

GE Fanuc Automation

Computer Numerical Control Products

Symbolic CAP T Basic Module V1

Operator's Manual

GFZ-62824EN/02

November 1998

Warnings, Cautions, and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

GE Fanuc Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

©Copyright 1998 GE Fanuc Automation North America, Inc. All Rights Reserved.

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 danger 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

Notes is used to indicate supplementary information other than Warnings and Cautions.

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

NOTE

In this manual, a program used to specify operations of the CNC machine tool is referred to as "NC data." In CNC machine tool manuals, such a program may be referred to as a "machining program," "part program," or "program," in place of "NC data." The above precautions must also be applied to those programs in a CNC machine tool manual, assuming that the programs have the same usage and purpose as those of the NC data as described in this manual.

GENERAL WARNINGS AND CAUTIONS RELATING TO USE WITH A CNC

This chapter presents the safety precautions which must be observed when the conversational automatic programming function, described in this manual, is used with the CNC machine tool.

WARNING

- (1) Before starting any operation related to the conversational function (such as preparing NC data or running an NC data program), thoroughly check the work area to ensure safety. For example, close the door of the machine, if it is open. Failure to ensure safety may result in death or serious injury. Operating the machine with incorrect NC data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or the tool itself, or injury to the user.
- (2) When the tool offset function is used, before activating the machine, check the direction and value of the offset to ensure that the tool will not collide with the workpiece or machine. Any collision may cause damage to the tool, machine, and/or workpiece, or injury to the user.
- (3) Before starting the NC data program prepared using the conversational function, thoroughly check the NC data to ensure that the tool path and machining processes are set correctly, and that the tool will not collide with the workpiece or machine (including the chuck and tailstock). Before starting a production run, perform a dry run to ensure that the tool will not collide with the workpiece or machine (including the the tool will not collide with the workpiece or machine (including the chuck and tailstock). For example, start the NC data program without mounting a workpiece on the machine. Any collision may cause damage to the tool, machine, and/or workpiece, or injury to the user.
- (4) Before using the conversational function to perform programming, ensure that all the data required for the conversational function, including tool data and cutting condition data, is set correctly. If these data values are not set appropriately, the cutting conditions required for machining may not be set correctly, possibly causing damage to the tool, machine, and/or workpiece, or injury to the user.
- (5) After pressing the power-on button, do not touch any keys on the keyboard until the window is displayed. Some keys are specifically designed for maintenance or other special operations; if any of these keys is pressed before the window is displayed, the machine may behave unexpectedly.

GENERAL WARNINGS AND CAUTIONS RELATING TO USE WITH A PERSONAL COMPUTER

This chapter presents the safety precautions which must be observed when the software, described in this manual, is used with a personal computer, and when the prepared NC data(Note) is used with a CNC machine tool.

WARNING

- (1) Before using NC data with a CNC machine tool to perform a production run, ensure that the machine and tool under the control of the NC data commands will operate safely. If incorrect NC data is used carelessly, or if correct NC data is handled improperly, the machine or tool may behave unexpectedly, possibly causing damage to the machine, tool, and/or workpiece, or injury to the user.
- (2) In this software, once NC data has been automatically created, whether the NC data program runs appropriately for the CNC machine tool is not checked. When using the NC data with a CNC machine tool to perform a production run, follow the instructions in (1) above.
- (3) In this software, whether other software for drawing or checking NC data runs appropriately for the CNC machine tool is not checked. When using the NC data with a CNC machine tool to perform a production run, follow the instructions in (1) above.
- (4) When even proven NC data is used repeatedly, the contents of the NC data may have changed when it is input to the CNC machine tool, due to degradation in the NC data storage, or changes in the communication facilities for data transfer. When using the NC data with a CNC machine tool to perform a production run, follow the instructions in (1) above.

PREFACE

Thank you for purchasing FANUC Symbolic CAP T.

The FANUC Symbolic CAP T software provides total support of all lathing phases from blank figure creation and parts figure creation through to NC data creation. To take full advantage of the functions and features of FANUC symbolic CAP T, users are advised to read and become familiar with the entire contents of this manual.

FANUC Symbolic CAP T is compatible with Microsoft[®] Windows[®]. This manual does not attempt to explain basic Windows operations. Those users who are unfamiliar with Windows should first read the Windows documentation for an explanation of Windows fundamentals.

