q)



Example)  C = 90.000
                N = 3
                Q = 270.000

3.    When data is input for ANGLE(A), a value of 2 or more is input
      for H.NUM(N), and no data is input for LST.ANG(Q)



Example)  C = 90.000
                A = 90.000
                N = 3
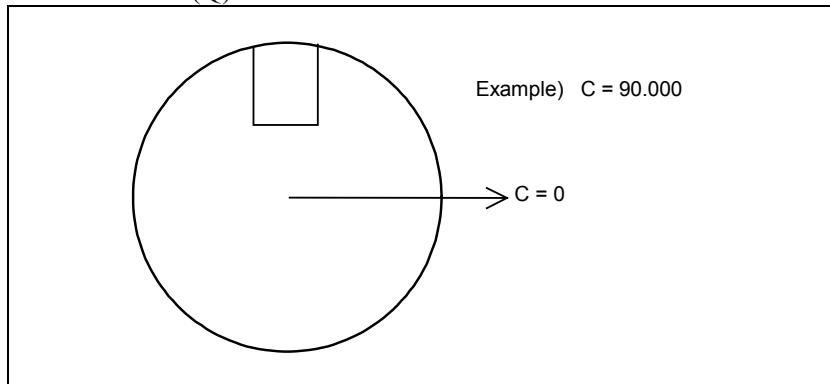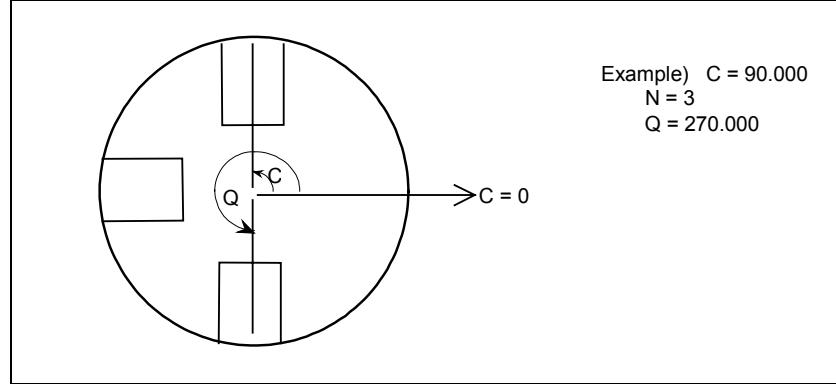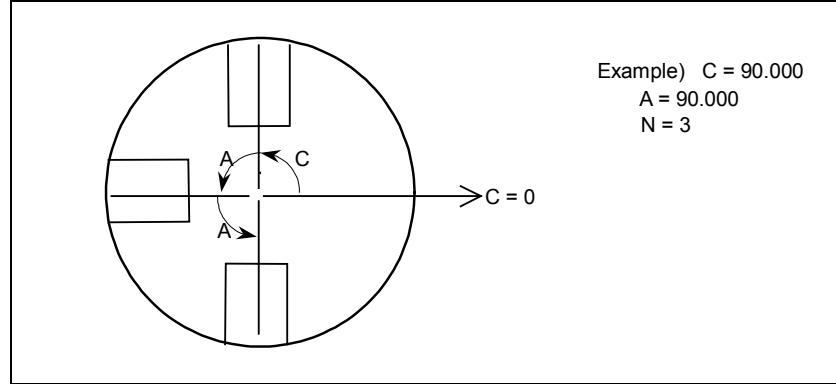
## (2) "Unevenly spaced" figure data

Up to four side face holes can be specified at any positions with any
depths.

POINT n X
      X coordinate of the position of the nth hole
POINT n Z
      Z coordinate of the start point of the nth hole
POINT n C
      C coordinate of the position of the nth hole
DEPTH n D
      Depth of the nth hole (Specify this value for each hole.)

---

**NOTE**
   A cutting motion in drilling is started the amount of
   drilling clearance (parameter No. 9855 (GRDLCL))
   before the start point.

---

- 221 -
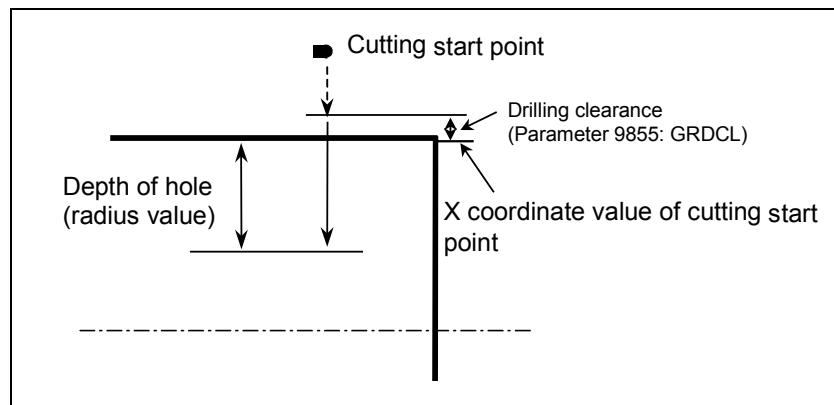
# *4.3*    **C-AXIS END FACE GROOVING**
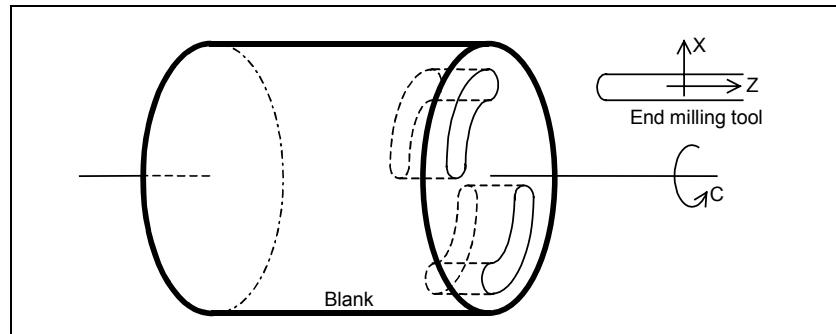


## *4.3.1*    **Selecting the Type of Machining**

Perform the following operation to select each type of machining.
C-AXIS GROOVING :
     Press soft-key [C-GROV], then press [FACE].
C-AXIS GROOVING/CHAMF :
     Press soft-key [C-GCHF], then press [FACE].

## *4.3.2*    **Detail of Process Data**

### **(1) C-axis grooving**

```
CYCLE CUTTING   ***C-AXIS GROOVING***
AREA          =        CUT ST Z    =
TOOL NUM.     =        MIL DIA     =
FEED X,C      =        SHAPE       =
FEED Z        =
REV/MIN       =
CUT ST X      =
```

AREA
>   Select a desired cutting area from the following soft-key menu:
>   [FACE] = End face of the workpiece
>   [SIDE] = Outer surface of the workpiece
>   The following soft-keys are displayed in addition to the above menu items. You can switch between C-axis grooving and C-axis groove chamfering using the specified figure data.
>   [C-GROV] = Chamfering → Grooving
>   [C-GCHF] = Grooving → Chamfering

TOOL NUM.
>   Tool number of the grooving tool (end face milling tool)

FEED X,C
>   Feedrate used for cutting the workpiece with moving the end face end milling tool sideward (along the X-axis or about the C-axis) (mm/min or inch/min)

FEED Z
>   Feedrate used for cutting the workpiece with moving the end face end milling tool axis-direction (along the Z-axis) (mm/min or inch/min)

REV/MIN
>   Milling spindle speed

CUT ST X
>   X coordinate of the retract position for cutting multiple grooves

CUT ST Z
>   Z coordinate of the retract position for cutting multiple grooves
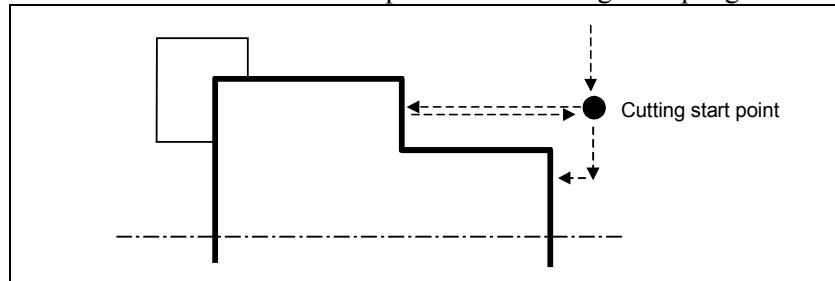


MIL DIA
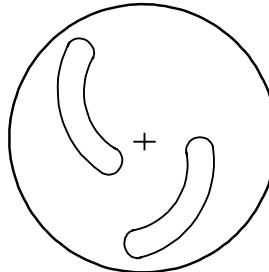>   Nominal diameter of the groove width or tool. This data is used only in animated drawing.

SHAPE
>   C-axis end face grooving type. Select a desired type from soft-keys [REGULR] and [IRREGL].

1.    [REGULR]
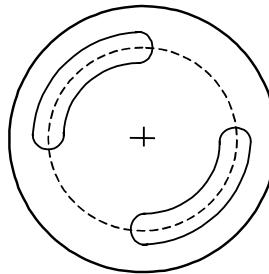


2.    [IRREGL]



## (2) C-axis groove chamfering

```
CYCLE CUTTING   *C-AXIS
GROOVING/CHAMF*
AREA        =           MIL DIA      =
TOOL NUM.   =           CHAMFER      =
FEED/MIN    =           SHAPE        =
REV/MIN     =
CUT ST X    =
CUT ST Z    =
```

TOOL NUM.
    Number of tool used for groove chamfering
FEED/MIN
    Feedrate in chamfering (mm/min or inch/min)
REV/MIN
    Milling spindle speed
CUT ST X
    X coordinate of the retract position for cutting multiple grooves
CUT ST Z
    Z coordinate of the retract position for cutting multiple grooves
MIL DIA
    Width of the groove cut before chamfering
CHAMFER
    Amount of chamfer after grooving
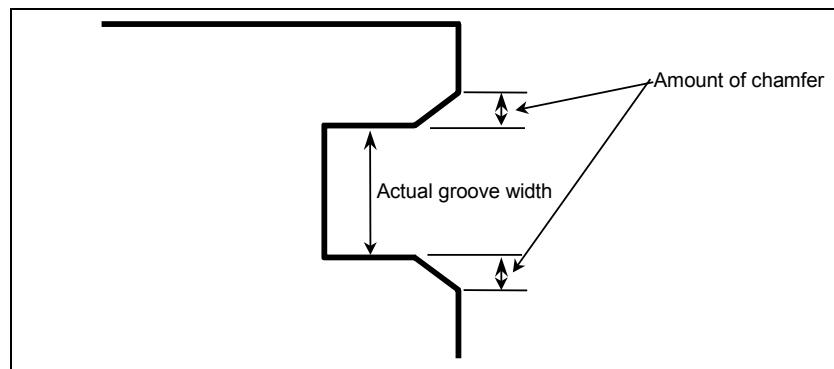
SGAPE

   C-axis end face grooving type. Select a desired type from soft-
   keys [REGULR] and [IRREGL].

## *4.3.3*    Supplemental Process Data in a Pop-up Window

### (1) C-axis grooving

| |
|---|
| APPROCH X  = |
| APPROCH Z  = |
| COOLANT     = |
| SPDL GEAR  = |

APPROCH X
    X coordinate of approaching intermediate point.
APPROCH Z
    Z coordinate of approaching intermediate point.
COOLANT
    COOLANT command at playback. Choose from the following menu. The default value is "ON".
    [ON]      : (M8)
    [OFF]     : (M9)
SPDL GEAR
    Select the milling gear. Directly input an M code (only when required).

### (2) C-axis groove chamfering

| |
|---|
| SMALL DIA    = |
| NOSE ANGL  = |
| APPROCH X  = |
| APPROCH Z  = |
| COOLANT     = |
| SPDL GEAR  = |

SMALL DIA
    Small diameter of the chamfering tool
NOSE ANGL
    Angle of the nose of the chamfering tool



**NOTE**
    Cutter compensation data in the tool file is used as the radius of the chamfering tool.

APPROCH X
    Similar to grooving

APPROCH Z
    Similar to grooving
COOLANT
    Similar to grooving
SPDL GEAR
    Similar to grooving

## *4.3.4*    **Detail of Figure Data**

### (1) "Regular" groove figure data

START X
    X coordinate of a groove (common)
START Z
    Z coordinate of the groove start point (common)
START C
    C coordinate of the first groove

> **NOTE**
> A cutting motion in grooving is started the amount
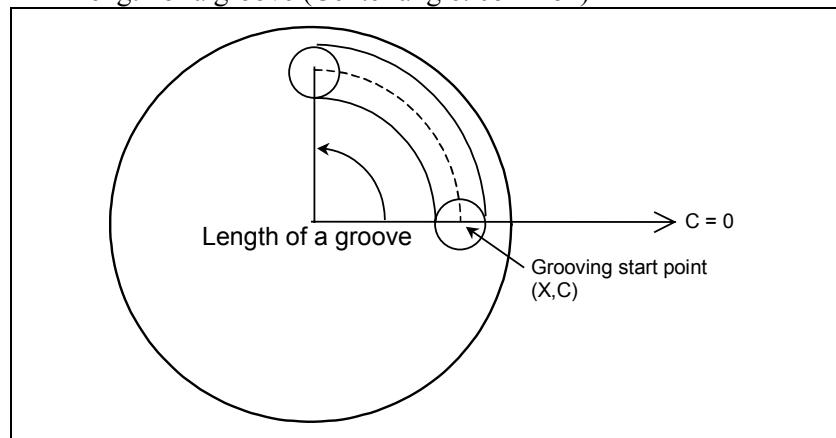> of grooving clearance (parameter No. 9855
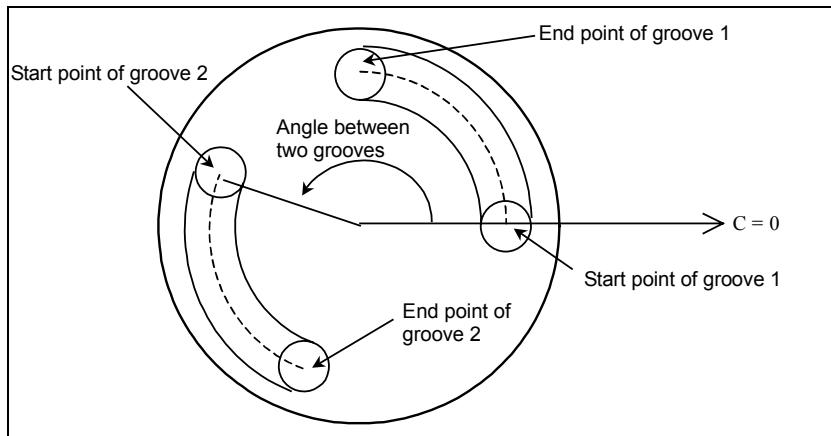> (GRDCL)) before the start point.
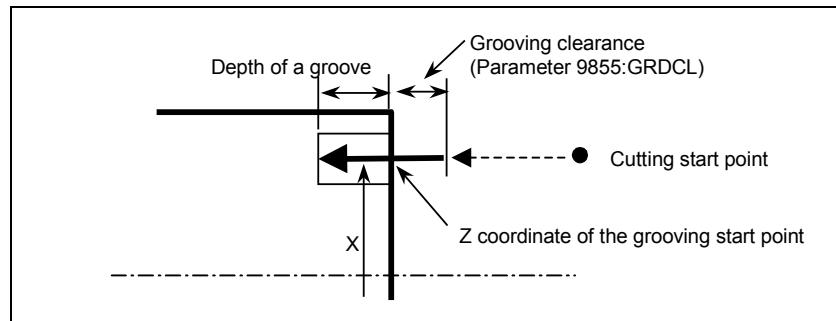


DEPTH D
    Depth of a groove (common)
GV-LNG LA
    Length of a groove (Center angle: common)



ANGLE A
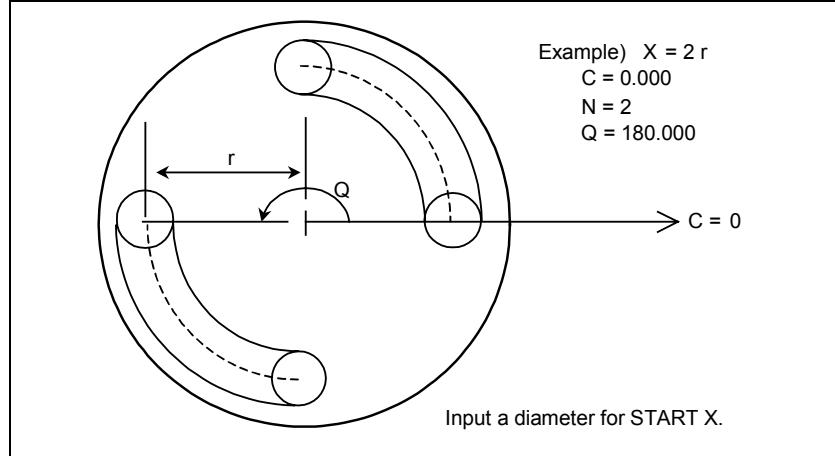    Angle between the start points of two grooves (center angle)

G.NUM N
  Total number of grooves
LST.ANG Q
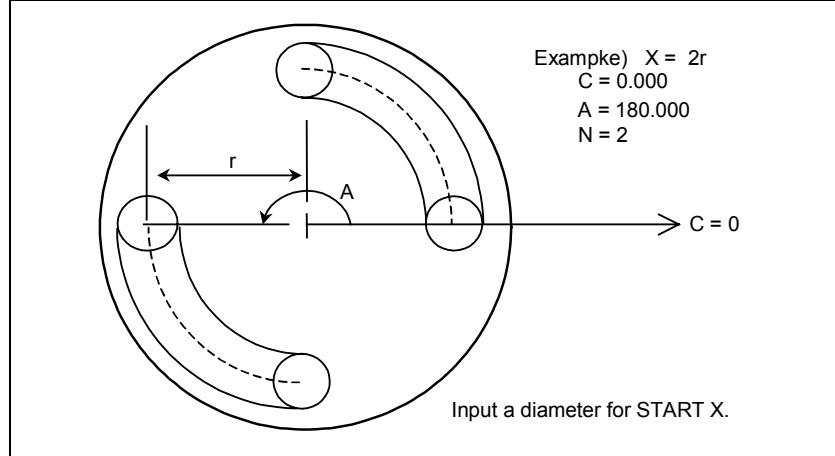  C coordinate of the start point of the last groove



1.  When no data is input for ANGLE(A), H.NUM(N), and LST.ANG(Q)

2.    When no data is input for ANGLE(A), a value of 2 or more is
      input for H.NUM(N), and data is input for LST.ANG(Q)



Example) X = 2 r
         C = 0.000
         N = 2
         Q = 180.000

Input a diameter for START X.

3.    When data is input for ANGLE(A), a value of 2 or more is input
      for H.NUM(N), and no data is input for LST.ANG(Q)



Exampke) X = 2r
         C = 0.000
         A = 180.000
         N = 2

Input a diameter for START X.