FANUC Symbolic CAP T uses the following products, whose copyright is owned by Microsoft of the USA.

- Microsoft® Windows® Software Development Kit
- Microsoft® Windows® Visual C++TM
- Microsoft[®] Windows[®] Visual BasicTM
- Microsoft® Windows® Visual Control Pack

NOTE

Microsoft and Windows are registered trademarks of Microsoft Corporation of the USA. Visual C++ and Visual Basic are trademarks of Microsoft Corporation.

Checking the contents of the package

Immediately after opening the package, check that you have received the following items.

- 1) Floppy disks
 - For OPEN CNC
 FANUC Symbolic CAP T Basic module V1 (A08B-9110-J550#EN07)
 - For Personal computer FANUC Symbolic CAP T Basic module V1 (A08B-9310-J550#EN07)
- Protector (Only for Personal computer) FANUC Symbolic CAP T Basic module Protector (A08B-0080-

J550#P550)

 Operator's manual FANUC Symbolic CAP T Basic module Operator's Manual (this document)

About this manual

This manual consists of the sections described below.

PREFACE

Describes the use of this manual and related materials. Also, briefly describes some of the features of FANUC Symbolic CAP T.

1. SETUP

Describes the environment needed to run FANUC Symbolic CAP T, and the method of setting up and enabling the use of FANUC Symbolic CAP T.

2. BASIC OPERATION

Explains how to start and stop FANUC Symbolic CAP T, and describes its basic operations. Also, briefly describes the functions provided by the side menu and menu bar.

3. PROGRAMMING EXAMPLE

Provides an example of the operation of FANUC Symbolic CAP T.

4. BLANK FIGURE CREATION

Describes the types of blank figures, and explains their creation.

5. PARTS FIGURE CREATION

Describes how to create parts figure by means of symbolic figure input.

6. PRE-MACHINING SETTING

Explains how to set the machine's home position and select tooling data; these operations must be completed before machining definition can be performed.

7. MACHINING DEFINITION

Describes the specification of machining types, machining area, and machining conditions, based on the blank and part figures. Also explains how these items can be determined automatically.

8. NC DATA PREPARATION

Describes how to prepare the NC data for a specified process. Also explains how to confirm NC data by animated simulation, and how to print NC data and process list.

9. TOOL PATH EDITING

Explains how to edit tool paths you prepared.

10. SAVING AND LOADING FILE

Describes how to save a newly created program under a specified name, and how to load a program that was created and saved previously.

11. CREATING NEW PROGRAM

Describes how to discard an existing program and create a new one.

12. FIGURE CREATION

Explains the basics of figure creation by means of CAD entry. Also describes how figures are copied and modified.

13. CUTTING CONDITION DATA

Describes how data is registered and modified, including data on the workpiece, tools, and machining conditions, needed to enable the automatic setting of machining conditions.

14. CUTTING CONDITION DATA UTILITY

Explains how to input and output cutting condition data.

15. TOOL DATA AND TOOLING DATA

Describes how to select the tool data to be referenced at Machining Definition, and how to register new tools and modify existing tool data.

16. TOOL/TOOLING DATA UTILITY

Explains how to input and output tool/tooling data.

17. NC MACHINE SETTING

Describes how to specify the technical data for the machine and CNC.

18. FILE TRANSFER

Describes how to transfer NC data to the CNC unit or other units.

19. PLOTTING

Describes how to output NC data from the drawing window to the plotter.

20. CAD DATA INPUT/OUTPUT

Describes the exchange of data between Symbolic CAP T and general CAD software. Also explains the precautions to be observed when using FANUC Symbolic CAP T to handle data prepared in another CAD system.

21. FAPT LANGUAGE FIGURE INPUT

Describes a function that is tended to analyze and execute part program figure definition statements written in FAPT language, and convert them to figure entities that can to be saved to the Symbolic CAP figure databases.

22. DIMENSION MEASUREMENT

Explains how to measure coordinates and distances from figures.

23. STANDARD SETTING

Describes the setting of the initial values displayed on each conversational window.

24. PARAMETER SETTING

Describes the specification of parameters related to FANUC Symbolic CAP T, such as the part figures and tool path colors to be prepared, the sequence in which axes are moved during approach/escape, and whether data is saved automatically at regular intervals.