## (2) "Irregular" figure data

Up to two C-axis end face grooves can be specified at any positions
with any depths.
ST.POS n X
      X coordinate of the start point of the nth groove
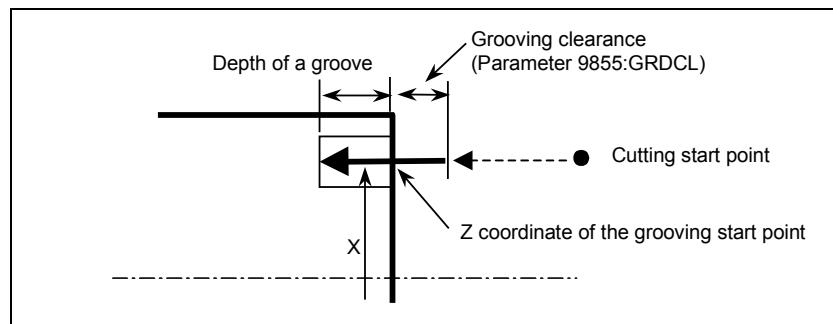ST.POS n Z
      Z coordinate of the start point of the nth groove
ST.POS n C
      C coordinate of the start point of the nth groove

---

**NOTE**
  A cutting motion in grooving is started the amount
  of grooving clearance (parameter No. 9855
  (GRDCL)) before the start point.
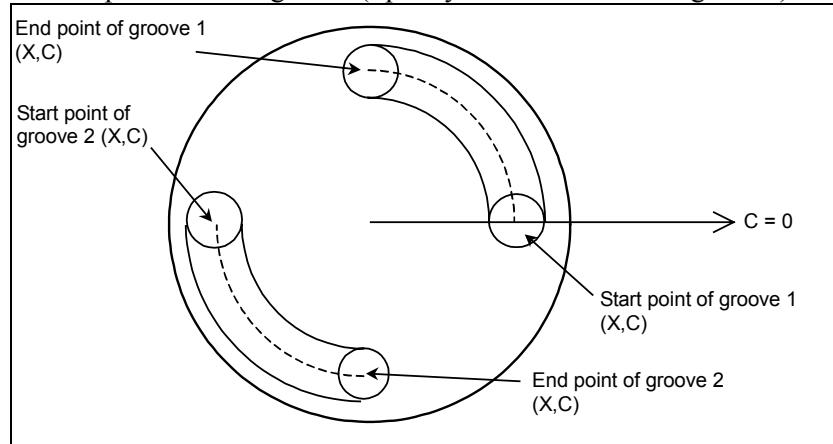
---

E.P n X/Z

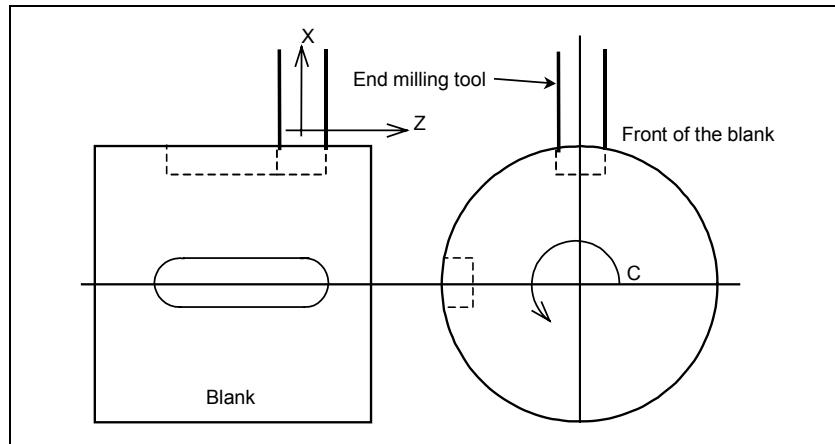    X coordinate of the end point of the nth groove

E.P n C

    C coordinate of the end point of the nth groove

DEPTH n D

    Depth of the nth groove (Specify this value for each groove.)

# *4.4*    **C-AXIS SIDE FACE GROOVING**



## *4.4.1*    **Selecting the Type of Machining**

Perform the following operation to select each type of machining.
C-axis grooving :
    Press soft-key [C-GROV], then press [SIDE].
C-axis groove chamfering:
    Press soft-key [C-GCHF], then press [SIDE].

## *4.4.2*    **Detail of Process Data**

### (1) C-axis grooving and C-axis groove chamfering

FEED Z,C

    Feedrate used for cutting the workpiece with moving the side
    face end milling tool sideward (along the Z-axis or about the C-
    axis) (mm/min or inch/min)
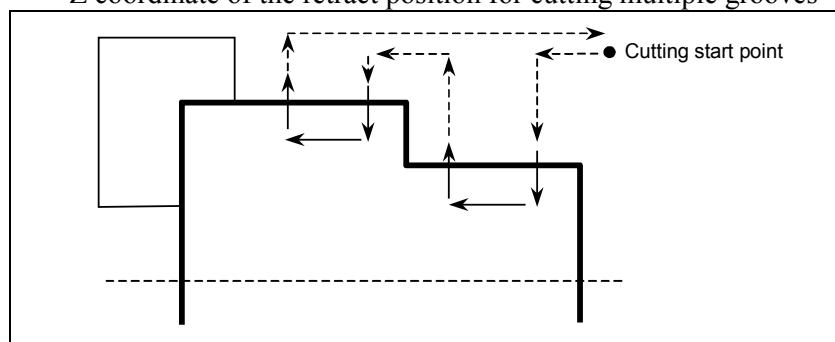
FEED X

    Feedrate used for cutting the workpiece with moving the side
    face end milling tool axis-direction (along the X-axis) (mm/min
    or inch/min)

CUT ST X

    X coordinate of the retract position for cutting multiple grooves

CUT ST Z

    Z coordinate of the retract position for cutting multiple grooves

All process data for C-axis side face grooving other than the above is the same as for C-axis end face grooving.

## *4.4.3*    **Supplemental Process Data in a Pop-up Window**

### (1) C-axis grooving

| |
|---|
| APPROCH X  = |
| APPROCH Z  = |
| COOLANT    = |
| SPDL GEAR  = |

APPROCH X
    X coordinate of approaching intermediate point.
APPROCH Z
    X coordinate of approaching intermediate point.
COOLANT
    COOLANT command at playback. Choose from the following menu. The default value is "ON".
    [ON]     : (M8)
    [OFF]    : (M9)
SPDL GEAR
    Select the milling gear. Directly input an M code (only when required).
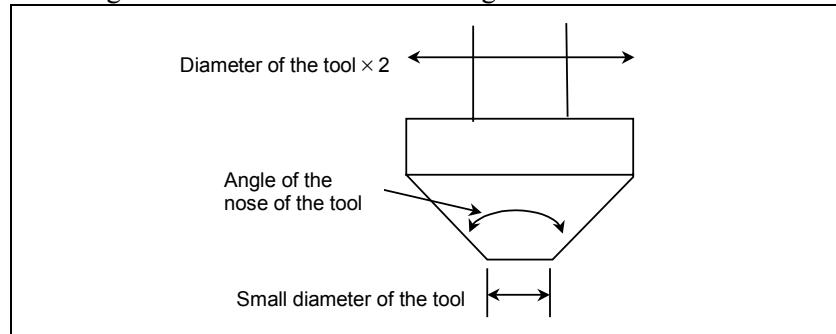
### (2) C-axis groove chamfering

| |
|---|
| SMALL DIA   = |
| NOSE ANGL  = |
| APPROCH X  = |
| APPROCH Z  = |
| COOLANT    = |
| SPDL GEAR  = |

SMALL DIA
    Small diameter of the chamfering tool
NOSE ANGL
    Angle of the nose of the chamfering tool



Diameter of the tool × 2

Angle of the
nose of the tool

Small diameter of the tool

> **NOTE**
> Cutter compensation data in the tool file is used as the radius of the chamfering tool.

APPROCH X
 Similar to grooving
APPROCH Z
 Similar to grooving
COOLANT
 Similar to grooving
SPDL GEAR
 Similar to grooving

# *4.4.4*   **Detail of Figure Data**

## (1) "Regular" groove figure data

START X

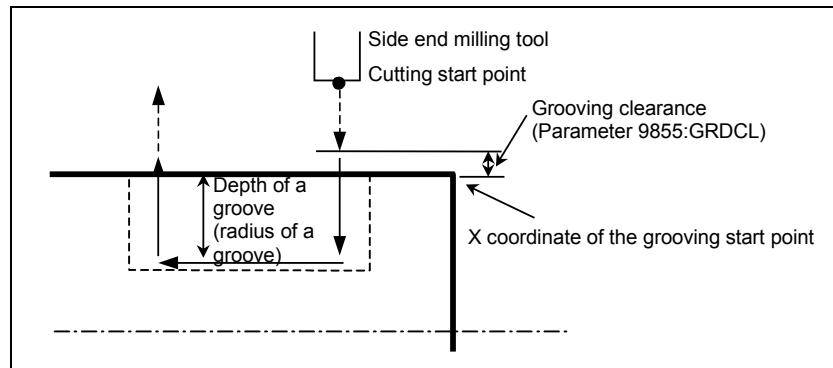    X coordinate of the groove start point (common)

START Z

    Z coordinate of the groove start point (common)

START C

    C coordinate of the first groove

> **NOTE**
>
> A cutting motion in grooving is started the amount of grooving clearance (parameter No. 9855 (GRDCL)) before the start point.
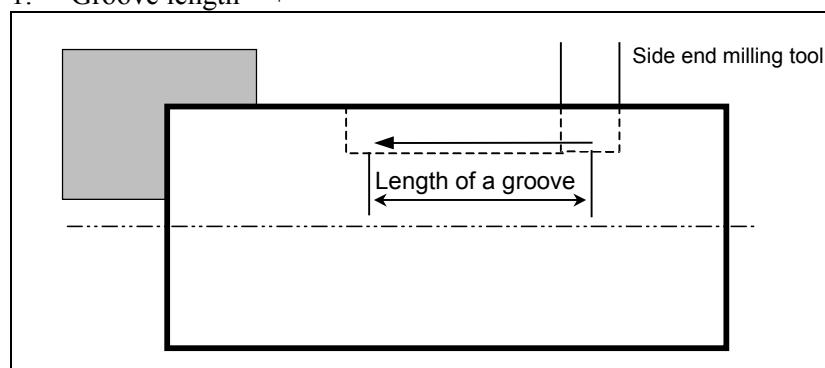


DEPTH D

    Depth of a groove (radius: common)
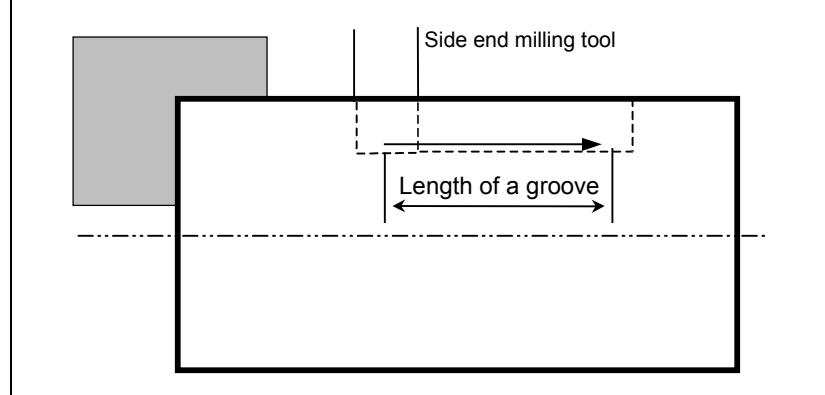
GV-LNG LA

    Length along the Z-axis of a groove (common)
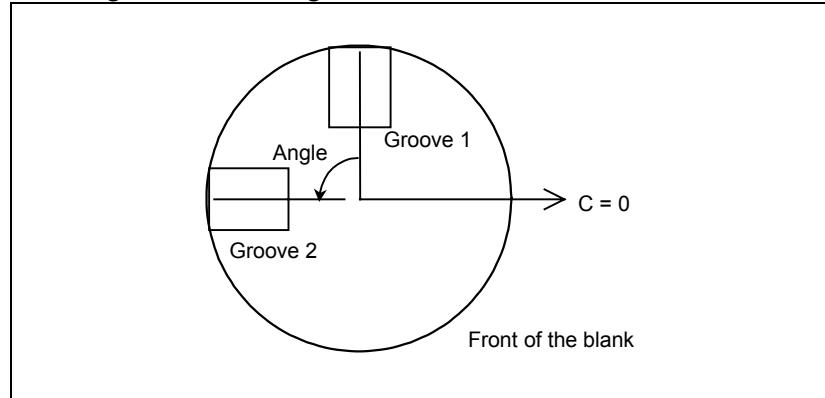
1.    Groove length = +

2.     Groove length = -

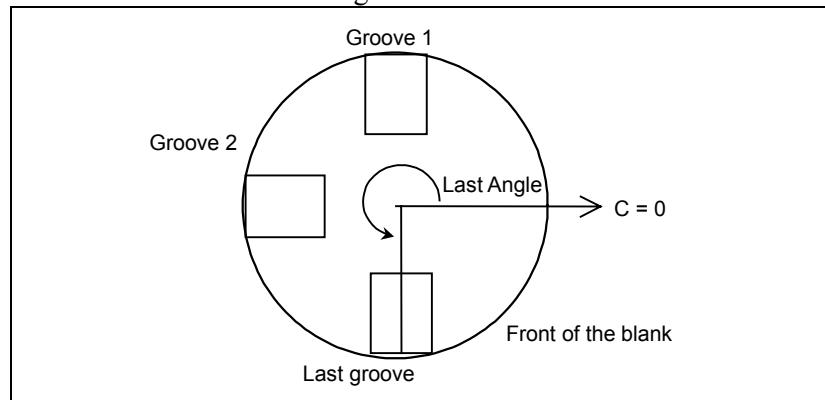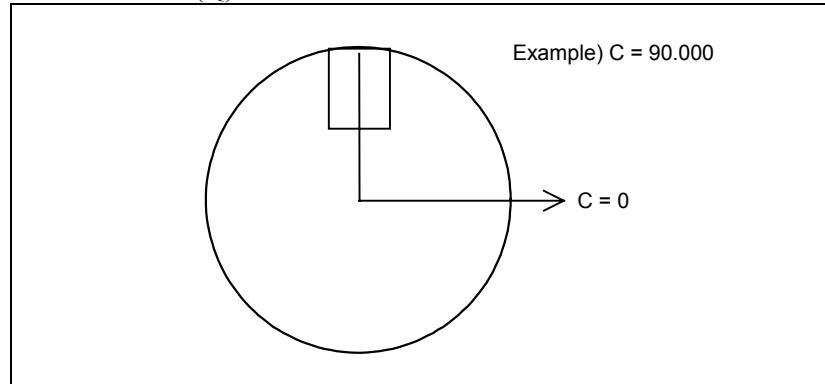

ANGLE A
     Angle between two grooves



G.NUM N
     Total number of grooves
LST.ANG Q
     C coordinate of the last groove

1.  When no data is input for ANGLE(A), H.NUM(N), and LST.ANG(Q)



Example) C = 90.000

C = 0

2.  When no data is input for ANGLE(A), a value of 2 or more is input for H.NUM(N), and data is input for LST.ANG(Q)

3.  When data is input for ANGLE(A), a value of 2 or more is input for H.NUM(N), and no data is input for LST.ANG(Q)



Example ) C = 0.000
N =
Q =180.000

C = 0

## (2) "Irregular" figure data

Up to two C-axis end face grooves can be specified at any positions with any depths.

ST.POS n X
    X coordinate of the start point of the nth groove
ST.POS n Z
    Z coordinate of the start point of the nth groove
ST.POS n C
    C coordinate of the start point of the nth groove

---

**NOTE**
  A cutting motion in grooving is started the amount of grooving clearance (parameter No. 9855 (GRDCL)) before the start point.

---

E.P n X/Z
    Z coordinate of the end point of the nth groove
E.P n C
    C coordinate of the end point of the nth groove
DEPTH n D
    Depth of the nth groove (Specify this value for each groove.)
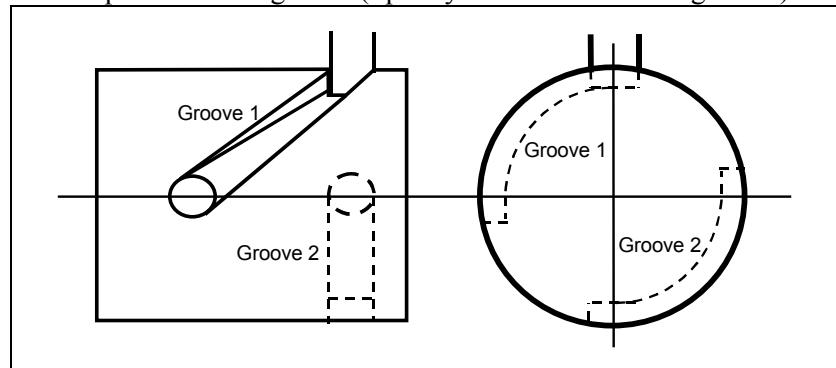
# 5

# WASTE CUTTING CANCEL FUNCTION

In bar machining (outer, inner, and end face forward machining), optimum cutting motions can be made by automatically eliminating waste motions according to the blank figure specified in advance. This function is enabled by setting bit 7 (ACC) of parameter No. 9777 and bit 2 (BSZ) of parameter No. 9761 to 1.

# *5.1*  INPUTTING A BLANK FIGURE

After creating a program for bar machining (roughing), perform the following operations to specify a desired blank figure.

The program create and edit screen cannot be used to specify a blank figure. Use the program list window shown below to specify and cancel a blank figure.

(1) Select the program list window.

| PROGRAM LIST          BLANK SIZE | |
|---|---|
| | LENGTH<br>112.500 |
| 01: CIRCLE MACHINING | OUTER DIA<br>110.000 |
| 02: LINE MACHINING | INNER DIA<br>0.000 |
| 03: BAR MACHINING | |
| 04: GROOVING | REMOVAL<br>0.000 |
| 05: DRILLING | WORK<br>BAR |

(2) Press soft-key [WORK] and position the cursor to the "WORK" item. The following soft-keys are displayed.

[PRGRM ] [ BAR ] [ROUGH ]. . .[          ] [          ] [          ]

[PRGRM ]: Moves the cursor back to the program list display section.

[ BAR ]:    Specifies a bar as the blank figure.

[ROUGH ]: Specifies a formed material as the blank figure.

When the "WORK" item in the blank window is left blank, the blank figure is assumed to be a bar. The "OUTER DIA," "INNER DIA," and "LENGTH" items in the blank size window are determined according to blank figure data when soft-key [FIGEND] is pressed.

(3) Press soft-key [ROUGH]. The following window for inputting the figure of a formed material is displayed.

```
┌─────────────────────────────────────────────────────────────┐
│ SPECIFYING ROUGH SHAPE                                        │
│                    OUTER                                      │
│  X1 �_____   Z1 _____                                   │
│  X2 �_____   Z2 _____                                   │
│  X3 �_____   Z3 _____                                   │
│  X4 �_____   Z4 _____                                   │
│  X5 �_____   Z5 _____                                   │
│   :             :                                            │
│   :             :                                            │
│  X11 _____  Z11 _____                                  │
│  X12 _____  Z12 _____                                  │
│                                                              │
└─────────────────────────────────────────────────────────────┘
```
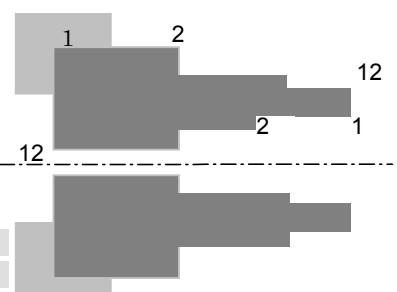
[DLTALL] [FIGEND] [          ]. . .. . .[RETURN] [       ] [        ]
(The above is an example of a window displayed for a back tool post lathe.)

[FIGEND]:

>Saves the input blank figure data into the program and closes this window. (The input blank figure data is saved into the program only when soft-key [FIGEND] is pressed.)

[DLTALL]:

>Deletes all data displayed in the window. (Only the data keyed in and stored in the key-in buffer is deleted, and the data saved into the program is not deleted.)

[RETURN]:

>Closes the window for inputting the blank figure.

Only blank figures as shown in the following examples can be specified. These blank figures are made only with vertical and horizontal lines and have the shape of a staircase whose coordinate monotonously increases (for outer machining) or decreases (for inner machining) in the direction from the end face of each blank to the chuck. The coordinates of up to 12 corner points of a blank figure can be specified for each of outer and inner machining. Select outer or inner machining using page keys [↑] and [↓].