25. TOOL BAR SETTING

Describes how to modify the contents of the tool bar.

26. TROUBLESHOOTING

Describes the procedure to be followed if a problem occurs. Also lists the errors that may occur.

27. APPENDIX

Contains information that should be read as required, such as the function codes used in NC machine files.

Notations

• Product name abbreviations

Major product names are abbreviated as stated below.

FANUC Symbolic CAP T->Symbolic CAP TFANUC Symbolic CAP T Basic module->Basic moduleFANUC Symbolic CAP T Basic module->ProtectorProtector->Protector

p-4

Microsoft® Windows®

-> Windows

• Keys

When a sequence of keys are to be pressed and held down, the keys in the sequence are indicated with an intervening dash (-), as shown below:

Example) CTRL-ALT-TAB, ALT-F4

• Menus

When a command is to be selected from the menu bars, a right arrow (=>) is used to link the menus in the sequence, as shown below:

"First menu => second menu => third menu"

Example) The menus shown below would be indicated as shown below:

60 ·	D	0.41	D11	E	D	E
Setting =>	Parameter	Setting =>	Blank	Figure.	Parts	Figure
Secting 7		Secting 7				

FANUC Symbolic CAP T - (Untitled) - [No.1]			
<u>Setting</u> <u>H</u> elp			
Parameter Setting	<u>V</u> iew Option		
<u>Standard Setting</u>	<u>F</u> igure Operation		
Library Data Setting	Overall Setting for Symbolic CAP T		
<u>T</u> ool Bar Setting	<u>B</u> lank Figure, Parts Figure		
Environment	Setting of pre-machining		
	<u>I</u> ool Path Creation <u>N</u> C Data Preparation		
	A <u>u</u> to Saving		

• Window examples

The windows shown in this document are merely examples. Note that the window layout and file names used may vary depending on the equipment or the resolution of the display unit you are using.

Features of Symbolic CAP T

• Operation on Windows

Comfortable operating environment is afforded by Windows which provides a de facto standard user interface.

Even inexperienced users can use the system by interactive operation using graphical menus and a mouse.

• User-friendly operating procedure that is easy to use, even by beginners

Symbolic CAP T is easy even for beginners to use, thanks to its support of graphical menus (icons) and mouse-driven conversational processing. In addition, it provides a wealth of help messages.

• Operating procedure for experienced operators

Symbolic CAP T commands can be entered from the keyboard, enabling experienced operators to operate the system quickly.

• Customizing functions

The user can improve the operability or add functions to suit the usage of customers. Customizing to call frequently used functions by a simple operation on tool bar is possible.

• One-click switching of the window

The window can be switched between that for machining definition and that for figure creation simply by clicking a button. Therefore, even if a figure is found to be missing during machining definition, figure input need not be repeated from the very beginning.

• Symbolic figure input

Blank and parts figure can be entered using conventional symbolic figure input based on arrows.

• Fully automatic process determination

An optimum machining type, a tool, a portion to be cut, and machining conditions can be automatically selected simply by entering blank and part figures. Even beginners can create machining programs quickly and easily.

• Machining simulation

Three-dimensional animated machining simulation and tool path drawing are performed simultaneously. Animated machining simulation allows the operator to actually observe the progress of machining. Tool path drawing enables the operator to check the tool path generated by a machining program in detail.

• Tool path editing function

The tool path editing function enables editing tool paths generated on Symbolic CAP T as figure images, using a mouse.

It is possible to re-prepare NC data from edited tool paths.