---

**NOTE**

>When specifying the figure of a formed material for which only inner machining is to be performed, always specify the outer figure.
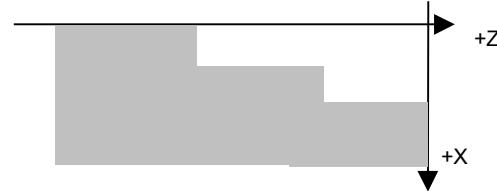>When only outer machining is to be performed, the inner figure need not be specified.

---

- Examples of blank figures that can be specified
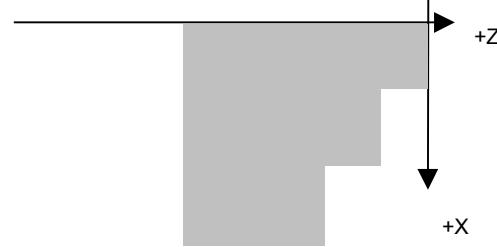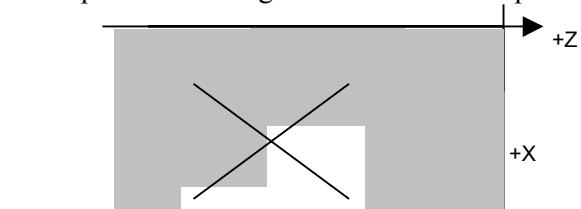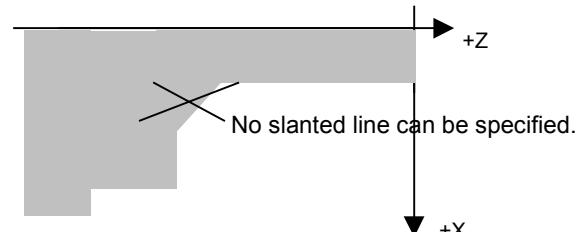  Outer machining

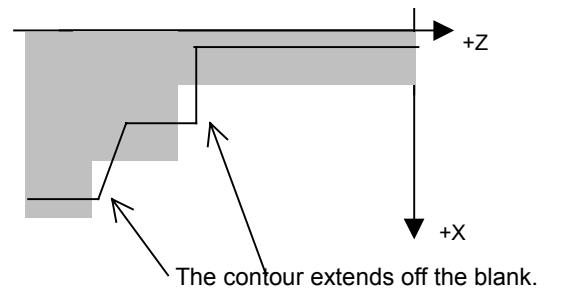Inner machining

End face machining

- Examples of blank figures that cannot be specified

The coordinate does not monotonously increase or decrease.

No slanted line can be specified.

The contour extends off the blank.

# *5.2*   **DETAILS OF CUTTING MOTIONS**

When the parameter for enabling the waste cutting cancel function is set and a formed material is specified as the blank figure, the bar machining process is executed with automatically eliminating the cutting motion for each area in which there is no blank as shown in the figure below.
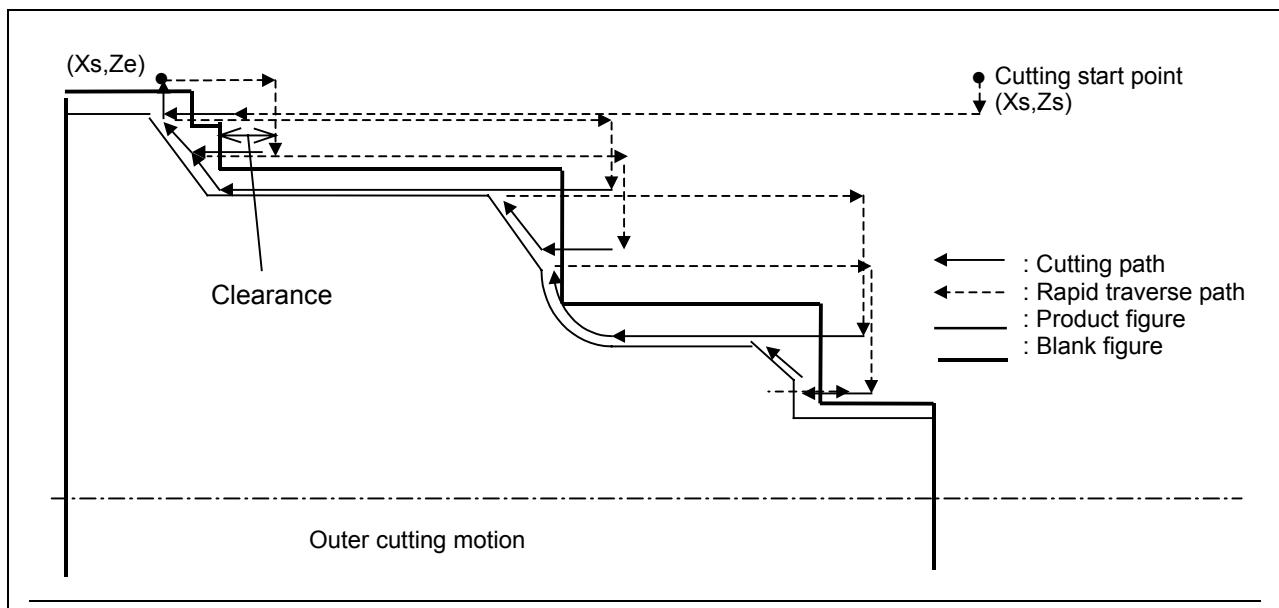


START X/Z

Also determined as described in the explanation of parameters Nos. 9784 and 9785 for a formed material.

APPROCH X/Z

Also determined as described in the explanation of parameters Nos. 9782 and 9783 for a formed material.

**NOTE**

Pressing soft-key [CALC.] in the figure data window at program editing recalculates the cutting start point and approaching points using the input blank size of the formed material.

Clearance

The cutting start position following retraction is determined according to the value of parameter No. 9784 (outer/inner machining) or No. 9785 (end face machining).

**NOTE**

These parameters are also used for automatically determining the cutting start point.
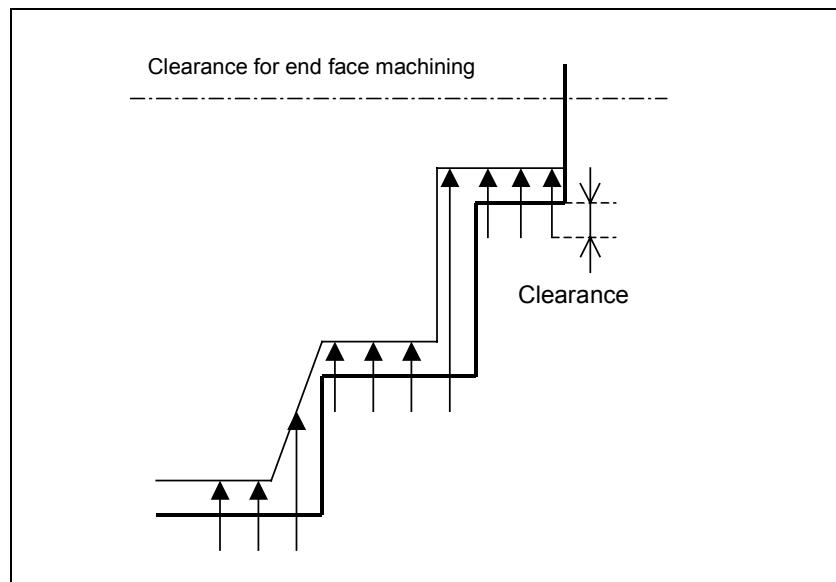
# 6

# RETHREADING

Rethreading is a function of cutting a worn-out or partially broken thread again. A thread is cut again by inputting thread cutting data and teaching the position of the thread. This function requires the function of controlling the spindle orientation to synchronize the tool feed with the mounted thread.

# *6.1* OVERVIEW

Applicable threads
- Following thread types
  General, metric, unified, piping PT, and PF
- Straight and tapered threads
- Single and multiple threads
- Normal (right-hand) thread and reverse (left-hand) thread

Spindle orientation
  Perform spindle orientation before teaching the thread position.

Input data
  Input data required for rethreading in the process data window for ordinary threading. Specify "REPAIR" as the "METHOD" input item to perform rethreading.
  Rethreading requires the following data items like ordinary threading:
  Process data: TYPE, METHOD, ANGLE, and LEAD
  Figure data: START X, START Z, END X, and END Z
  For details of each item, see the corresponding chapter in this operator's manual.

Teaching the thread position
  Teach the thread position to synchronize the tool feed with the mounted thread. Touch the tool on the bottom of an appropriate thread and press soft-key [SYNC P] to teach the thread position. The taught position is used to provide timing to start feeding the tool at the cutting start point.

Tool motions
  The tool moves for rethreading as shown in the figure below:
  a) Moves from the current tool position to the approaching point.
  b) Moves from the approaching point to the cutting start point.
  c) Performs threading.
  For details of each point, see the corresponding chapter of this operator's manual.
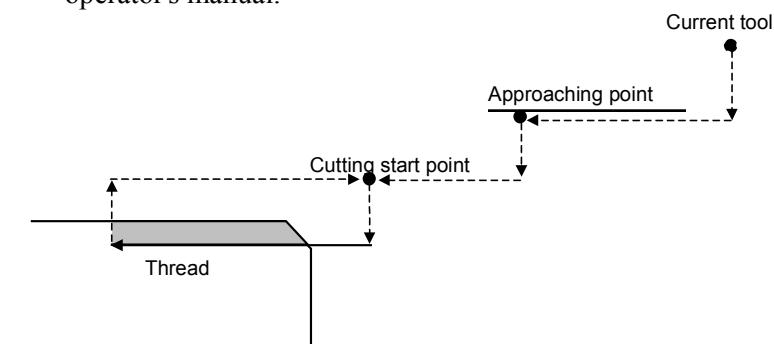


Synchronization
  To synchronize the tool feed with the mounted thread, the tool is made wait for the start of tool feed at point A shown in the figure below by the number of spindle rotations that is obtained by the following equation:
  $Q = Qs + Qz - Qe - Qp$ (degree)

Qs:  Spindle angular displacement for providing timing to start feeding the tool at point A and synchronize the tool feed with the phase of the thread
Qs= 360(Lead-r)/Lead
Lead , r : Remainder of L/Lead
L : Distance between the taught thread position and point A

Qz :  Angle indicating the difference between the spindle one-rotation signal detection position and spindle orientation stop position. Qz differs depending on the machine. Set Qz for parameter No. 9844.
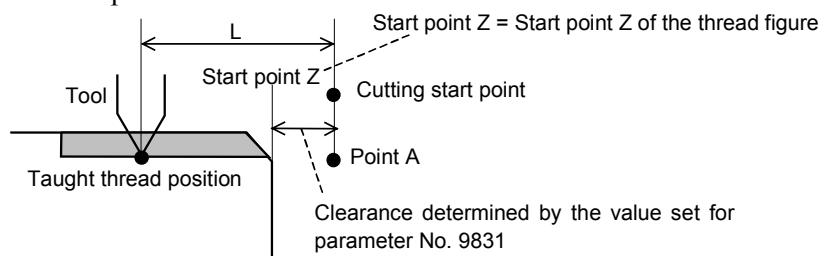


Qe :  Spindle angular displacement equivalent to the amount of the position error of the servo axis (Z)

$Qe = \Delta Ez \times 360 / Lead$

$\Delta Ez$: Amount of the servo position error at the threading feedrate

Qp :  Spindle angular displacement equivalent to the amount of accumulation during acceleration
This value can be changed by setting a new value for the "SHIFT" figure data input item when required.

$Qp = Shift\ amount \times 360 / Lead$

Shift angle Q above is calculated with the user macro (O1041). If the tool feed cannot be synchronized with the phase of the thread, the machine tool builder can change the expression according to the machine.

# *6.2*    **OPERATION**

(1) Input process data.

When rethreading is selected, the default values are automatically set for the input items below as follows:

CUT AMUNT/CUT TIMES :

CUT TIMES is selected and 1 is set.

MULTI. : 1 is set.

For a multiple thread, input the number of threads. For multiple cuts, input the number of cuts or the depth of cut.

(2) When the cursor is positioned at the "METHOD" input item in the process data window, press soft-key [REPAIR].

[S-AMNT] [B-AMNT] . . ... [          ] [REPAIR]

When rethreading is performed according to the value input for "CUT AMUNT/CUT TIMES", " BOTH AMOUNT " is automatically selected for "METHOD".

(3) Manually input the lead of the thread.

(4) When the cursor is positioned in the figure data window and rethreading is specified, soft-keys [ORIENT] and [CAN OR] are displayed.

Press soft-key [ORIENT] to perform spindle orientation. At this time, the M code for orientation set for parameter No. 4960 is output.

(5) Teach the figure of the thread or directly input numeric values to specify the figure.

After inputting process data, the operator should directly input the values for START X, START Z, END X, and END Z for the thread figure or teach the thread figure as follows to specify the figure:
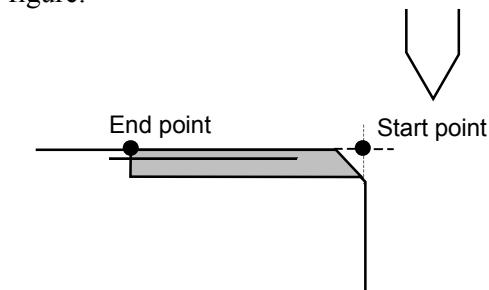


- Move the tool to the thread start point.
- Position the cursor to the "START X" input item.
- Press soft-key [RD POS].

[RD POS] [SYNC P] [         ] [         ] . . .[         ] [         ]

At that time, the current workpiece coordinate X is set for the "START X" input item.
- Set other figure data in the same way.

(6) Teach the thread position to synchronize the tool feed with the thread.

Before this operation, at least START X and START Y must be set for the figure.
- Touch the tool on the bottom of the thread to be rethreaded.
- Press soft-key [SYNC P].

At this time, angular displacement Q for synchronization with the thread described above is calculated and the calculation result and taught position are displayed as follows:

```
*                                              *
SHIFT   =            ←Calculation result Q     START X  =
TYPE    =                                      START Z  =
METHOD  = REPAIR                               END X    =
ANGL       =                                   END Z    =
LEAD    =                                      SHIFT OFS  =      ←Amount of accumulation during acceleration Qp
                                               SYNC POS   =      ←Taught thread position
```

If threading is executed before soft-key [SYNC P] is pressed, macro alarm, "3001 CHECK PROCESS DATA," is issued.

(7) Manually remove the tool from the thread and move it to a safe position at which cutting can be started. Then, press soft-key [CAN OR] to cancel spindle orientation.

When soft-key [CAN OR] is pressed, the M code for canceling orientation set for parameter No. 4961 is output. If the tool is not at the distance specified for parameter No. 9843 from the thread, the warning message, "MOVE TOOL AWAY FROM THREAD," is displayed.

(8) Press soft-key [EXEC] or the [START] button on the machine operator's panel to execute rethreading.

When rethreading is selected, process data "CUT TIMES" is automatically set to 1. For this reason, rethreading is performed only once in ordinary cases. The operator can change the value set for "CUT TIMES/CUT AMUNT", however. When the value is changed, rethreading is executed according to the specified value.

To use this function, the parameter 9848 bit 7 (TRP) must be set to 1.

# APPENDIX

# A  PARAMETERS

> ⚠ **WARNING**
> Be sure to use the parameters set by the machine tool builder.
> If you change the setting of a parameter, the machining program may not work correctly.
> If the machining program does not work correctly, the tool may bump against the workpiece, and the machine may be forced to perform unnatural machining, possibly causing damage to the tool and/or machine, and even injuries.

# *A.1*     PARAMETERS FOR DRILLING (1)

> **NOTE**
> In addition to the parameter listed below, parameter Nos.9850 to 9862 are provided for drilling.

| 9050 | STFECF |
|------|--------|

STFECF     Cutting feedrate override at the start of cutting in drilling.
Valid data range : 0 to 255    Units : 1%

| 9054 | RIMKLN |
|------|--------|

RIMKLN     Cutting-lip length to be set automatically at the start point of cutting in reaming (chamfer length on the program screen).
Valid data range    :    0 to 99,999,999
Units :    0.001mm
           0.0001inch

| 9056 | CBRORM |
|------|--------|

CBRORM     M code for spindle orientation in C-axis boring
Valid data range    :    0 to 999

# *A.2*   **PARAMETERS FOR MANUAL GUIDE PERIPHERAL FUNCTIONS**

| 9080 | ANMTWO |
|---|---|

ANMTWO    Shank width of an outer general tool for animated drawing only.
          Valid data range  :    0 to 99,999,999
          Units :    0.001mm
                     0.0001inch

| 9081 | ANMTWI |
|---|---|

ANMTWI    Shank width of an inner general tool for animated drawing only.
          Valid data range   :    0 to 99,999,999
          Units :    0.001mm
                     0.0001inch

| 9082 | ANMTWF |
|---|---|

ANMTWF    Shank width of a face general tool for animated drawing only.
          Valid data range   :    0 to 99,999,999
          Units :    0.001mm
                     0.0001inch

| 9083 | ANMTAN |
|---|---|

ANMTAN    Nose angle of a drilling tool for animated drawing only.
          Valid data range : 0 to 180
          Units : 1 degree

| 9090 | SPDLMX |
|---|---|

SPDLMX    Maximum spindle speed used for constant surface speed control.
          G50 S(SPDLMX) ; is outputted at the top of each program in
          MANUAL GUIDE format program. And also, this is used for the
          default value for an external CSN (CSS mode command) switch on a
          machine operator's panel.
          Valid data range : 0 to 65535
          Units : 1

| 9091 | INITTCD |
|---|---|

          Default value of a tool number used for an external T-code switch on
          a machine operator's panel. This value is applied when no data is
          inputted in a guidance or single cutting process, or on an initial
          screen.
          Valid data range : 1 to 99
          Units : 1

- 255 -

| 9092 | INISMIN |
|------|---------|

INISMIN   Default value of a spindle speed used for an external CSN(CSS mode command) switch on a machine operator's panel. This value is applied when no data is inputted in a guidance or single cutting process, or on an initial screen.
Valid data range : 0 to 65535
Units : 1

| 9093 | INISRPM |
|------|---------|

INISRPM   Default value of a spindle speed used for an external CSF(CSS mode cancel command) switch on a machine operator's panel. This value is applied when no data is inputted in a guidance or single cutting process, or on an initial screen.
Valid data range : 0 to 65535
Units : 1

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|------|----|----|----|----|----|----|----|----|
| 9640 | | | | PR5 | PR4 | PR3 | PR2 | PR1 |

PRn   1 : Enables custom process n (= 1 to 5).
0 : Disables custom process n (= 1 to 5).

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|------|----|----|----|----|----|----|----|----|
| 9641 | | | | | EFV | SM5 | NT0 | RF2 |

RF2   1 : It is available to change the parameter 1241 for setting the second reference position in Manual Guide Screen.
0 : Disables to change the parameter 1241 in Manual Guide Screen.

NT0   1 : After finish machining, output to with the command of moving a tool to the approach point before returning to the tool change position.
0 : After finish machining, output to as one block command.

SM5   1 : In tapping, output M05 as one block command after the command of G32.
0 : In tapping, output M05 with the command of G32 as one block command.