SAF	SAFETY PRECAUTIONS s-1		
GEN	IERAL WARNINGS AND CAUTIONS RELATING TO USE WITH A CNCs	-2	
GEN PER	IERAL WARNINGS AND CAUTIONS RELATING TO USE WITH A SONAL COMPUTERs	-3	
PRE	FACEp	-1	
	Checking the contents of the package p	-1	
	About this manualp	-2	
	Notationsp	-4	
	Features of Symbolic CAP Tp	-5	
1 9	ETUD	1	
13			
	1.1 Prior to Setting Up (Operating Environment)	. 1	
	1.1.1 Prerequisites	1 د	
	1.1.2 Other usable devices	∠ כ	
	1.2 Setting Up Symbolic CAP 1	. J . 3	
	1.2.1 About Setup	5	
	1.2.3 Reinstalling Symbolic CAP T.	8	
	1.2.4 About the protector (Only for use with a personal computer)	8	
<u>л</u> р		•	
2 D		9	
		. 9	
	2.1.1 Starting Symbolic CAP 1	9	
	2.1.2 Terminating Symbolic CAP 1	9	
	2.1.5 Reading the fatest information about Symbolic CAP 1	۶ و	
	2.2 Contents of Symbolic CAP T Window		
	2.2.1 Title bar	11	
	2.2.2 Side menu	12	
	2.2.3 Menu bar	13	
	2.2.4 Tool bar	13	
	2.2.5 Drawing window	13	
	2.2.6 Prompt area	14	
	2.2.7 Status window	15	
	2.3 Basic Operation	16	
	2.3.1 Using the Help function to become familiar with the operating procedure	16	
	2.3.2 Confirming the version	16	
	2.3.3 Modifying the operation flow	17	
	2.4 Functions Provided by Side Menu	19	
	2.4.1 Programming mode	19	
	2.4.2 Figure creation mode	19	
	2.4.2 Figure creation mode	19 26	
	2.4.2 Figure creation mode	19 26 26	

	2.5.3 Edit menu	
	2.5.4 View menu	
	2.5.5 Auxiliary menu	
	2.5.6 Setting menu	
	2.5.7 Help menu	
	2.6 Functions Provided by the Tool Bar	34
	2.7 Flow of Operation	35
	2.7 1 Overview of each function	35
	2.7.2 Overview of data	
	2.8 Lavers	
	2.8.1 About layers	
	2.8.2 Major classification of layers	
	2.8.3 Figure creation layer setting	
	2.9 Drawing Format Setting	
3	PROGRAMMING EXAMPLE	
	3.1 Flow of Operation	41
	3.1.1 Procedure	41
	3.1.2 Explanation of each step	
	3.1.3 Example of operation	
4	BLANK FIGURE CREATION	
	4.1 Blank Figure Creation	
	4.2 Special Figure	
	4.3 Blank Figure Color/Line Type Modification	
5	PARTS FIGURE CREATION	
	5.1 Turning Figure Creation	
	5.2 Symbolic Input Method	
	5.2.1 Entity selection	61
	5.2.2 Symbolic figure data input	
	5.2.3 Symbolic Figure data editing	
	5.2.4 Symbolic figure creation by figure reference	
6	PRE-MACHINING SETTING	
	6.1 Selecting Tooling Data	
	6.2 Selecting an NC Machine File	
	6.3 Setting the Home Position	
	6.4 Setting the Index Position	
	6.5 Setting the Chuck	
	6.5.1 Setting items of the chuck	
	6.5.2 Chuck figure registration/edit window	
7	MACHINING DEFINITION	
	7.1 Machining Definition Window	
	7.2 Selecting the Machining Type	
	7.3 Specifying the Cutting Area and Condition Data	
	7.3.1 Specifying the Cutting Area	

	7.3.2 Location of the division point and the cutting area	
	7.3.3 Condition setting	
	7.3.4 Specifying Another Area	
	7.4 Changing and Modifying the Machining Process	109
	7.5 Defining New Machining Process	111
	7.6 Fully Automatic Determination	112
8	NC DATA PREPARATION	115
	8.1 Preparing and Displaying NC Data	115
	8.2 NC Data Preparation Window	116
	8.3 Process Order Editing Window	119
	8.4 NC Data Preparation Data Setting Window	121
	8.5 Animation Window	
	8.6 Data Setting Window for Animated Simulation	
	8.7 Writing NC Data to a Floppy Disk and Transferring NC Data to a Peripheral Device	
	8.8 Printing NC Data and a Process Table	
٥		107
3	0.4 Tool Dath Editing	127
	9.1 Tool Faul Euling	127
	9.1.1 Tool paths subjected to enting	127
	9.1.2 Control cutting	
	9.1.4 Editing	
	9.2 Execution of Tool Path Editing	129
	9.3 Tool Path Editing Dialog Box	
	9.3.1 Machining process list	
	9.3.2 Selecting a tool path for editing	130
	9.3.3 Deleting tool paths	131
	9.3.4 Terminating tool path editing	131
	9.4 Editing Movement Entities	132
	9.4.1 Adding movement entities	
	9.4.2 Erasing movement entities	
	9.4.3 Trimming/Extending movement entities	
	9.4.4 Undoing/redoing editing.	
	9.4.5 Specifying start entities	134
	9.4.7 Listing entities	136
	9.4.8 Canceling movement entity editing	
	9.4.9 Terminating movement entity editing	
10	SAVING AND LOADING FILE	138
	10.1 Saving a File	
	10.2 Loading a File	
11	CREATING NEW PROGRAM	140
12	FIGURE CREATION	141
	12.1 Types of Figures	141
	12.2 Specifying a Radius Distance and Δngle	1/12