EFV1   1 : Enables feedrate override and cutting amount override in case of the end face cutting with outer bar machining.
0 : Disables the above function.
    <Related parameters : No. 9643, 9644>

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|------|----|----|----|----|----|----|----|----|
| 9642 | | | | | | | | SLM |

SLM   1 : Spindle load meter displays the spindle load data which is read from PMC data R991.
    PMC data R991 has to be 8 bit data with binary format.
0 : Spindle load meter displays the CNC's spindle load data.

| 9643 | FACFOV |
|------|--------|

FACFOV   Feedrate override for the end face cutting with executing outer bar machining.
Valid data range : 0 to 255 (when 0 is set, it means 100%.)
Units : %
<Related parameter : No. 9641 bit3 (EFV)>

| 9644 | CUTAOV |
|------|--------|

CUTAOV   Cutting amount override for the end face cutting with executing outer bar machining
Valid data range : 0 to 255 (when 0 is set, it means 100%.)
Units : %
<Related parameter : No. 9641 bit3 (EFV)>

| 9645 | PRGFMC |
|------|--------|

PRGFMC   M-code which is output at the program top as the following example.
O0002 (BAR/THREAD) ;
G80 ;
G40 ;
M45 ; ← In case of P9645=45
G30 L10 W0 ;
Valid data range : 0 to 32767
Units : 1

# *A.3*    USER PARAMETERS

| 9646 | Bit type parameter |
|------|---------------------|

to

| 9655 | Bit type parameter |
|------|---------------------|

| 9656 | Word type parameter |
|------|----------------------|

to

| 9685 | Word type parameter |
|------|----------------------|

> **NOTE**
> For details of parameter Nos.9646 to 9685, refer to the manual supplied by the machine tool builder.

## *A.4* **BIT TYPE PARAMETERS FOR MANUAL GUIDE COMMON USE**

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9760 | NPL | NCO | G30 | APZ | MGE | | | |

MGE  1 :  The following block is output at the end of last cutting process.
                G97 M05;
                M09;
        0 :  The above function is disabled.

APZ  1 :  Approaching point Z for bar machining, threading, and grooving is calculated using the cutting start point as follows:
           Z=[START Z] + Parameter [OTSFPZ] or [INSFPZ]
           [OTSFPZ] : Parameter 9781
           [INSFPZ] : Parameter 9783
        0 :  The above function is disabled. For how the approaching point is calculated, see the explanation of parameter No. 9783.

G30  1 :  G30 is not executed in each machining processes.
        0 :  G30 is executed.

NCO  1 :  Conversion of a machining program into NC program is valid.
        0 :  This cannot be used.

NPL  This must be set to 1 always.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9761 | AFR | SEM | SRV | AWD | TCP | BSZ | PWN | UWN |

UWN  1 :  User window in an initial screen is available.
        0 :  This window is not available.

PWN  1 :  [PRESET] in an initial screen is not available.
        0 :  [PRESET] can be used.

BSZ  1 :  A window for setting the blank size is displayed in the program list window. The blank size is not determined automatically. It is set after a program is created or edited.
        0 :  The above function is disabled. The blank size is automatically determined when a new program is created. It is not automatically determined again when a program consisting of multiple processes is created or edited.

TCP  1 :  Approaching point data is treated as tool-change point data. (This function is not enabled simply by setting this parameter. To enable this function, also customize a macro.)
        0 :  The above function is disabled.

AWD  1 :  An alarm window is displayed on the MANUAL GUIDE screen.
        0 :  The above function is disabled.

SRV  1 :  Soft-keys for a near-end tool post lathe are displayed.
        0 :  Soft-keys for a back tool post lathe are displayed.

SEM  1 :  Semifinishing only can be specified for bar machining (roughing). When this parameter is set to 1, bit 5 (RFN) of parameter No. 9772 is invalidated.
        0 :  The above function is disabled.

AFR   1 :   The feed per revolution is displayed for actual feedrate F in the status indicator window.
      0 :   The above function is disabled.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9762 | LG1 | LG2 | LG3 | LG4 | NCA | NGS | TCA | NFK |

NFK   1 :   The function keys are disabled on the MANUAL GUIDE screen. To select a CNC screen such as POS or OFFSET, press both ends of the related soft-key.
      0 :   The function keys are enabled on the MANUAL GUIDE screen.
TCA   1 :   In linear or circular guidance cutting, approach and guidance cutting motions are automatically stored as different blocks according to the nature of each motion and taught in the CNC. Up to 30 blocks can be created at a time.
      0 :   The above function is disabled.
NGS   1 :   The soft-keys for selecting guidance cutting and single cutting are not displayed.
      0 :   The above soft-keys are displayed.
NCA   1 :   NC statements are also displayed on the machining simulation screen during machining or drawing.
      0 :   No NC statements are displayed in the above case.

> **NOTE**
> When this parameter is set to 1, the tool may temporarily stop along the axis at transition from one block to the next depending on the feedrate. In this case, set this parameter to 0.

LG1   1 :   Use the extension language 1.
      0 :   This language is not use.
LG2   1 :   Use the extension language 1.
      0 :   This language is not use.
LG3   1 :   Use the extension language 1.
      0 :   This language is not use.
LG4   1 :   Use the extension language 1.
      0 :   This language is not use.
      If some parameters set to 1 at the same time, the lower parameter is in effect.

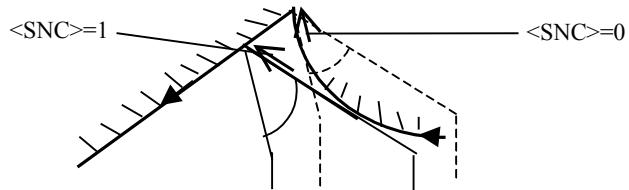| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9763 | DRL | CNT | TGF | TGR | GRV | TRD | BAF | BAR |

Each bit type parameter shown blow can be used to disable the soft-key for selecting the corresponding machining type.

BAR   1 :   The soft-key for selecting bar machining (roughing) is disabled.
      0 :   The above soft-key is enabled.
BAF   1 :   The soft-key for selecting bar machining (finishing) is disabled.
      0 :   The above soft-key is enabled.

TRD  1 :  The soft-key for selecting threading is disabled.
     0 :  The above soft-key is enabled.
GRV  1 :  The soft-key for selecting normal grooving is disabled.
     0 :  The above soft-key is enabled.
TGR  1 :  The soft-key for selecting trapezoidal grooving (roughing) is disabled.
     0 :  The above soft-key is enabled.
TGF  1 :  The soft-key for selecting trapezoidal grooving (finishing) is disabled.
     0 :  The above soft-key is enabled.
CNT  1 :  The soft-key for selecting center drilling is disabled.
     0 :  The above soft-key is enabled.
DRL  1 :  The soft-key for selecting drilling is disabled.
     0 :  The above soft-key is enabled.

|      |      | #7  | #6  | #5  | #4  | #3  | #2  | #1  | #0  |
|------|------|-----|-----|-----|-----|-----|-----|-----|-----|
| 9764 |      | SNC | ANG | G1W | INC | ZOM | TAP | BOR | REM |

REM  1 :  The soft-key for selecting reaming is disabled.
     0 :  The above soft-key is enabled.
BOR  1 :  The soft-key for selecting boring is disabled.
     0 :  The above soft-key is enabled.
TAP  1 :  The soft-key for selecting tapping is disabled.
     0 :  The above soft-key is enabled.
ZOM  1 :  The soft-key for scaling animation is displayed on page 1.
     0 :  The soft-key for scaling animation is displayed on page 2.
INC  1 :  Incremental values can be input as figure data in single machining, bar machining, threading, and grooving.
     0 :  The above function is disabled.
G1W  1 :  In grooving, a groove is cut in the same Z direction (from Z+ to Z-).
     0 :  In grooving, the center of a groove is first cut and the tool is moved right and left to cut the groove.
ANG  1 :  The angle of a line is input, assuming the clockwise direction as the negative direction.
     0 :  The angle of a line is input, assuming the clockwise direction as the positive direction.
SNC  1 :  In semifinishing for bar machining, tool-nose geometric compensation is applied. It may take time until the tool starts cutting after it moves to the cutting start point depending on the specified number of figures.
     0 :  In semifinishing for bar machining, tool-nose geometric compensation is not applied.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9765 | | | | | | BSH | CBR | |

CBR　　1 :　　C-axis boring is enabled.
　　　　0 :　　The above function is disabled.
BSH　　1 :　　The negative shift direction is used in C-axis boring.
　　　　0 :　　The positive shift direction is used in C-axis boring.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9766 | | | | | | BSH | CBR | |

Soft-keys (custom soft-keys) can be displayed in the tool data window to call and execute the corresponding macro program by pressing each soft-key.

TU1　　1 :　　Custom soft-key 1 is enabled in the tool data window.
　　　　0 :　　Custom soft-key 1 is not enabled in the tool data window.
TU2　　1 :　　Custom soft-key 2 is enabled in the tool data window.
　　　　0 :　　Custom soft-key 2 is not enabled in the tool data window.
TU3　　1 :　　Custom soft-key 3 is enabled in the tool data window.
　　　　0 :　　Custom soft-key 3 is not enabled in the tool data window.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9767 | | NT0 | SFG | MTA | NCR | | SGT | SFC |

SFC　　1 :　　For arcs created in bar machining, tool radius compensation is applied using both tool-tip radius and finishing amount.
　　　　0 :　　The compensation stated above uses only the tool-tip radius. The finishing amount is used as a shift amount for compensation.
SGT　　1 :　　Cutting edge compensation is enabled.
　　　　0 :　　Cutting edge compensation is disabled.
NCR　　1 :　　G41 or G42 is not used in the finishing process of bar machining.
　　　　0 :　　G41 and G42 is used.
MTA　　1 :　　Multiple threading is carried out by specifying an angle (Q).
　　　　0 :　　Multiple threading is carried out by shifting the start point of each thread.
SFG　　1 :　　Bar machining including very small steps is enabled.
　　　　0 :　　Bar machining including very small steps is disabled.
NT0　　1 :　　For an offset cancel command, T**00(** is the tool selection number) is used.
　　　　0 :　　T0 is used.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9768 | | | | | | | NBC | SBC |

SBC　　1 :　　Titles of the contour data input window are displayed in black.
　　　　0 :　　Displayed in white.
NBC　　1 :　　Titles of the each windows are displayed in black.
　　　　0 :　　Displayed in white.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9770 | | | | | | | NBC | SBC |

NM0-NM7　Distribution for the NC format program area and MANUAL GUIDE format program area.

| Length of tape storage | | 160m | 320m | 640m | 1280m |
|---|---|---|---|---|---|
| NM0-NM7=0 | Number of processes | 99 | 254 | 559 | 983 |
| | Length of tape memory for NC format program | 49m | 99m | 198m | 499m |
| NM0=1 (others=0) | Number of processes | 89 | 234 | 519 | 983 |
| | Length of tape memory for NC format program | 57m | 115m | 229m | 499m |
| NM1=1 (others=0) | Number of processes | 79 | 214 | 479 | 983 |
| | Length of tape memory for NC format program | 65m | 130m | 261m | 523m |
| NM2=1 (others=0) | Number of processes | 69 | 194 | 434 | 924 |
| | Length of tape memory for NC format program | 73m | 146m | 293m | 586m |
| NM3=1 (others=0) | Number of processes | 51 | 109 | 269 | 594 |
| | Length of tape memory for NC format program | 89m | 210m | 420m | 840m |
| NM7=1 (others=0) | Number of processes | - | - | - | 114 |
| | Length of tape memory for NC format program | | | | 1204 |

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9771 | | | | | | | | |

All of these bits must be set to 0

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9772 | | | RFN | | RLF | EDM | M5O | CM5 |

CM5　1 :　When C-axis cutting processes are continuously executed, M05 is output at the beginning of each following process.
　　　0 :　M05 is not output in the above case.
M5O　1 :　M05 is outputted when the spindle rotation changed and when the spindle gear is changed.
　　　0 :　M05 is not outputted at the above case.
EDM　1 :　M30 is output at the program end.
　　　0 :　M02 is output at the program end.
RLF　1 :　Between the tool change position and the approach point, the tool moves simultaneously along 2 axes.
　　　0 :　The tool moves along 1 axis at a time at the above case.
RFN　1 :　Semi finish machining is not carried out.
　　　0 :　Semi finish machining is carried out always.
　　　This parameter is available in bar machining and normal grooving.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| **9773** | TRT | MDL | | TCD | | | GRO | |

GRO 1 : Data for graphic drawing is not outputted at NC format program conversion.

0 : Data for graphic drawing is outputted at the above case.

TCD 1 : Sub program is called instead of T-code output.

0 : T-code is outputted.

< Related parameter : No.9778#7bit(CLT) >

MDL 1 : At NC format program conversion, a continuous state G code (01 group) and F code are not outputted if they are the same as those in the previous block.

0 : The above G code and F code are outputted always.

TRT 1 : When the tool is first move to the tool change position after machining is started, it moves along the Z-axis then along the X-axis.

0 : When the tool is first moved to the tool change position after machining is started, it moves first along the X-axis then along the Z-axis.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| **9774** | | | T0O | M1O | 28O | MLT | MLS | |

MLS 1 : The specified M code (parameter No. 9876) is output to stop the milling axis or change the direction of its rotation.

0 : When the direction of rotation of the milling axis changes, the M code for stopping the axis is not output.

MLT 1 : To rotate the milling axis in the forward or reverse direction, the specified M code (parameter No. 9877 or 9878) is output.

0 : M03 or M04 is output in the above case.

28O 1 : At the end of the program, T0 and G28 are not outputted.

0 : T0 and G28 are outputted automatically at the above case.

M1O 1 : M01 is output after the tool returns to the tool-change position at the end of each process.

When using soft-key [PROCESS] on the animation screen, always set M1O to 1.

0 : M01 is not outputted at the above case.

T0O 1 : T0 is outputted immediately before a return to the tool change position (G30).

0 : T0 is not outputted at the above case.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| **9775** | | | | | | | | |

All of these bits must be set to 0

|      |  #7  |  #6  |  #5  |  #4  |  #3  |  #2  |  #1  |  #0  |
|------|------|------|------|------|------|------|------|------|
| 9776 |      |      |      |      |      | MTS  | STM  |      |

  STM  1 :  To switch from the turning mode to the milling mode, NC
            program O9994 is called instead of outputting an M code.
       0 :  An M code is output in the above case.
            <Related parameter : 9880 (STMMCD)>
  MTS  1 :  To switch from the milling mode to the turning mode, NC
            program O9995 is called instead of outputting an M code.
       0 :  An M code is output in the above case.
            <Related parameter : 9881 (MTSMCD)>

|      |  #7  |  #6  |  #5  |  #4  |  #3  |  #2  |  #1  |  #0  |
|------|------|------|------|------|------|------|------|------|
| 9777 | ACC  |      |      |      |      |      |      |      |

  ACC  1 :  The waste cutting cancel function for a formed material is
            enabled.
       0 :  The above function is disabled.
            When this parameter (ACC) is set to 1, also set bit 2 (BSZ) of
            parameter No. 9761 to 1. Set the other bits of this parameter to 0.

|      |  #7  |  #6  |  #5  |  #4  |  #3  |  #2  |  #1  |  #0  |
|------|------|------|------|------|------|------|------|------|
| 9778 | CLT  |      |      | SMS  | SSM  |      |      |      |

  SSM  1 :  A subprogram for switching from the turning mode to the
            milling mode is created in the P code area using the macro
            compiler.
       0 :  The NC program (O9994) is used as the above subprogram.
  SMS  1 :  A subprogram for switching from the milling mode to the
            turning mode is created in the P code area using the macro
            compiler.
       0 :  The NC program (O9995) is used as the above subprogram.
  CLT  1 :  The sub program to be called by a T code is stored in the ROM
            area by the macro compiler.
       0 :  The above sub program is stored in a tape storage area.
            <Related parameter : No.9773#4(TCD)>

> **NOTE**
> When this parameter is set to 1, the tool may
> temporarily stop along the axis at transition from
> one block to the next depending on the feedrate. In
> this case, set this parameter to 0.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9779 | | G32 | | | | | CRC | CRF |

CRF 1 : When forward rotation is specified by the C-axis end face tool rotation command, the M code for reverse rotation is output. When reverse rotation is specified, the M code for forward rotation is output. For C-axis tapping, the M code specified for parameter No. 9861 (CRVMCD) is output immediately before the tapping command.

0 : The above function is disabled.

CRC 1 : When forward rotation is specified by the C-axis side face tool rotation command, the M code for reverse rotation is output. When reverse rotation is specified, the M code for forward rotation is output. For C-axis tapping, the M code specified for parameter No. 9861 (CRVMCD) is output immediately before the tapping command.

0 : The above function is disabled.

G32 1 : Tapping for turning is done by G32.

0 : Tapping for turning is done by G84.
Example)

| | |
|---|---|
| G32=0 :<br>X0. M8 ;<br>Z6. ;<br>G99 G84 Z-30. R0. F0.5 P225 ;<br>G80 ; | G32=1 :<br>X0. M8 ;<br>Z6. ;<br>G32 Z-30. R0. F0.5 M5 ;<br>Z6. M4 ;<br>G99 ; |

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9845 | SPE | RSC | ETC | WOP | UMC | NEX | | NTL |

NTL 1 : The tool data window is not displayed.

0 : The tool data window is displayed.

NEX 1 : Soft-key for executing playback [EXEC] is not displayed.

0 : The above soft-key is displayed.

UMC 1 : Each auxiliary function specified by calling a user program is enabled on the MANUAL GUIDE initial screen, guidance and single cutting screens, and cycle screen.

0 : Each auxiliary function specified by calling a user program is enabled only on the MANUAL GUIDE initial screen and guidance and single cutting screens.

WOP 1 : A user window can be displayed using an external key. Eight types of windows can be displayed. A user window can be displayed at a time; multiple user windows cannot be displayed simultaneously.

0 : No user window can be displayed using any external key.

ETC  1 :  Teach-in with an external key is enabled.
       0 :  Teach-in with any external key is disabled.
          When each teach-in button (TEACH AUX, TEACH RAPID, or TEACH CUT) on the operator's panel is pressed, set the corresponding PMC signal listed below to 1:
          R988#0<CUTT> : Teaching [CUT]
          #1<RPDT>   : Teaching [RAPID]
          #2<AUXT>   : Teaching [AUX]

RSC  1 :  The RESET key is used to locate the beginning of teach-in data displayed in the figure data window in guidance or single cutting.
       0 :  The RESET key is not used to locate the beginning of teach-in data displayed in the figure data window in guidance or single cutting.

SPE  1 :  The direction of spindle rotation displayed in the status indicator window is changed by the PMC signals.
          R987  = 1 : M03 (CW)
                = 2 : M04 (CCW)
       0 :  The direction of spindle rotation displayed in the status indicator window is changed by the specified spindle rotation value.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9846 | RTN | CME | MTM | TTM | ENM | SNM | POS | JOG |

JOG  1 :  The jog feedrate input function is enabled.
       0 :  The above function is disabled.

POS  1 :  The current position data displayed in the status indicator window can be switched between absolute, relative, and machine coordinates. The function of presetting a relative coordinate is also enabled.
       0 :  The above function is disabled.

SNM  1 :  The M code for switching from the milling mode to the turning mode is not output at the start of single cutting.
       0 :  The above M code is output.

ENM  1 :  At the end of each process in cycle cutting, the M code specified in process data is output.

> **NOTE**
> When this parameter is set to 1, the SPDL GEAR input item in process data cannot be used.

       0 :  The above M code is not output.

TTM  In tapping by turning, an M code (for indicating forward/reverse rotation in tapping) is:
       1 :  Issued before tapping (the M code specified for parameter No. 9292 or 9293 is issued).
       0 :  Not issued before tapping (the rigid tapping stop M code is issued).

> **NOTE**
> When bit 6 (G32) of parameter No. 9779 is set to 1, the setting of this parameter is invalid in tapping by turning.

- 267 -

MTM    In tapping by milling, an M code (for indicating forward/reverse rotation in tapping) is:
  0 :  Not issued before tapping (the milling rigid tapping stop M code is issued).
  1 :  Issued before tapping (the M code specified for parameter No. 9088 or 9089 is issued). In this case, always set bit 0 (CRF) and bit 1 (CRC) of parameter No. 9779 to 0.

CME    1 :  The valid range of M codes to be specified for parameters Nos. 9876 to 9878 is between 0 and 255.
  0 :  The valid range of the above M codes is between 0 to 99.
       <Related parameters P9876 to 9878>

RTN    1 :  The tool returns to the tool-change position unconditionally regardless of whether tool change is required at transition from one process to the next.
  0 :  The above function is disabled.