12.3 Speci	ifying Positions	143
12.4 Selec	ting Entities	
12.5 Coord	dinate System	147
12.5.1	World coordinate system	147
12.5.2	Machining plane coordinate system	147
12.6 Modif	ying Figures by Extension/Trimming	148
12.7 Copyi	ing Figures	149
12.8 Creat	ing Polygons	
12.9 Fitting	g Curve	
12.9.1	Creation method of a fitting curve	
12.9.2	Adjusting the creation method of a fitting curve	
12.10 Offs	et	159
12.10.	1 Outward offset for a corner	
12.10.2	2 Inward offset for a corner	
13 CUTTING	CONDITION DATA	
13.1 Overv	<i>v</i> iew	
13.2 Funct	ions	
13.3 Startir	ng of Editing Cutting Condition Data	
13.4 Addin	ig a Material Name	
13.5 Deleti	ing a Material Name [Edit Cutting Condition Data]	
13.6 Editin	a Cutting Condition Data	
13.6.1	Modifying cutting condition data on a machining type basis	
13.6.2	Adding cutting condition data	
13.6.3	Modifying cutting condition data	
13.6.4	Deleting cutting condition data	
13.6.5	Changing the order of the tool types	
13.6.6	Deleting cutting condition data on a machining type basis	
13.7 Calcu	Ilating Method of the Machining Condition Data	176
14 CUTTING	CONDITION DATA UTILITY	179
14.1 Funct	ions	
14.2 Startir	ng the Cutting Condition Data Utility	
14.3 Selec	ting Utility Functions	
14.4 Outpu	utting Cutting Condition Files	
14.5 Inputt	ing Cutting Condition Files	
14.6 Chan	ging Cutting Condition Files	
14.7 Unitin	g Cutting Condition Files	
15 TOOL DA	TA AND TOOLING DATA	
15.1 Overv	view	
15.2 Funct	ions	194
15.3 Starti	ng of Tool Data/Tooling Data Editing	195
15 4 Additi	ion of Tooling Data	106
15.5 Deleti	ing a Tooling	109 108
15 6 Modif	fication of the Tooling Data	100
15.0 10001	Adding or inserting the tool data for a tool into the tooling data	199
15.6.2	Registering a new tool and adding or inserting the data into the tooling data	
	c-4	

	15.6.3 Modifying tool registered in the tooling data	
	15.6.4 Duplicating a tool in tooling data	
	15.6.5 Changing the order of the tools in the tooling data	
	15.6.6 Deleting tooling from the tooling data	
	15.6.7 Registering a new tool and adding or inserting tool data	
	15.6.8 Modifying tool registered in the tool data	
	15.6.9 Duplicating the tool in the tool data	
	15.6.10 Changing the order of the tools in the tool data	
	15.6.12 Adding or delating a tool material name only	
	15.0.12 Adding of deleting a tool material name only	204
16	TOOL/TOOLING DATA UTILITY	205
	16.1 Functions	
	16.2 Starting the Tool/Tooling Data Utility	
	16.3 Selecting Utility Functions	
	16.4 Outputting Tool/Tooling Files	
	16.5 Inputting Tool/Tooling Files	
	16.6 Changing Tooling Files	
17	NC MACHINE SETTING	
	17.1 NC Machine File	214
	17.2 Overview of NC Data Preparation Procedure	
	17.3 Modifying the NC Data Output Format	
	17.4 Machine Specifications	219
	17.4.1 NC machine name	219
	17.4.2 Home position/index position	
	17.4.3 Total number of axes/number of simultaneously controlled axes	
	17.4.4 Feedrate	
	17.4.5 Spindle speed	
	17.4.6 Whether to use the ATC function	
	17.5 CNC Specifications	
	17.5.1 Address setting	
	17.5.2 G code	
	17.5.3 M code	
	17.5.4 Arbitrary character string	
	17.5.5 Specifying required functions	
	17.6 NC Data Format	
	17.6.1 Example of referencing NC data formats	
	17.6.2 Timing of NC data format reference	
	17.7 Example of NC Data Output	
	17.8 NC Machine Setting Tool Operation	
	17.9 Customization	
	17.9.1 File configuration	
	17.9.2 Modifying initial values	
	17.9.4 D being setting item names	
	1 / .9.4 Deleting setting items	
	1/.y.J Adding setting items	
18	FILE TRANSFER	
	18.1 Function Outline	