> **NOTE**
> When bit 5 (G30) of parameter No. 9760 is set to 0, this function is disabled even when this parameter is set to 1.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| 9847 | SPR | EFS | SGL | ATS | SGR | NCD | | WRN |

WRN    1 :  The warning message is output if residual part is remained in bar machining. (Enable is animated drawing.)
  0 :  The above function is disabled.

NCD    1 :  NC statements are displayed on the machining simulation screen.
  0 :  The above function is disabled.
       <Related parameter P9762<NCA>>

SGR    1 :  The animation screen is displayed when soft-key [SELPR] is pressed.
  0 :  The tool path screen is displayed when soft-key [SELPR] is pressed.
       <Related parameter P9847<SPR>>

ATS    1 :  Automatic scaling is not performed in machining simulation.
  0 :  The above function is disabled.

> **NOTE**
> When the MANUAL GUIDE selection DI signal is turned off, then on again and the machining simulation screen is displayed again with this parameter set to 1, the value set by the NC is used as the graphic scale.

EFS    1 :  A contour input figure can be scaled up.
  0 :  The above function is disabled.

SPR    To prevent operation with a program erroneously selected by an operator error, only the program selected using soft-key [SELPR] can be used for machining operation.
0 :   The above function is disabled.
1 :   Only the program selected by pressing soft-key [SELPR] can be used for machining operation.
       <Related parameters P9847 bit 3 (SGR)>

SGL    1 :   (1)   In case of deleting a taught block in guidance or trace cutting, the confirmation message for deleting one block data is not displayed.
             (2)   When feed/rev or feed/min is commanded with PMC signals R950.2 or R950.3, F-command is not output.
             (3)   In guidance or single cutting, input entry SSM and RPM are displayed as the spindle speed entry.
       0 :   (1)   In case of deleting a taught block in guidance or trace cutting, the confirmation message for deleting one block data is displayed.
             (2)   When feed/rev or feed/min is commanded with PMC signals R950.2 or R950.3, F-command is output.
             (3)   In guidance or single cutting, spindle speed SSM/PRM is selected by pushing the soft key [/MIN] or [/RPM].

0 :   The above function is disabled.
1 :   Only the program selected by pressing soft-key [SELPR] can be used for machining operation.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| **9848** | **TRP** | | | | **NDV** | | | |

WRN    1 :   Thread repair function is available.
       0 :   Thread repair function is not available.
NDV    1 :   Avoiding P/S3015 alarm or wrong tool path in the arc command in bar machining.
       0 :   The above function is disabled.

| **9749** | **MCPEND** |
|---|---|

MCPEND    M-code at end of process

- 269 -

## *A.5* NUMERICAL PARAMETERS FOR MANUAL GUIDE COMMON USE

| 9780 | OTSFPX |
|---|---|

OTSFPX Distance (diameter) from the cutting start point to the approaching point along the X-axis.
Valid data range : 0 to 99,999,999
Units : 0.001mm
0.0001inch

| 9781 | OTSFPZ |
|---|---|

OTSFPZ Distance from the work-piece end face (Z=0) to the approaching point along the Z-axis.
Valid data range : 0 to 99,999,999
Units : 0.001mm
0.0001inch

| 9782 | INCLRX |
|---|---|

INCLRX Distance (diameter) from the cutting start point to the approaching point for inner cutting along the X-axis.
Valid data range : 0 to 99,999,999
Units : 0.001mm
0.0001inch

| 9783 | INCLRZ |
|---|---|

INCLRZ Distance from the work-piece end face (Z=0) to the approaching point for inner cutting along the Z-axis.
Valid data range : 0 to 99,999,999
Units : 0.001mm
0.0001inch

> **NOTE**
>  The approaching point is calculated as follows.
> 1  Bar machining, Threading, Grooving
>     1)  [OUTER] / [OUTER REVERSE]
>         X = [Cutting start point X] + [OTSFPX]
>         Z = [Z=0] + [OTSFPZ]
>     2)  [INNER] / [INNER REVERSE]
>         X = [Cutting start point X] - [INSFPX]
>         Z = [Z=0] + [INSFPZ]
>     3)  [FACE] / [FACE REVERSE]
>         X = [Cutting start point X] + [OTSFPX]
>         Z = [Z=0] + [OTSFPZ]
> 2  Drilling
>     X = [HLSFPX : Param.No.9787]
>     Z = [HLSFPZ : Param.No.9788]

| 9784 | SFCLRX |
|------|--------|

SFCLRXC    learance to the cutting start point along the X-axis (diameter).
           Valid data range    :    0 to 99,999,999
           Units :    0.001mm
                      0.0001inch

| 9785 | SFCLRX |
|------|--------|

SFCLRXC    learance to the cutting start point along the Z-axis.

           Valid data range    :    0 to 99,999,999
           Units :    0.001mm
                      0.0001inch

**NOTE**
The cutting start point is calculated as follows.
1.  Bar machining
    1) [OUTER]
       X = [Maximum outside coordinate of a figure] + [SFLCLRX]
       Z = [Z coordinate of figure start point] + [SFCLRZ]
    2) [INNER]
       X = [Minimum inside coordinate of a figure] - [SFLCLRX]
       Z = [Z coordinate of figure start point] + [SFCLRZ]
    3) [FACE]
       X = [X coordinate of figure start point] + [SFCLRX]
       Z = [Maximum Z coordinate of a figure] + [SFLCLRZ]
    4) [OUTER REVERS]
       X = [Maximum outside coordinate of a figure] + [SFLCLRX]
       Z = [Z coordinate of figure start point] - [SFCLRZ]
    5) [INNER REVERS
       X = [Minimum inside coordinate of a figure] - [SFLCLRX]
       Z = [Z coordinate of figure start point] - [SFCLRZ]
    6) [FACE REVERS]
       X = [X coordinate of figure start point] - [SFCLRX]
       Z = [Minimum Z coordinate of a figure] - [SFLCLRZ]
2.  Threading, Grooving
    1) [OUTER]
       X = [Start point X-coordinate of a figure] + [SFLCLRX]
       Z = [Start point Z-coordinate of a figure] + [SFCLRZ]
    2) [INNER]
       X = [Start point X-coordinate of a figure] - [SFLCLRX]
       Z = [Start point Z-coordinate of a figure] + [SFCLRZ]
    3) [FACE]
       X = [Start point X-coordinate of a figure] + [SFCLRX]
       Z = [Start point Z-coordinate of a figure] + [SFLCLRZ]
3.  Drilling(Center drilling, Drilling, Reaming, Tapping)
    X = 0
    Z = [Z coordinate of figure start point] + [SFCLRZ]
4.  Drilling(Boring)
    X = [Hole Diameter]
    Z = [Z coordinate of figure start point] + [SFCLRZ]
5.  C-axis drilling
    X = [Minimum inside coordinate of a figure] - [SFLCLRX]
    Z = [Z coordinate of the end face of a blank] - [SFLCLRX]
6.  C-axis grooving
    1) [FACE]
       X = [Start point X-coordinate of a figure]
       Z = [Start point Z-coordinate of a figure] + [SFCLRZ]
    2) [SIDE]
       X = [Start point X-coordinate of a figure] - [SFLCLRX]
       Z = [Start point Z-coordinate of a figure]

| 9786 | ZEROPT |
|------|--------|

This data must be set to 0 always.

| 9787 | HLSFPX |
|------|--------|

HLSFPX  X-coordinate (diameter) of the approaching point for drilling.
Valid data range   :   0 to 99,999,999
Units :   0.001mm
              0.0001inch

| 9788 | HLSFPZ |
|------|--------|

HLSFPZ  Z-coordinate of the approaching point for drilling.
Valid data range   :   0 to 99,999,999
Units :   0.001mm
              0.0001inch

| 9791 | CAXIS1 |
|------|--------|

CAXIS1  0:   The C-axis cutting menu is not displayed.
            2:   C-axis cutting A is enabled (the C-axis cutting A option is
                  required).

# *A.6*    PARAMETER FOR BAR MACHINING

| 9795 | CUTCHG |
|---|---|

CUTCHG    Rate of change in the depth of cut in bar machining. If the parameter
is set to 0, the depth of cut remains unchanged.
Valid data range : 0 to 200
Units : 1%
Example)
When the depth of cut is 5mm, CUTCHG is 70%, and CUTMIN
is 1.5mm, the depth of cuts are as follows.
First cut   5mm
2nd         3.5mm
3rd         2.45mm
4th         1.715mm
5th         1.5mm
6th         1.5mm

| 9796 | CUTMIN |
|---|---|

CUTMIN    Minimum limit of depth of cut (radius) in bar machining.
Valid data range   :   0 to 99,999,999
Units :   0.001mm
          0.0001inch

| 9797 | RELFX |
|---|---|

RELFX    Default value of X component (diameter) of a clearance from the
cutting surface in in-feed machining of Outer and Inner bar
machining.
Valid data range   :   0 to 99,999,999
Units :   0.001mm
          0.0001inch

| 9798 | RELFZ |
|---|---|

RELFZ    Default value of Z component (radius) of a clearance from the cutting
surface in in-feed machining of Face bar machining.
Valid data range   :   0 to 99,999,999
Units :   0.001mm
          0.0001inch

| 9801 | TLBACK |
|------|--------|

TLBACK    Angle at which the back of the tool is raised above the work-piece in cutting onto an intermediate part.
Valid data range : 0 to 180
Units : 1 degree
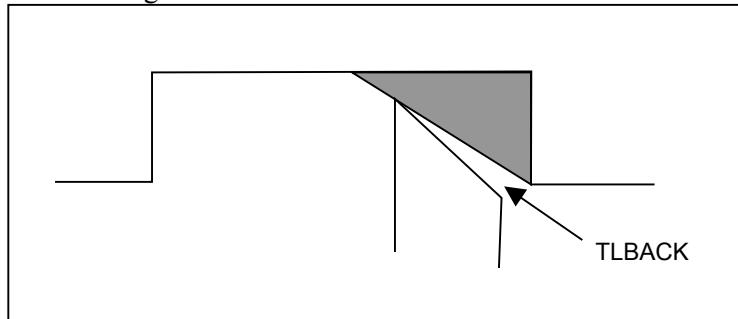


| 9802 | PCOVR1 |
|------|--------|

PCOVR1    Feed amount override when the tool cut angle is larger than 90 degrees and smaller than or equal to 135 degrees
Valid data range   :   0 to 20
Units :   10%

| 9803 | PCOVR2 |
|------|--------|

PCOVR2    Feed amount override when the tool cut angle is larger than 135 degrees and smaller than to 180 degrees
Valid data range   :   0 to 20
Units :   10%

| 9804 | PCOVR3 |
|------|--------|

PCOVR3    Feed amount override when the tool cut angle is larger than 180 degrees and smaller than or equal to 225 degrees
Valid data range   :   0 to 20
Units :   10%

| 9805 | PCOVR4 |
|------|--------|

PCOVR4    Feed amount override when the tool cut angle is larger than 225 degrees and smaller than to 280 degrees
Valid data range   :   0 to 20
Units :   10%

**NOTE**
    When the tool cut angle is 90, 180, or 270 degrees, override is disabled. When a
    value of 0 is set, override is also disabled.



Outer diameter
(forward direction)

Outer diameter
(reverse direction)

Inner diameter
(forward direction)

Inner diameter
(reverse direction)

End face
(forward direction)

End face
(reverse direction)
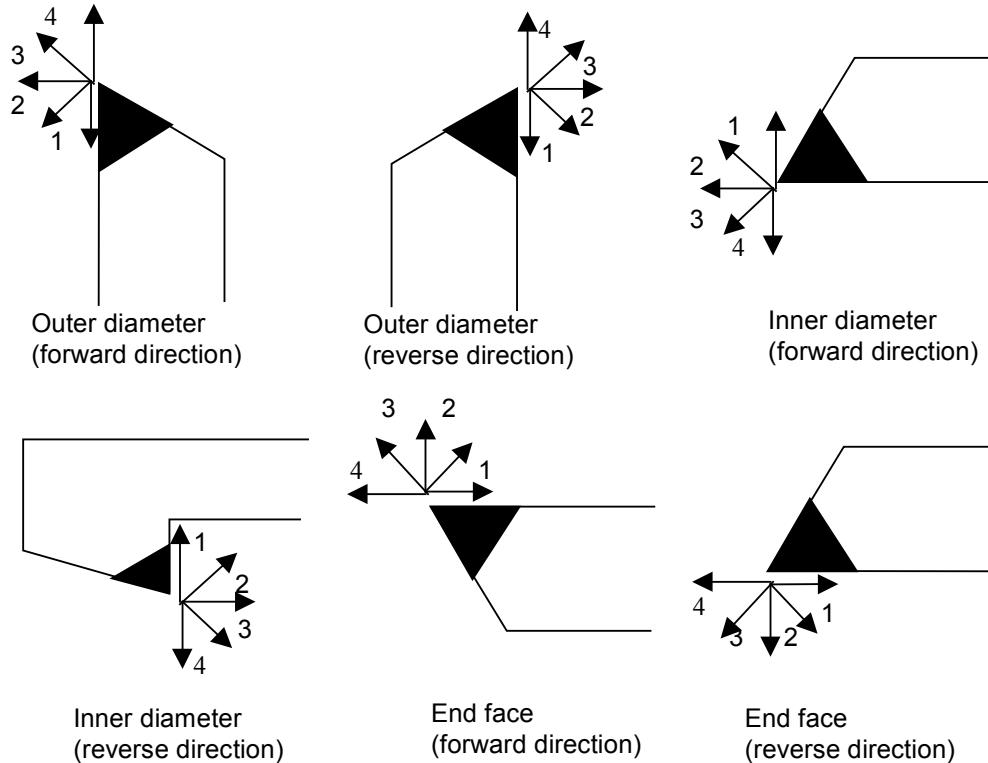
| 9805 | FSTOVR |
|------|--------|

This data must be set to 0 always.

| 9809 | CUTANG |
|------|--------|

CUTANG    Default value of cutting edge angle for bar machining.
          Valid data range : 0 to 1800
          Units : 0.1 degree

| 9810 | TIPANG |
|------|--------|

TIPANG    Default value of tool tip angle for bar machining.
          Valid data range : 0 to 1800
          Units : 0.1 degree

| 9811 | CLRLNGX |
|------|---------|

CLRLNGX    When the defined contour has a concave along the Z-axis as shown in the following example, the tool may move toward point Pc through an uncut section at rapid traverse. Distance DI in the example can be set for this parameter to move the tool from point Pa to point Pc at cutting feed to prevent the above situation.

Example) Reverse end face bar machining (roughing)
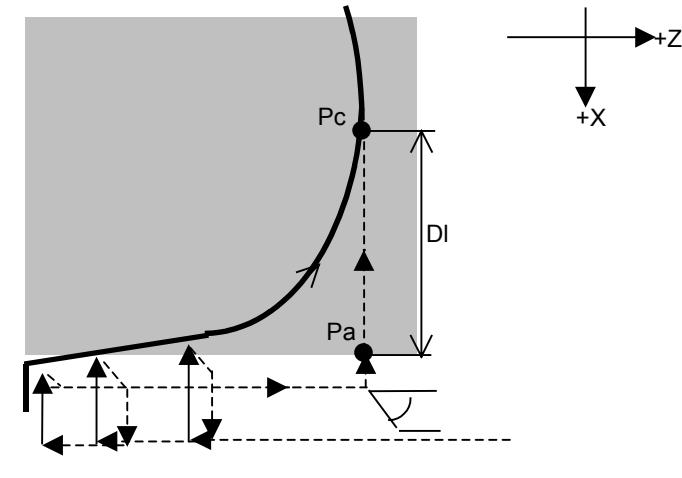
| 9812 | CLRLNGZ |
|------|---------|

CLRLNGZ    The above parameter sets distance DI along the X-axis. This parameter sets distance DI along the Z-axis as shown in the following example.

Example) Outer bar machining (roughing)
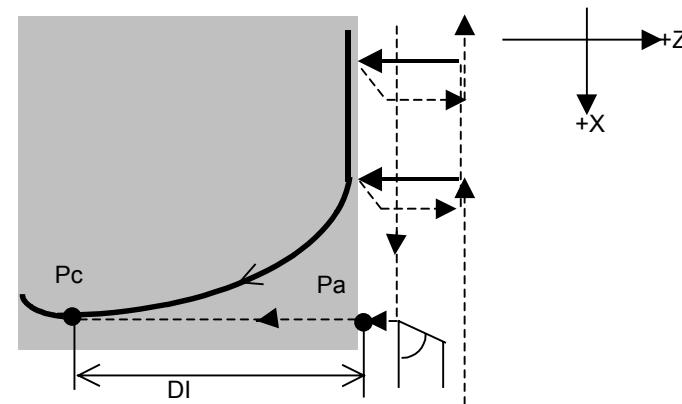
# *A.7*    **PARAMETER FOR GROOVING**

| 9820 | CLGRVX |
|------|--------|

CLGRVX    Clearance (diameter) of the X-axis in outer or inner grooving.
Valid data range   :    0 to 99,999,999
Units :   0.001mm
0.0001inch

| 9821 | CLGRVZ |
|------|--------|

CLGRVZ    Clearance (radius) of the Z-axis in end face grooving.
Valid data range   :    0 to 99,999,999
Units :   0.001mm
0.0001inch
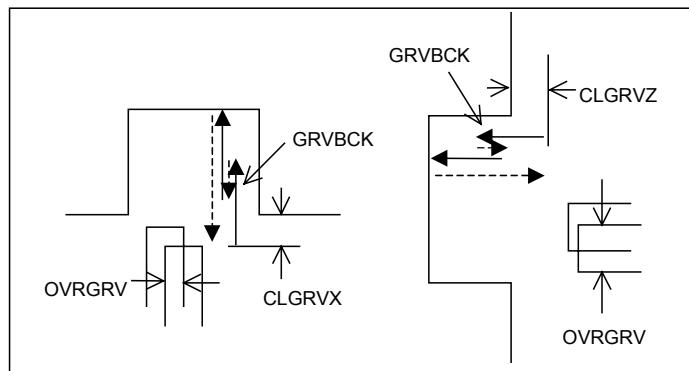


| 9822 | GRVCHG |
|------|--------|

GRVCHG    Rate of change in the depth of cut in grooving. If the parameter is set
to 0, the depth of cut remains unchanged.
Valid data range : 0 to 200
Units : 1%
Example)
When the depth of cut is 5mm, CUTCHG is 70%, and CUTMIN
is 1.5mm, the depth of cuts are as follows.
First cut   5mm
2nd         3.5mm
3rd         2.45mm
4th         1.715mm
5th         1.5mm
6th         1.5mm

| 9823 | GRVMIN |
|------|--------|

GRVMIN    Minimum limit value (radius) for the depth of cut in grooving.
Valid data range   :    0 to 99,999,999
Units :   0.001mm
0.0001inch

| 9824 | GRVBCK |
|------|--------|

GRVBCK    Pecking clearance for grooving (radius).
Valid data range  :    0 to 99,999,999
Units :   0.001mm
          0.0001inch

| 9825 | OVLGRV |
|------|--------|

OVLGRV    Overlaps between each cutting in grooving (proportion to the tool width).
Valid data range : 0 to 100
Units : 1%

# *A.8*     **PARAMETER FOR THREADING**

| 9830 | CLSCRX |
|------|--------|

CLSCRX    Default value of the clearance (diameter) of the X-axis for threading.
Valid data range  :   0 to 99,999,999
Units :  0.001mm
         0.0001inch

| 9831 | CLSCRZ |
|------|--------|

CLSCRZ    Default value of the acceleration distance (radius) of the Z axis for threading.
Valid data range  :   0 to 99,999,999
Units :  0.001mm
         0.0001inch
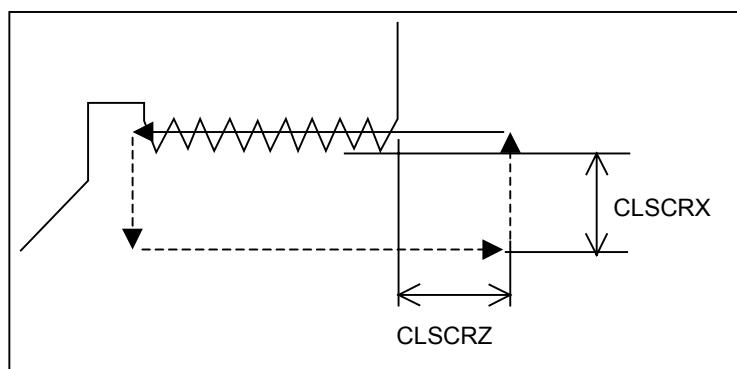


| 9832 | SCWCF |
|------|-------|

SCWCF    Coefficient for the height of a general thread.
Valid data range : 0 to 32,767
Units : 1/10000
The system automatically sets the height of the thread for process data from the following formula.
Height of a thread = thread pitch x SCWCF / 10000
Standard set value 6495 for thread angle 60degrees
                             6403 for thread angle 55degrees

| 9833 | TRDMIN |
|------|--------|

TRDMIN    Minimum depth (radius) of cut for threading.
Valid data range  :   0 to 99,999,999
Units :  0.001mm
         0.0001inch

| 9834 | TRDFNX |
|---|---|

TRDFNX   Default value of the finishing allowance along the X-axis for
         threading (diameter)
         Valid data range   :   0 to 99,999,999
         Units :   0.001mm
                   0.0001inch

| 9835 | TRDAGL |
|---|---|

TRDAGL   Default value of the threading tool angle to be set automatically. If 0
         is set, 60degrees is automatically set.
         Valid data range : 0 to 255
         Units : 1degrees

| 9836 | CMFONM |
|---|---|

CMFONM   M code for enabling chamfering in threading.
         Valid data range : 0 to 999

| 9837 | CMFOFM |
|---|---|

CMFOFM   M code for disabling chamfering in threading.
         Valid data range : 0 to 999

| 9838 | MTUNCF |
|---|---|

MTUNCF   Coefficient for the height of a metric or unified thread.
         Valid data range : 0 to 32,767          Units : 1/10000
         The system automatically sets the height of the thread for process data
         from the following formula.
         Height of a thread = thread pitch x MTUNCF / 10000
         Standard set value  6495 for thread angle 60degrees
                             6403 for thread angle 55degrees

| 9839 | PTPFCF |
|---|---|

PTPFCF   Coefficient for the height of a PT or PF thread.
         Valid data range : 0 to 32,767
         Units : 1/10000
         The system automatically sets the height of the thread for process data
         from the following formula.
         Height of a thread = thread pitch x PTPFCF / 10000
         Standard set value  6495 for thread angle 60degrees
                             6403 for thread angle 55degrees

| 9843 | ESCRNG |
|------|--------|

ESCRNG   When the tool is within the distance from the thread phase synchronization position that is specified by this parameter and soft-key [CAN OR] or [EXEC] is pressed, a warning message is displayed.
Valid data range : 0 to 99999999
Units :   0.001mm
          0.0001inch
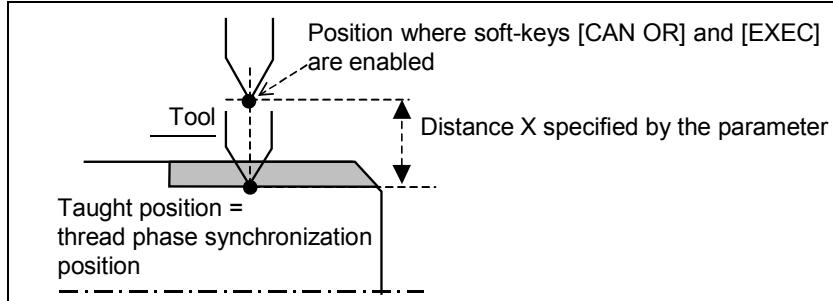


| 9844 | SPLANG |
|------|--------|

SPLANG   Qz is measured and set for this parameter. If P0 is equal to P1, set this parameter to 0.



| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|------|-----|-----|-----|-----|-----|-----|-----|-----|
| 9845 | SPE | RSC | ETC | WOP | UMC | NEX | | NTL |

NTL   1 :   The tool data window is not displayed.
      0 :   The tool data window is displayed.
NEX   1 :   Soft-key for executing playback [EXEC] is not displayed.
      0 :   The above soft-key is displayed.
UMC   1 :   Each auxiliary function specified by calling a user program is enabled on the MANUAL GUIDE initial screen, guidance and single cutting screens, and cycle screen.
      0 :   Each auxiliary function specified by calling a user program is enabled only on the MANUAL GUIDE initial screen and guidance and single cutting screens.
WOP   1 :   A user window can be displayed using an external key. Eight types of windows can be displayed. A user window can be displayed at a time; multiple user windows cannot be displayed simultaneously.
      0 :   No user window can be displayed using any external key.

- 282 -

ETC   1 :   Teach-in with any external key is enabled.

       0 :   Teach-in with any external key is disabled.
When each teach-in button (TEACH AUX, TEACH RAPID, or TEACH CUT) on the operator's panel is pressed, set the corresponding PMC signal listed below to 1:
R988#0<CUTT> : Teaching [CUT]
#1<RPDT>   : Teaching [RAPID]
#2<AUXT>   : Teaching [AUX]

RSC   1 :   The RESET key is used to locate the beginning of teach-in data displayed in the figure data window in guidance or single cutting.

       0 :   The RESET key is not used to locate the beginning of teach-in data displayed in the figure data window in guidance or single cutting.

SPE   1 :   The direction of spindle rotation displayed in the status indicator window is changed by the PMC signals.
R987 = 1 : M03 (CW)
= 2 : M04 (CCW)

       0 :   The direction of spindle rotation displayed in the status indicator window is changed by the specified spindle rotation value.

# A.9 PARAMETERS FOR DRILLING

| 9088 | MILTMN |
|------|--------|

MILTMN M code specified before forward tapping by milling
Valid data range : 0 to 999
When this parameter is set to 0, nothing is output.

| 9089 | MILTMR |
|------|--------|

MILTMR M code specified before reverse tapping by milling
Valid data range : 0 to 999
When this parameter is set to 0, nothing is output.

| 9292 | S1TTMN |
|------|--------|

S1TTMN M code specified before forward tapping by turning
Valid data range : 0 to 999
When this parameter is set to 0, nothing is output.

| 9293 | S1TTMR |
|------|--------|

S1TTMR M code specified before reverse tapping by turning
Valid data range : 0 to 999
When this parameter is set to 0, nothing is output.

| 9294 | S1STPM |
|------|--------|

S1STPM M code specified to spindle stop
Valid data range : 0 to 999
When this parameter is set to 0, M5 is output.

| 9295 | S1NMLM |
|------|--------|

S1NMLM M code specified forward spindle rotation
Valid data range : 0 to 999
When this parameter is set to 0, M3 is output.

| 9296 | S1RVSM |
|------|--------|

S1RVSM M code specified reverse spindle rotation
Valid data range : 0 to 999
When this parameter is set to 0, M4 is output.

| 9850 | DRLDEC |
|------|--------|

DRLDEC  Default value of the reduced depth of cut in peck or high-speed peck drilling (radius) to be set automatically.
Valid data range  :  0 to 99,999,999
Units :  0.001mm
          0.0001inch

| 9851 | DRLRET |
|------|--------|

DRLRET  Default value of the return clearance for peck or high-speed peck drilling (radius) to be set automatically.
Valid data range  :  0 to 99,999,999
Units :  0.001mm
          0.0001inch

| 9852 | DRLMIN |
|------|--------|

DRLMIN  Default value of the minimum depth of cut for peck or high-speed peck drilling (radius) to be set automatically.
Valid data range  :  0 to 99,999,999
Units :  0.001mm
          0.0001inch

| 9853 | DRILCF |
|------|--------|

DRILCF  Coefficient for the depth of cut in peck or high-speed peck drilling (radius) to be set automatically.
Valid data range : 0 to 3000
Units : 1%
The system automatically sets the cutting depth from the following formula.
Drilling depth of cut = drilling diameter x DRILCF /100

| 9855 | GRDLCL |
|------|--------|

GRDLCL  Clearance (radius) from each start point in C-axis drilling and grooving
Valid data range  :  0 to 99,999,999
Units :  0.001mm
          0.0001inch



- 285 -
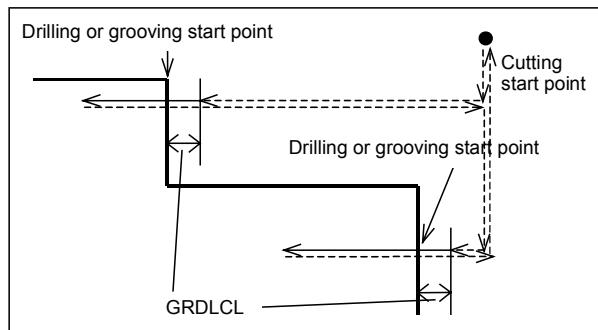
| 9856 | TAPCCL |
|------|--------|

TAPCCL Clearance (radius) from each start point in C-axis tapping
Valid data range : 0 to 99,999,999
Units : 0.001mm
0.0001inch

| 9858 | BRSHFT |
|------|--------|

BRSHFT Default value of the amount of shift for returning in boring (radius)
Valid data range : 0 to 99,999,999
Units : 0.001mm
0.0001inch
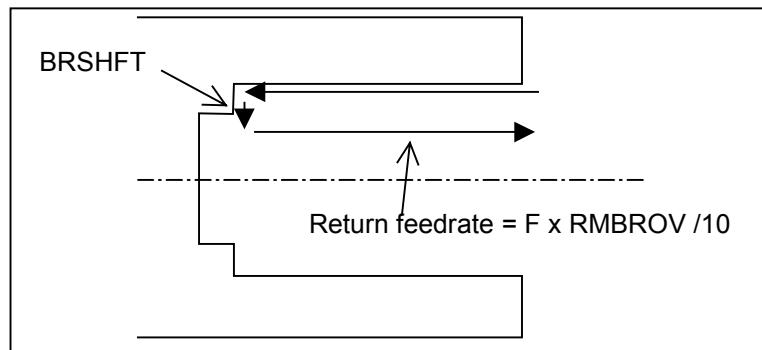


| 9858 | RMBROV |
|------|--------|

RMBROV Feedrate override for returning in reaming or boring
Return F = F x RMBROV / 10
The tool always returns at return F in reaming.
In boring by turning, the tool normally returns at the rapid traverse
rate. If the hole diameter is small and the clearance is insufficient (the
amount of shift for returning < BRCLER), the tool returns at return F.
Valid data range : 0 to 20
Units : 10%

| 9861 | CRVMCD |
|------|--------|

CRVMCD M code for rotating the milling axis in the reverse direction in C-axis
tapping
<Related parameters : No.9779#0bit(CRF),No.9779#1bit(CRC)>
Valid data range : 0 to 999

# *A.10*  PARAMETERS FOR OTHER MANUAL GUIDE FUNCTIONS

| 9870 | GERMC1 |
|------|--------|

GERMC1   M-code outputted when a low-speed gear is selected.
Valid data range : 0 to 255

| 9871 | GERMC2 |
|------|--------|

GERMC2   M-code outputted when an intermediate speed gear 2 is selected.
Valid data range : 0 to 255

| 9872 | GERMC3 |
|------|--------|

GERMC3   M-code outputted when an intermediate speed gear 2 is selected.
Valid data range : 0 to 255

| 9873 | GERMC4 |
|------|--------|

GERMC4   M-code outputted when a high-speed gear is selected.
Valid data range : 0 to 255

| 9876 | MLSTPM |
|------|--------|

MLSTPM   M code specified milling tool axis stop
<Related parameters : No.9774#1bit(MLS), No.9846#6bit(CME)>
Valid data range : 0 to 99
When this parameter is set to 0, M5 is output.

| 9877 | MLNMLM |
|------|--------|

MLNMLM   M code specified forward milling axis rotation
<Related parameters : No.9774#2bit(MLT) , No.9846#6bit(CME)>
Valid data range : 0 to 99
When this parameter is set to 0 or a value outside the valid data range
is set, this M code is not output.

| 9878 | MLRVSM |
|------|--------|

MLRVSM   M code specified reverse milling axis rotation
<Related parameters : No.9774#2bit(MLT) , No.9846#6bit(CME)>
Valid data range : 0 to 99
When this parameter is set to 0 or a value outside the valid data range
is set, this M code is not output.

| 9880 | STMMCD |
|------|--------|

STMMCD   M-code for switching the turning mode to the milling mode
Valid data range : 0 to 99

| 9881 | MTSMCD |
|---|---|

MTSMCD   M-code for switching the milling mode to the turning mode
Valid data range : 0 to 255

| 9883 | SQNOIC |
|---|---|

SQNOIC   Sequence number (N) to be outputted for each block during NC format program converting.

0             : No sequence number is outputted.
1 to 8999   : This value is added to the sequence number for the first block of each process.
-8999 to –1 : The absolute value of this data is added to the sequence number for all blocks.

> **NOTE**
> If nonzero data is set, process names (comment statement in English only) are outputted at the beginning of each process of the converted NC format program.

| 9884 | GRPMGN |
|---|---|

GRPMGN   Scale factor of the blank length and blank diameter in the graphic parameters for the length and the maximum outside diameter of a work material used in a machining.
Valid setting range : 0 to 20
Units : 10 percent
Blank length and blank diameter in the graphic parameters = maximum outside diameter and length of a work material used in a machining * (GRPMGN)/10

> **NOTE**
> When this parameter is set to 0, automatic setting is not performed.

| 9889 | MFNCTM |
|---|---|

MFNCTM   Average time required to execute a M code function.
Valid data range : 0 to 99,999,999
Units : 10msec

> **NOTE**
> Parameter from No.9889 to 9891 are used to calculate the machining time in machining simulation.

| 9890 | MFNCTM |
|------|--------|

MFNCTM     Average time required to execute a S code function.
           Valid data range : 0 to 99,999,999
           Units : 10msec

| 9891 | MFNCTM |
|------|--------|

MFNCTM     Average time required to execute a T code function.
           Valid data range : 0 to 99,999,999
           Units : 10msec

# *A.11*  SETTING PARAMETERS IN THE NC

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| **3405** | | | | CCR | | | | |

CCR   To specify a corner radius during chamfering.
  1 :  Address C and R are used. Address C cannot be used for C-axis.
  0 :  Address I and K are used. In direct programming of a drawing dimension, ",C" and ",R" are used.
  This bit must be set to 0 always.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| **5002** | | | LGC | LGT | | LWT | LGN | LD1 |

LD1   1 :  The offset number of tool offset is specified by one digit after the T code.
  0 :  The offset number of tool offset is specified by two digits after the T code.
  This bit must be set to 0 always.

LGN   1 :  The upper two digits in the T code indicate a tool geometry compensation number.
  0 :  The lower two digits in the T code indicate a tool geometry compensation number.
  This bit must be set to 0 always.

LWT   1 :  Tool wear compensation is applied by shifting the coordinate system.
  0 :  Tool wear compensation is applied by moving the tool.
  This bit must be set to 1 always.

LGT   1 :  The tool geometry is compensated according to tool movement.
  0 :  The tool geometry is compensated according to the shift of the work coordinate system.
  This bit must be set to 0 always.

LGC   1 :  Tool geometry compensation is canceled by specifying 0 for the offset number.
  0 :  Tool geometry compensation is not canceled by specifying 0 for the offset number.
  This bit must be set to 1 always.

| **5110** | **C-axis clamp M code** |
|---|---|

The value obtained by adding 1 to the M code set for this parameter is used as the C-axis unclamp M code.
Valid data range : 0 to 99

| **5111** | **Dwell time when C-axis unclamping is specified** |
|---|---|

Valid data range : 0 to 32767
Units : 1msec

| 5210 | M code for specifying the rigid tapping mode |
|------|----------------------------------------------|

Set an M code for specifying the rigid tapping mode.
Valid data range : 0 to 255

| 5212 | M code for specifying the rigid tapping mode |
|------|----------------------------------------------|

Set an M code for specifying the rigid tapping mode.
The M code for specifying the rigid tapping mode is normally set for parameter No. 5210. If you want to use an M code having a number 256 or more, set the M code for this parameter.
Valid data range : 0 to 65535

> **NOTE**
> As the M code for specifying the rigid tapping mode, the M code set for parameter No. 5212 is issued when the parameter is set to a value other than 0 or the M code set for parameter No. 5210 is issued when parameter No. 5212 is set to 0. If both parameters Nos. 5210 and 5212 are set to 0, M29 is issued, however.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|------|----|----|----|----|----|----|----|----|
| 6500 |    | NZM |   | CSF | DPA |   |   |   |

DPA  1 : The current position shown on the graphic screen is the programmed position.
     0 : The actual position obtained after the too radius compensation is applied.
     This bit must be set to 0 always.

CSF  Product figure drawing in cycle cutting is:
     1 : Enabled.
     0 : Disabled.

NZM  1 : The new style of scale magnification is enabled.
     0 : The conventional scale magnification is enabled.
     This bit must be set to 1 always.

> **NOTE**
> Set the other bits of this parameter to 0.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|------|----|----|----|----|----|----|----|----|
| 6501 |    |    |    | CSR |   |   |   |   |

CSR  1 : A cross cursor (x) is appeared on the scale magnifying screen.
     0 : A square cursor is appeared.
     This bit must be set to 0 always.

| | #7 | #6 | #5 | #4 | #3 | #2 | #1 | #0 |
|---|---|---|---|---|---|---|---|---|
| **6502** | | | | | | | | HSA |

HSA    1 :    High speed machining simulation is enabled.
         0 :    Conventional type machining simulation is enabled.
         This bit must be set to 1 always.

---

**NOTE**
   Set the other bits of this parameter to 0.

---

| **5212** | GRPAX |
|---|---|

GRPAX    Drawing coordinate system in the machining simulation.
         This data must be set to 15 always.
         Set 14 for the back tool post lathe.

# B ALARMS

If one or more of the set of the parameters or inputted programs are not correct when an attempt is made to execute that program, the following P/S alarms are raised.