18.1.1 Application software		
18.1.2 Devices to which data can	n be transmitted	
18.1.3 Data format that can be the	ransmitted	
18.1.4 Transmission protocols		
18.1.5 File operation functions		
18.2 Activating the File Transmi	ission Tool	249
18.2.1 Activation using the Sym	bolic CAP T output function	
18.2.2 Activating only the File T	Fransmission Tool	
18.2.3 Activating the File Transport	mission Manager	
18.3 File Transmission Tool Wir	ndow	251
18.3.1 Menus		
18.3.2 File specification areas		
18.3.3 Buttons		
18.3.4 Operation guide		
18.4 Operating Procedure		
18.4.1 Selecting the device and s	setting the transmission conditions	
18.4.2 Specifying files	Ç	
18.4.3 Selecting a command		
18.4.4 Changing the contents of	the transmission operation	
18.4.5 Starting transmission	-	
18.4.6 Checking the transmissio	on status	
18.4.7 Ending the transmission of	operation	
18.5 Outline of Processing for E	ach Function	
18.5.1 Send		
18.5.2 Receive		
18.5.3 Delete		
18.5.4 Clear		
18.5.5 Rename		
18.5.6 Read-only and read/write	9	
18.5.7 Refresh		
18.6 Setting Transmission Cond	litions	
18.6.1 Setting procedure		
18.6.2 Specifying the device and	d plotter	
18.6.3 Standard setting files	•	
18.6.4 Setting the communicatio	on conditions	
18.6.5 Setting the transmission of	control conditions	
18.6.6 Setting reception control	conditions	
18.7 Transmission Schedule Ma	anagement	
18.7.1 Contents of the window.	с 	
18.7.2 Changing the schedule		
18.7.3 Exiting the File Transmis	ssion Manager	
18.8 Example Transmission Pro	ocedures	
18.8.1 Transmission to and from	the CNC	
19 PLOTTING		272
19.1 Setting Up a Plotter		272
19.2 Procedures		274
	T	~~~
	I	
20.1 CAD Data Input/Output		