When an alarm other than the following P/S alarms is raised, refer to the relevant NC operator's manual.

| Alarm | | Description |
|---|---|---|
| 3000 | Cause | Some initial setting data commonly used for a machining program is missing. For example, this alarm occurs by the following mistake. <br> - No cutting motion in a work-piece was taught-in in a guidance or single cutting process. <br> - Incorrect figure data is inputted in cycle motions |
| | Action | Delete that program at once, and input it again with regular figure data or cutting motions. |
| | Reference | None |
| 3001 | Cause | The value of process data such as the machining area, tool number, or spindle speed is incorrect. <br> Example)  A minus value is entered for an item that must be positive. |
| | Action | On the process data window corresponding to this alarm, check the entered process data values. If there is an incorrect value, enter a correct value. |
| | Reference | II.4 Guidance Cutting Operations, II.5 Single Cutting Operations   III Types of Cycle Motions |
| 3002 | Cause | Some figure data is missing. |
| | Action | On the figure data window for the process corresponding to this alarm, check the entered data. If some data is missing, enter correct data. |
| | Reference | III Types of Cycle Motions |
| 3005 | Cause | On the process data window, the value for the feedrate is missing or set to 0. |
| | Action | On the process data window corresponding to this alarm, enter a correct value for the feedrate. |
| | Reference | II.4 Guidance Cutting Operations, II.5 Single Cutting Operations   III Types of Cycle Motions |
| 3006 | Cause | On the process data window, the value for the depth of cut is missing or set to 0. |
| | Action | On the process data window corresponding to this alarm, enter a correct value for the depth of cut. |
| | Reference | III Types of Cycle Motions |

| Alarm | | Description |
|---|---|---|
| 3007 | Cause | On the process data window, the value for the spindle speed is missing or set to 0. |
| | Action | On the process data window corresponding to this alarm, enter a correct value for the feedrate. |
| | Reference | II.4 Guidance Cutting Operations, II.5 Single Cutting Operations<br>III Types of Cycle Motions |
| 3011 | Cause | The MANUAL GUIDE function does not operate normally. This alarm is raised when the NC internal memory is damaged and the MANUAL GUIDE function cannot operate normally. |
| | Action | Delete the program corresponding to this alarm. Even though this alarm cannot be canceled, delete the all program by turning on a power with pushing a DELETE key, and input the program again. |
| | Reference | None |
| 3014 | Cause | The program is not selected. This alarm is raised when a playback machining starting is performed without any program being selected. |
| | Action | Select a program on a program list window, and perform a playback machining starting. |
| | Reference | II.8 Playback Machining and Machining Simulation |
| 3015 (Finishing)<br><br>3016 (Roughing) | Cause | The finishing allowance or radius of the tool tip radius specified for bar machining or grooving is large. Normal rough machining cannot be executed. |
| | Action | Reduce the finishing. Alternatively, use a tool with a smaller radius at the tool tip. |
| | Reference | III.1 Bar Machining (Roughing), III.2 Bar Machining (Finishing)<br>III.5 Grooving (Trapezoidal) |
| 3020 | Cause | Cutting in bar machining is impossible because of the relationship between the tool angle, cutting edge angle, and clearance angle (parameter No.9801) for the tool used.<br>This alarm is raised when the sum of the above three angles is less than or equal to 90degrees or is greater than or equal to 180degrees. |
| | Action | Check the above three data, and input correct values or select other tools. |
| | Reference | III.1 Bar Machining (Roughing), III.2 Bar Machining (Finishing) |
| 3021 | Cause | The part to be left in cutting tool back compensation cannot be determined at bar machining. |
| | Action | On the figure data window for the process corresponding to this alarm, check the machining figure. If the figure is incorrect, enter correct figure data. If the figure is correct, the tool cannot be used for this figure. So, change the tool for the process data or change the clearance angle (parameter No.9801). |
| | Reference | III.1 Bar Machining (Roughing), III.2 Bar Machining (Finishing) |

| Alarm | | Description |
|---|---|---|
| 3022 | Cause | The figure data is incorrect in bar machining. This alarm is raised when the relationship between start point coordinate and figures is incorrect. |
| | Action | On the figure data window for the process corresponding to this alarm, check the machining figure. If the figure is incorrect, enter correct figure data. |
| | Reference | III.1 Bar Machining (Roughing), III.2 Bar Machining (Finishing) |
| 3025 | Cause | A correct tool path cannot be calculated in bar machining. This alarm is raised when there is an error in the result of internal calculation (for example, when the value under a square root sign is negative because of an error in calculation). |
| | Action | On the figure data window for the process corresponding to this alarm, check the machining figure. If the figure is incorrect, enter correct figure data. If the figure is correct, the tool or finishing amount cannot be used for this figure. Reduce the finishing. Alternatively, use a tool with a smaller radius at the tool tip. |
| | Reference | III.1 Bar Machining (Roughing), III.2 Bar Machining (Finishing) |
| 3026 | Cause | The groove width is incorrect in grooving. This alarm is raised when the width of the tool used is larger than the groove width. |
| | Action | On the process data window for the grooving process corresponding to this alarm, check the groove width. If the width is incorrect, enter correct value. Otherwise, select a tool whose cutting edge width is smaller than the groove width. |
| | Reference | III.4 Grooving (Standard) |
| 3028 | Cause | Cutting in trapezoidal grooving is impossible because of the relationship of the groove bottom width and the tool width. This alarm is raised when the cutting edge width of the tool used is larger than the groove bottom width in trapezoidal grooving. |
| | Action | Check the figure, on the figure data window for the grooving process corresponding to this alarm. If the figure is incorrect, enter correct data. Otherwise, select a tool whose cutting edge width is smaller than the groove width. |
| | Reference | III.5 Grooving (Trapezoidal) |
| 3029 | Cause | The start point for threading coincides with the end point. |
| | Action | On the figure data window for the threading process corresponding to this alarm, enter correct data for the figure. |
| | Reference | III.3 Threading |
| 3031 | Cause | An invalid chamfering tool is set for C-axis grooving. This alarm is raised when the small diameter of the chamfering tool used is larger than the groove width. |
| | Action | Use a chamfering tool whose small diameter is smaller than the groove width. |
| | Reference | IV.4.3 C-axis end face grooving |

# C

# DYNAMIC GRAPHIC DISPLAY FOR NC PROGRAMS

# *C.1*      **OVERVIEW**

Motions in a playback machining program created using the MANUAL GUIDE can be checked using the machining simulation function (Chapter 8, "PLAYBACK MACHINING AND MACHINING SIMULATION," in Part II).

For an NC program consisting of G or M codes, this machining simulation function cannot be used under normal conditions.

An option function, "dynamic graphic display for NC programs," can be used to use the machining simulation function.

The machining simulation function of the MANUAL GUIDE automatically creates blank figure and tool figure data to be drawn. With dynamic graphic display for NC programs, the operator should input the data additionally. (For details, see Section C.4, "DYNAMIC GRAPHIC COMMANDS.")

---

⚠ **WARNING**

An NC program which operates normally with machining simulation of this function does not operate normally in actual machining if tool offset data to be used is invalid.

During machining simulation, the tool coordinates specified in the NC program are checked, but additional data required for actual machining such as the tool offset to be used and amount of shift of the workpiece coordinate system is not checked. For this reason, before performing actual machining with an NC program checked with machining simulation, be sure to check that required all data is set correctly, according to the manual supplied by the machine tool builder or NC function manual.

If the machine is operated using invalid data, the tool may strike the workpiece or machine, possibly breaking the tool or machine or injuring you.

---

**NOTE**

Machining simulation for NC programs cannot perform animated drawing of solid model cutting. Machining simulation can be performed only for turning and cannot be performed for C-axis cutting.

---

# C.2     OPERATION (MACHINING SIMULATION)

1) Disable the MANUAL GUIDE according to the instructions in the manual supplied by the machine tool builder. In ordinary cases, turn the MANUAL GUIDE selection switch on the machine operator's panel off.

2) Select an NC program for machining simulation according to the instructions in the NC function operator's manual. Also, select the memory operation mode.

3) Press function button GRAPH to display the machining simulation screen.

4) Press soft-key [G.PRM] to display the graphic parameter screen. On this screen, set required data. In ordinary cases, only the following data items are required. (For details of other data items, refer to the explanation of the graphic function in the NC function operator's manual.)

WORK LENGTH W:
Length of the blank to be machined
(Valid data range 0 to 99999999,
Increment system 0.001mm/0.0001inch)

WORK DIAMETER D     :
Diameter of the blank to be machined (The valid data range and increment system are the same as for WORK LENGTH.)

GRAPHIC MODE M     :
Drawing format. Be sure to set a value between 1 and 3.
1 =  Blank (cross section) and tool path
2 =  Animated drawing of cutting the blank (cross section) with the tool
3 =  Animated drawing of cutting the blank (cross section) with the tool and tool path
In ordinary cases, set 1.

---

**NOTE**
Graphic parameters WORK LENGTH and WORK DIAMETER are used only for determining the drawing scaling factor and position and not used for drawing the blank figure.
To draw the blank figure, input dynamic commands described in Section C.4 additionally.

---

5) Press soft-key [GRAPH] to display the machining simulation screen. Press soft-key [(OPRT)] to display the following soft-keys. Use these soft-keys for required operation.

SPEED UP     :
Increases the machining simulation speed. (Valid data range: 0 to 500) In ordinary cases, set 500 (maximum speed).

SPEED DOWN     :
Reduces the machining simulation speed.

HEAD    :
   Returns to the top of the program and initializes the screen.
WORK PIECE :
   Initializes blank figure drawing.
PROCESS      :
   Starts machining simulation. When M01 is found in a block in
   the NC program, a single block stop is caused in the block.
EXEC     :
   Starts continuous machining simulation.
SINGLE STEP:
   Causes a single block stop while continuous machining
   simulation is being executed or starts single block machining
   simulation when the function is stopped.

---

**NOTE**
   The soft-keys other than the above ([PROT] and
   [PROG ON/OFF]) cannot be used.

---

# *C.3*     **OPERATION (SCALING)**

1) Press the rightmost soft-key ([+]) when the above soft-keys are displayed. Soft-key [ZOOM] is displayed.

2) Press soft-key [ZOOM] to display the drawing scaling screen. Use the following soft-keys and cursor keys to perform required operation:

NORMAL :
> Returns to the standard scaling factor automatically determined according to WORK LENGTH and WORD DIAMETER.

SCAL ↑ :
> Scales the drawing up.

SCAL ↓ :
> Scales the drawing down.

CAN :
> Cancels scaling for drawing.

EXEC :
> Determines the specified scaling factor.

CURSOR KEY(← → ↑ ↓):
> Moves the graphic cursor indicating the center of the screen after scaling.

3) Press the leftmost soft-key ([<]) and soft-key [(OPRT)] to display the soft-keys for machining simulation shown in Section C.2. Press [EXEC]. Machining simulation starts with the specified scaling factor for drawing.

# *C.4*     DYNAMIC GRAPHIC COMMANDS

To draw the blank to be machined or tool in machining simulation for NC programs, input the following dedicated NC commands in the NC program.

## (1) Blank figure

G10 P90054 X(Xs) Z(Zs) ;
> Start of registration of a blank figure. Xs and Zs indicate the drawing start point of the blank.

G10 P90001 X(Xn) Z(Zn) ;
> Line for the blank figure. Xn and Zn indicate the end point of the line.
>
> :
> : (Specify the lines required for drawing the blank figure.)
> :

G10 P90050 ;
> End of registration of the blank figure

> **NOTE**
> If no blank figure data is input, the blank figure specified immediately before drawing, including that specified with MANUAL GUIDE machining simulation, is drawn.

## (2) Tool figure

For all tools used in the currently selected NC program, register tool figure data at the beginning of the NC program as shown below:

G10 P90051 X(Xt) Z(Zt) Q(Gn) ;
> Start of registration of the tool figure having graphic registration number Gn. Xt and Zt indicate the tool tip position. Gn indicates the graphic registration number of the tool (1 to 16).

G10 P90001 X(Xtn) Z(Ztn) ;
> Line for the tool tip figure for graphic registration number Gn. Xn and Zn indicate the end point of the line. (tool tip figure is closed area specified the maximum 4 points)
>
> :
> : (Specify the lines required for drawing the tool tip figure.)
> :

G10 P09000 X(Xss) Z(Zss) ;
> Start point of the tool shank figure for graphic registration number Gn(Xss/Zss)

G10 P90001 X(Xsn) Z(Zsn) ;
> Line for the tool shank figure for graphic registration number Xsn and Zsn indicate the end point of the line (up to six tool shanks).
>
> :
> : (Specify the lines required for the tool shank figure.)
> :

G10 P90050 ;

> End of registration of the tool figure
>
> Assign a graphic registration number (1 to 16) to each tool to be used and input the above registration commands.

## (3)    Tool selection

Immediately before a tool is used in the NC program, select the tool from multiple tool figures registered in advance as described above.

G10 P90060 Q(Gn) R(R) ;

> Selects the tool figure having graphic registration number Gn (1 to 16) as a tool for drawing. R can be specified for mirroring the drawing.

R=   0    : X-axis off and Z-axis off

       1    : X-axis on and Z-axis off

       2    : X-axis off and Z-axis on

       3    : X-axis on and Z-axis on

---

**NOTE**

When the drawing mode is set to 1, the tool selection command is also required. In this case, specify a dummy number between 1 and 16 for Q.

---

## (4) Screen initialization

Specify the following command to draw a blank after completion of registration of the blank figure.

> G10 P90099 ;

# D

# ADDITIONAL INFORMATION ON EACH CUTTING MOTION

The tool moves in each type of machining as follows in principle:

Tool-change point
    ↓ <1>
Approaching point (*1)
    ↓ <2>
Cutting start point (*2)
    ↓ <3>
Cutting motion
    ↓ <4>
Retraction
    ↓ <5>
Cutting start point (*3)
    ↓ <6>
Approaching point (*1)
    ↓ <7>
Tool-change point

(*1) An attempt to directly move the tool from the tool-change point to the cutting start point may fail because the tool strikes against an obstacle such as the workpiece. For this reason, the tool is moved safely via the approaching point to the cutting start point so that it does not strike against any obstacle.
(*2) The cutting start point is the position at which a cutting motion starts. The tool may not pass through the cutting start point depending on the type of machining.
(*3) The tool may not pass through the cutting start point depending on the type of machining.

The following sections describe machining requiring additional information.

# *D.1*     BAR MACHINING

## (1) Outer roughing

### - Without semifinishing
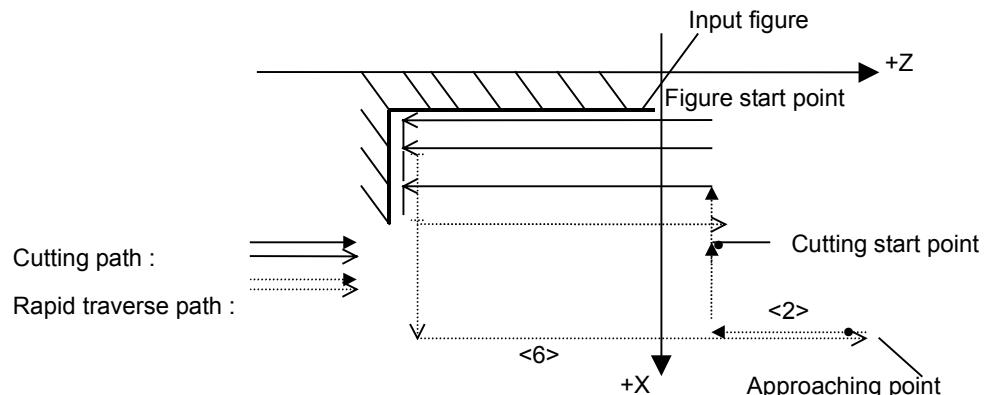


Input figure

+Z

Figure start point

Cutting path :

Rapid traverse path :

Cutting start point

<2>

<6>

+X

Approaching point

### - With semifinishing



Input figure

Cutting start point

Semifinishing path

Cutting start X + clearance X

Cutting start X + release amount

<6>

<2>

Approaching point
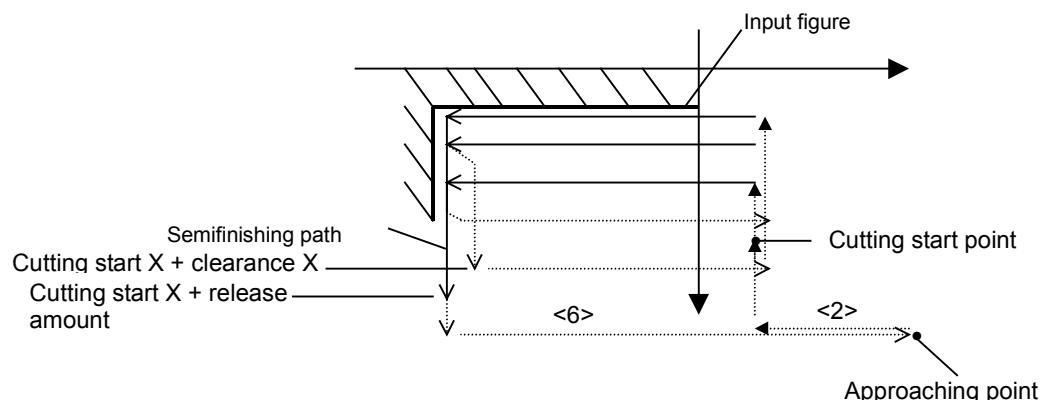
The release amount is specified as an detail process data item.
Clearance X is specified for parameter No. 9784.
In machining in the forward direction, the input figure is defined from the end face of the workpiece to the chuck side.

## (2) Outer finishing



Input figure

Tool-nose radius ×2

Cutting start point

Figure end point X + release amount

<6>

<2>

Approaching point
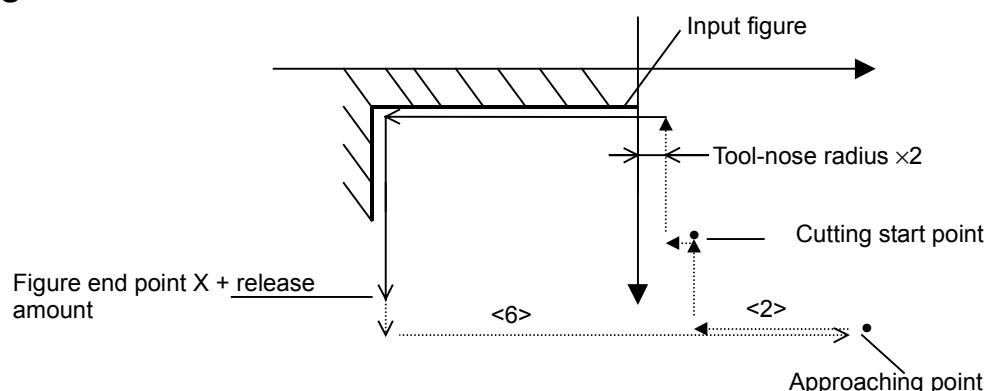
## (3) Outer reverse roughing and finishing

The input figure is defined from the end face of the chuck to that of the workpiece as shown in the figure below. For this reason, the cutting motion is the same as in outer roughing and finishing except for the machining direction.
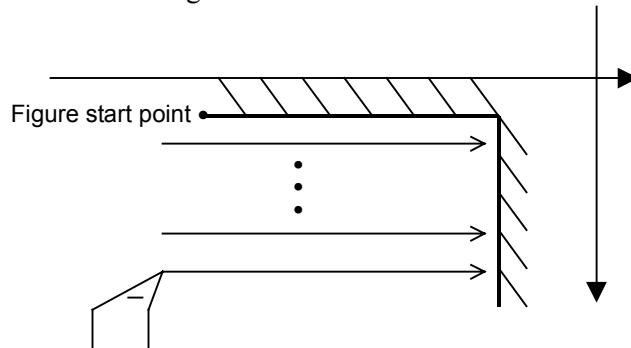
## (4) Inner roughing

### - Without semifinishing
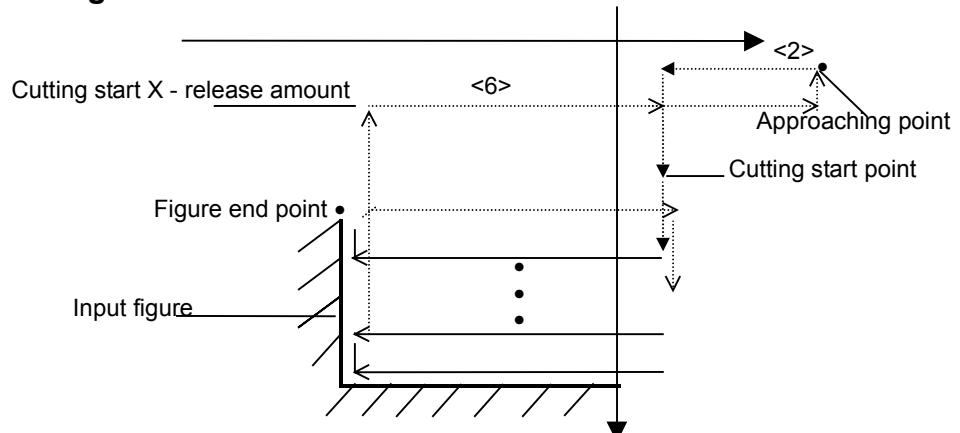
The release amount is specified as an detail process data item.