	20.1.1 CAD data	
	20.1.2 Handling of CAD data by this system	
	20.2 DXF File Input/Output	
	20.2.1 Overview of DXF	
	20.2.2 DXF file input	
	20.2.3 Setting the input conversion method	
	20.2.4 DXF file output	
	20.2.5 Setting the output conversion method	
	20.3 Handling CAD Data Efficiently	
	20.4 Conversion Specifications for DXF File Input/Output	
	20.4.1 DXF input conversion specifications	
	20.4.2 DXF output conversion specifications	
21	FAPT LANGUAGE FIGURE INPUT	301
	21.1 Overview	
	21.1.1 Figure types that can be input	
	21.1.2 FAPT statement types that can be input	
	21.2 Operation	
	21.2.1 Start-up	
	21.2.2 Part program selection	
	21.2.3 FAPT language figure input	
	21.2.4 Settings related to the FAPT language figure input function	
	21.2.5 Displaying the progress of input processing	
	21.2.6 Displaying input information.	
	21.3 Gautions and Restrictions	
22	DIMENSION MEASUREMENT	308
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function	
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values	
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement	
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates	
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points	308
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity	308 308 309 310 311 311 313 314
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius	308 308 309 310 311 311 313 314 315
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines	308 308 309 310 311 311 313 314 314 315 316
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements	308 308 309 310 311 313 313 314 315 316 317
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions	308 308 309 310 311 311 313 314 314 315 316 317 318
22	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions	308
22 23	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions	308 308 309 310 311 313 314 314 315 316 317 318 319
22 23	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions STANDARD SETTING 23.1 Standard Setting for Blank Figure and Parts Figure	308 308 309 310 311 313 314 315 316 317 318 319
22 23	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions STANDARD SETTING 23.1 Standard Setting for Blank Figure and Parts Figure 23.2 Standard Setting for NC Data Preparation	308 .308 .309 .310 .311 .313 .314 .315 .316 .317 .318
22 23 24	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions STANDARD SETTING 23.1 Standard Setting for Blank Figure and Parts Figure 23.2 Standard Setting for NC Data Preparation PARAMETER SETTING	
22 23 24	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions STANDARD SETTING 23.1 Standard Setting for Blank Figure and Parts Figure 23.2 Standard Setting for NC Data Preparation PARAMETER SETTING 24.1 Parameter Setting for View	
22 23 24	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions STANDARD SETTING 23.1 Standard Setting for Blank Figure and Parts Figure 23.2 Standard Setting for NC Data Preparation PARAMETER SETTING 24.1 Parameter Setting for View 24.2 Parameter Setting for Figure Operation	
22 23 24	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions STANDARD SETTING 23.1 Standard Setting for Blank Figure and Parts Figure 23.2 Standard Setting for NC Data Preparation PARAMETER SETTING 24.1 Parameter Setting for View 24.2 Parameter Setting for View 24.3 Parameter Setting for Overall Setting of Symbolic CAP T	308 .308 .309 .310 .311 .313 .314 .315 .316 .317 .318
22 23 24	DIMENSION MEASUREMENT 22.1 Dimension Measurement Function 22.2 Measurable Dimensional Values 22.3 Activating Dimension Measurement 22.4 Measuring Coordinates 22.5 Distance between Two Points 22.6 Distance between Point and Entity 22.7 Circle/Arc Radius 22.8 Angles Formed by Two Lines 22.9 Displaying Measurements 22.10 Cautions STANDARD SETTING 23.1 Standard Setting for Blank Figure and Parts Figure 23.2 Standard Setting for NC Data Preparation PARAMETER SETTING 24.1 Parameter Setting for View 24.2 Parameter Setting for View 24.3 Parameter Setting for Overall Setting of Symbolic CAP T 24.4 Parameter Setting for Blank Figure and Part Figure	

	24.6 Parameter Setting for Machining Definition	325
	24.7 Parameter Setting for Machining Condition Automatic Decision	
	24.8 Parameter Setting for Tool Path Creation	
	24.9 Parameter Setting for NC Data Preparation	332
	24.0 Parameter Setting for Auto Saving	333
25	TOOL BAR SETTING	334
	25.1 Tool Bar Setting	
	25.1.1 Storing the tool bar definition file	
	25.1.2 Creating icons	
	25.1.3 Displaying the Tool Bar Setting window	
	25.1.4 Registering menu items	
	25.1.5 Deleting buttons/spaces	
	25.1.6 Inserting spaces	
	25.1.7 Changing icons	
	25.1.8 Saving settings/returning to the normal state	
	25.1.9 Discarding the settings/returning to the normal state	
26		
20	26.1 Before Trouble Occurs	
	26.2 If Trouble Occurs	
	26.3 Error Messages	330
	26.3 1 Abbreviations of functions	340
	26.3.2 Messages related to blank and parts figure preparation	341
	26.3.3 Messages related to pre-machining settings	345
	26.3.4 Messages concerning machining surface setting	347
	26.3.5 Messages related to machining definition	347
	26.3.6 Messages related to NC data preparation	
	26.3.7 Messages concerning the milling library	363
	26.3.8 Messages related to loading part files	
	26.3.9 Messages related to cutting condition setting	
	26.3.10 Messages related to tool and tooling data.	
	26.3.11 Other messages	
27		202
21	27.1 Eurotian Codea	
	27.1 Fullculoff Codes	
	27.1.2 NC data format setting function codes (special functions)	
	27.1.2 NC data format setting function codes (move command words)	
	27.1.4 NC data format setting function codes (G codes)	
	27.1.4 NC data format setting function codes (integer words)	
	27.1.5 NC data format setting function codes (special codes)	
	27.1.7 NC data format setting function codes (N codes).	
	27.1.8 NC data format setting function codes (S codes)	
	27.1.0 NC data format satting function codes (1 codes)	
	27.1.9 NC data format setting function codes (outputting arourary character strings)	
	27.1.10 NC data format setting function codes (modal clearing)	
	27.2.1 General	
	27.2.2 Components	
	21.2.3 Structure	

27.2.4 Declare statement	
27.2.5 Expression and assignment	
27.2.6 Control statement	400
27.2.7 Procedure statement	402
27.2.8 EL statement	405
27.2.9 Preprocessor pseudo instruction	405
27.2.10 Restrictions	406

SETUP

1.1 Prior to Setting Up (Operating Environment)

1.1.1 Prerequisites

The following operating environment is needed to run Symbolic CAP T.