### - With semifinishing
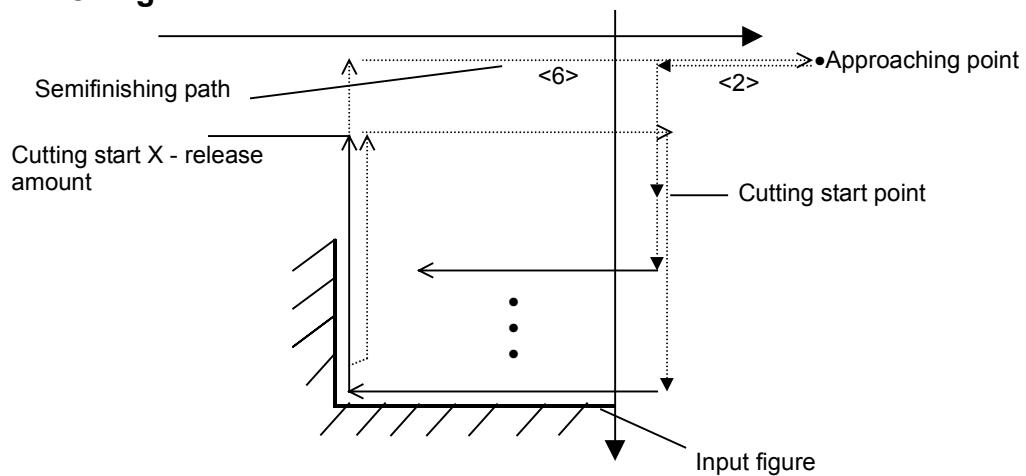
## (5) Inner finishing
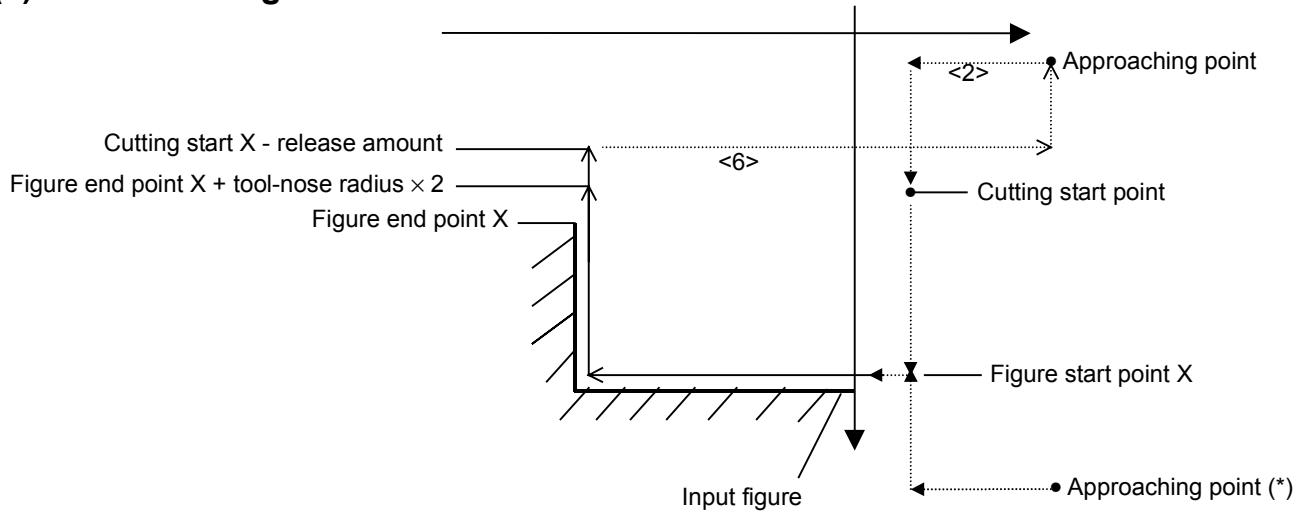


The release amount is specified as an detail process data item.

(*)   When the approaching point is specified outside the cutting start point (+X side), the tool does not pass through the cutting start point.

## (6) Inner roughing and finishing (reverse)

The input figure is defined from the end face of the chuck to that of the workpiece as shown in the figure below. For this reason, the cutting motion is the same as in inner roughing and finishing except for the machining direction.
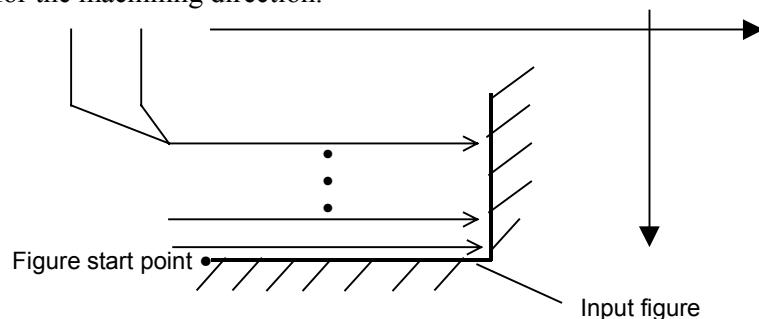
## (7) End face roughing

### - Without semifinishing
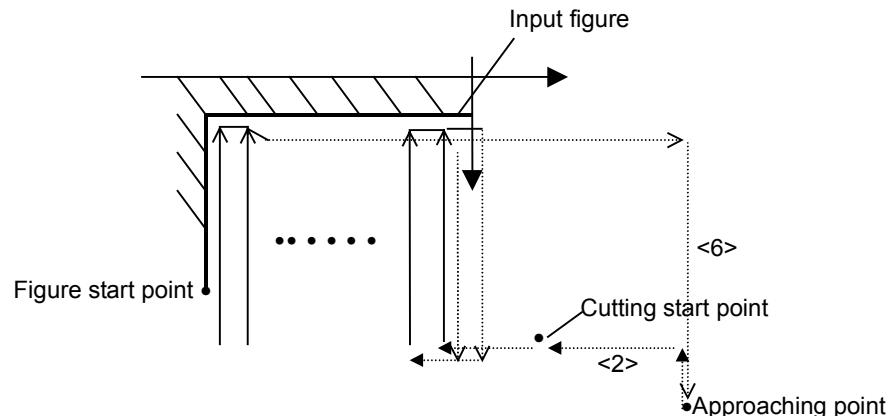


Clearance X is specified for parameter No. 9784.

### - With semifinishing
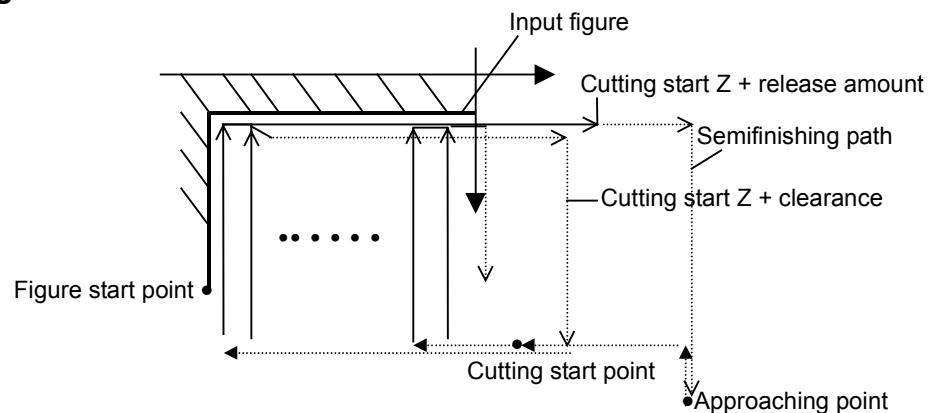


The release amount is specified as an detail process data item.
Clearance Z is specified for parameter No. 9785.
In end facing, the figure is defined from the outer surface of the work-piece (+X) to the center in the direction of the diameter as shown in the above figure.

## (8) End face finishing



The release amount is specified as an detail process data item.

- 307 -
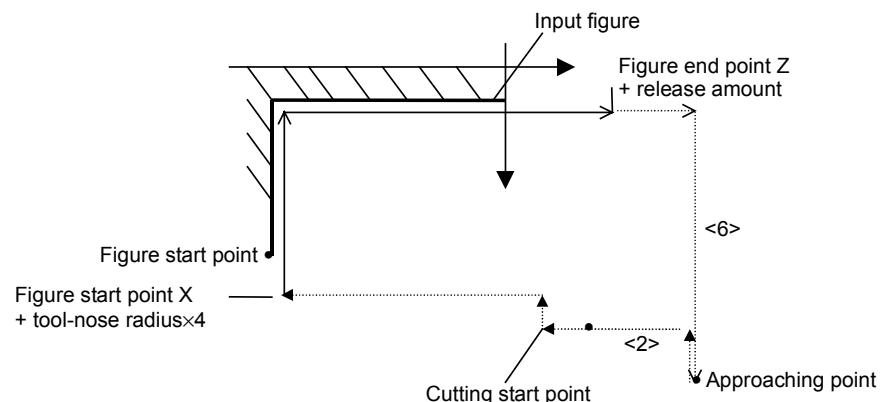
## (9) End face roughing (reverse)

### - Without semifinishing
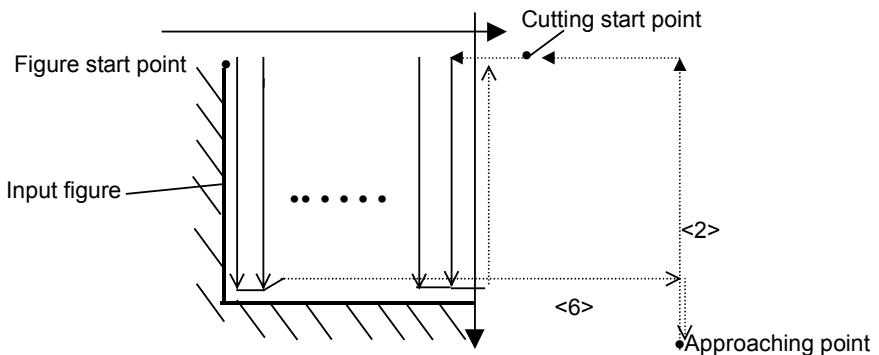


### - With semifinishing
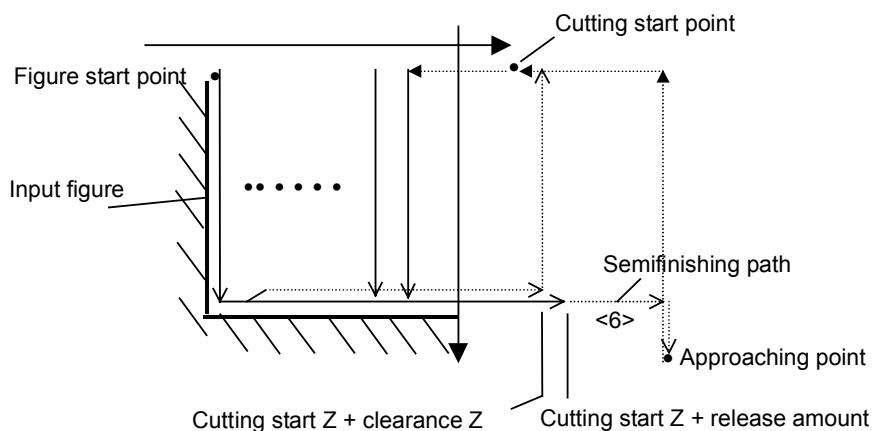


The release amount is specified as an detail process data item.
Clearance Z is specified for parameter No. 9785.
In reverse end facing, the figure is defined from the center of the work-piece to the outer surface (+X) in the direction of the diameter as shown in the above figure.

## (10) End face finishing (reverse)
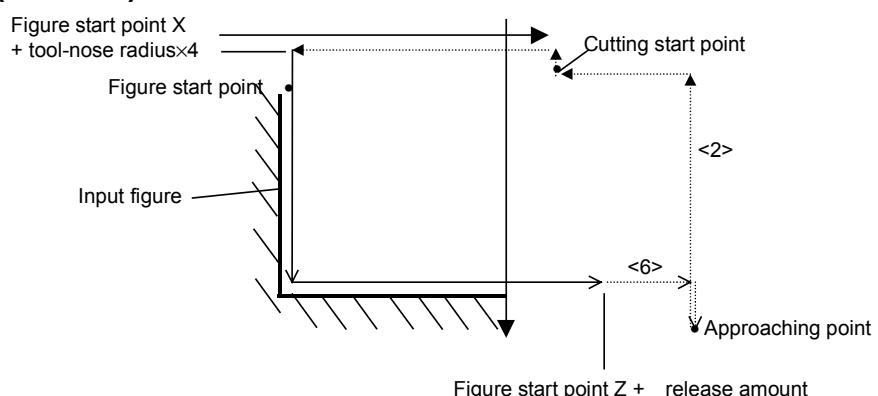


The release amount is specified as an detail process data item.

# *D.2*    **NORMAL GROOVING**

## (1) Outer grooving
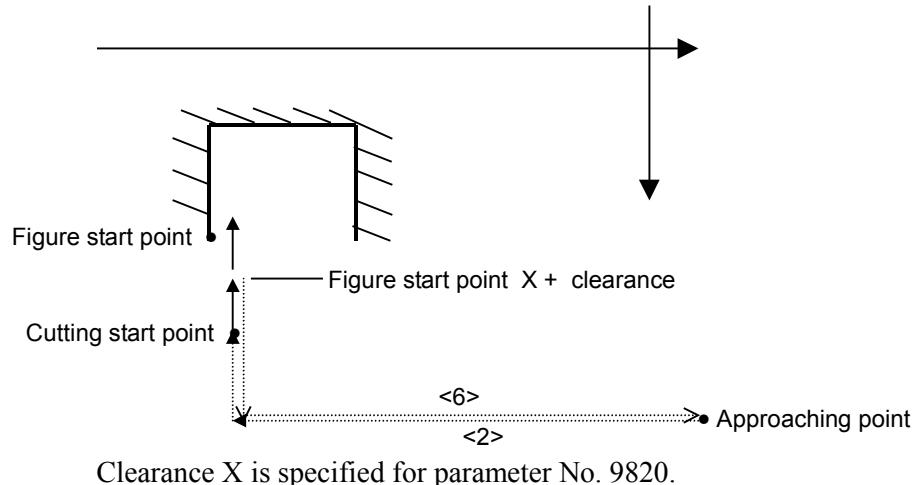


Clearance X is specified for parameter No. 9820.

> **NOTE**
> Be sure to set a point further apart from the center of the blank than figure start point X (+X side) as the approaching point.

## (2) Inner Grooving



Clearance X is specified for parameter No. 9820.
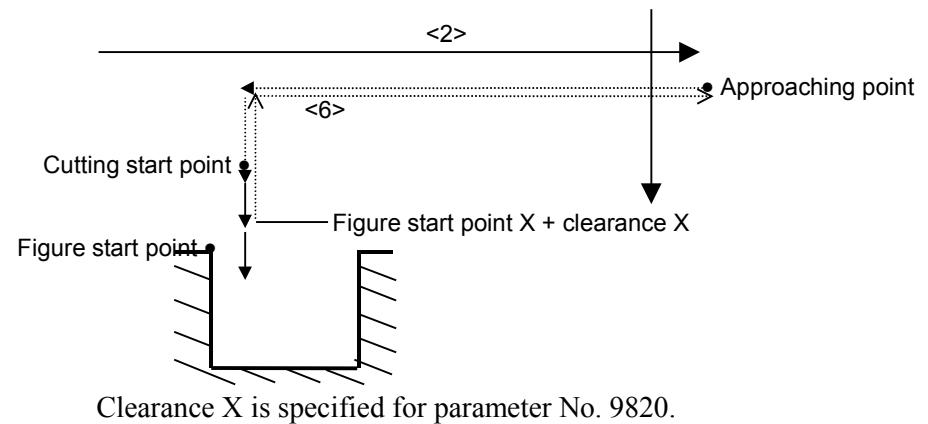
> **NOTE**
> Be sure to set a point nearer to the center of the blank than figure start point X as the approaching point.

## (3) End face cutting
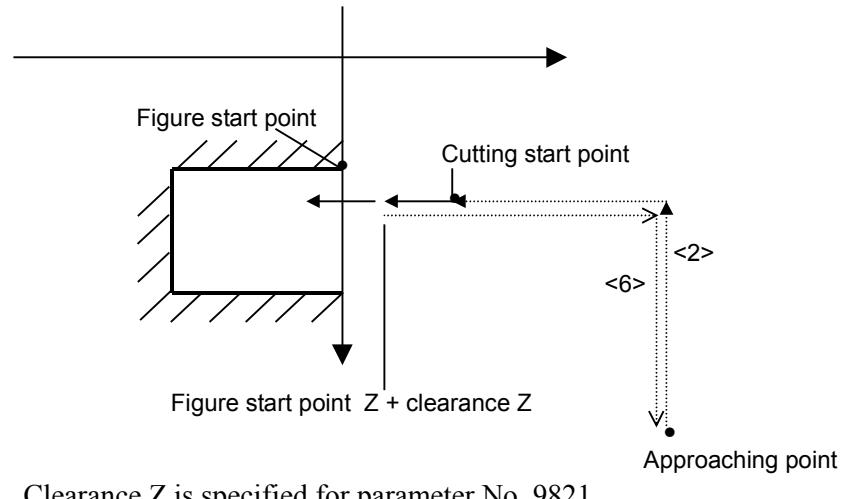


Clearance Z is specified for parameter No. 9821.

# D.3      TRAPEZOIDAL GROOVING

## (1) Outer grooving (Roughing and finishing)
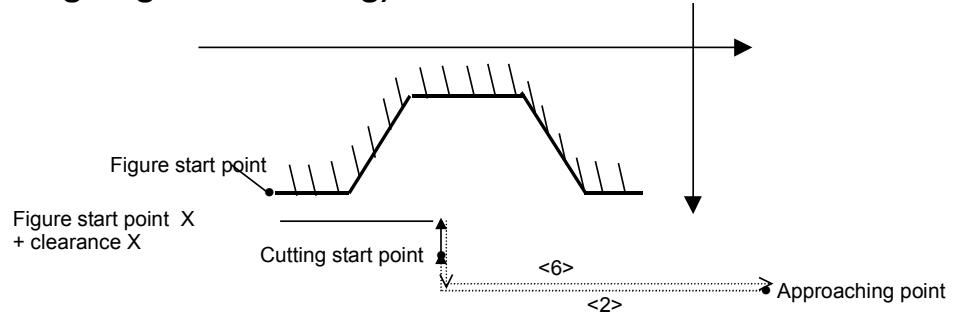


Clearance X is specified for parameter No. 9820.

> **NOTE**
> Be sure to set a point further apart from the center of the blank than figure start point X (+X side) as the approaching point.

## (2) Inner grooving (Roughing and finishing)



> **NOTE**
> Be sure to set a point nearer to the center of the blank than figure start point X as the approaching point.

## (3) End face grooving (Roughing and finishing)



Figure start point

Cutting start point

Figure start point  Z
+ clearance Z

<2>

<6>

Approaching point

# *D.4*    **THREADING**

## (1) Outer threading

Figure start point

Clearance X

Clearance Z

Cutting start point  <2>  <6>

Approaching point

## (2) Inner threading

Approaching point

Cutting start point  <2>  <6>

Figure start point

Clearance X

Clearance Z

Clearance X is specified for parameter No. 9830.
Clearance Z is specified for parameter No. 9831.

# INDEX

Revision Record

## FANUC MANUAL GUIDE FOR LATHE OPERATOR'S MANUAL (B-63344EN)

| Edition | Date | Contents |
|---|---|---|
| | | |
| | | |
| | | |
| 02 | Mar., 2002 | Addition of following items :<br>- Necking Figure Inputting Of Din509-E, Din509-F, and Din76 In III. Types Of Cycle Motions<br>- V. Supplement<br>- Appendix C Dynamic Graphic Display for NC Programs<br>- Appendix D Additional Information on Each Cutting Motion |
| 01 | Jul., 1998 | ——————— |

| Edition | Date | Contents |
|---|---|---|