• Computer

	FANUC ope	n CNC (with HSSB personal computer connection), or personal
	computer	
	CPU:	i486 TM DX or higher (math co-processor required)
	Memory:	12 MB or more
	Floppy disk:	3.5-inch, 1.44MB
	Hard disk:	Total capacity of 80 MB or more (20 MB or more of
		dedicated area and 32 MB or more of swap file area)
•	Display un	it
	Resolution:	640×480 dots or higher
	Color:	256 colors
•	Keyboard a	and mouse
	(Essential for	r use with a personal computer)
•	Parallel-in	terface printer cable
	(Essential for	r use with a personal computer)
•	Microsoft@ mode	Windows® Version 3.1 or higher, set to enhanced
NO.	TE	
	For use wit	h a personal computer, printer cable is required to
	connect the	protector.

1.1.2 Other usable devices

• Printer

Windows printer driver

• Plotter

Only for use with a personal computer

NOTE

The plotter having the command system of HP-GL is supported. HP-GL2 or other command system is not supported.

1.2 Setting Up Symbolic CAP T

1.2.1 About Setup

Start with the setup of the Symbolic CAP T basic module. (When the basic module has already been set up, and only the optional module is to be set up, no additional setup is required for the basic module.)

Preparation

 First, create a directory on the hard disk to which the Symbolic CAP T system program will be stored. (Example: C:\CAP T)

NOTE

Do not set up Symbolic CAP T in the root directory (\).

CAUTION

If other programs, such as FANUC PC FAPT, have already been set up, do not set up Symbolic CAP T in the same directory as these programs.

If the same directory is used, both programs may malfunction.

(2) After making sure that the computer and printer are switched off, attach the protector for the Symbolic CAP T basic module.

NOTE

The open CNC architecture does not require this procedure.

CAUTION

The computer and printer must be switched off while the protector is being attached.

If the protector is attached or detached while the computer and printer are switched on, the computer, printer, or protector may be damaged.

NOTE

If other programs, such as FANUC PC FAPT, have already been set up, their protectors can be assembled into the Symbolic CAP T basic module protector. The protectors to be assembled into the Symbolic CAP T basic module protector must be attached before installing Super CAP T. They can be removed after a setting has been made such that they are assembled into the Symbolic CAP T protector. They can be connected in any order.



(3) Switch on the computer and start Windows.

• Setup procedure

- Insert the basic module floppy disk (labeled FANUC Symbolic CAP T Basic module Vx Disk#1) into the floppy drive (where x in Vx is a numeral).
- (2) Select File => Run menu of Program Manager for Windows 3.1, or select Start => Run menu of Task Bar for Windows 95.
- (3) Key in drive-name:\SETUP (where drive-name is the name of the 3.5-inch floppy drive into which the floppy disk has been inserted). Then, click OK.
- (4) When the installation process prompts you to specify where Symbolic CAP T is to be installed, key in the name of the directory where Symbolic CAP T is to be installed, then press the Continue button.
- (5) When the installation process prompts you to specify the data file to be set up, select a data file, then click Continue.

NOTE

When setting up Symbolic CAP T for the first time, all data files must be registered.

(6) Installing the sysytem files to the hard disk begins. As the installation progress, you will be prompted to insert the next floppy disk. Insert the next disk, then click OK. Repeat this until all of the floppy disks have been processed.

(7) When the installation process prompts you to specify the directory into which each type of data is to be stored, change the directory name as required, then click OK.

NOTE

When it is necessary to change a directory name, select the desired data type from the list, then click Change. After selecting or keying in a new drive name and directory name, click OK. Even once setup has been completed, the data directory can be

changed by selecting "Setting => Environment" from the Symbolic CAP T menu.

(8) A prompt is displayed to inquire whether the optional module is to be set up. Click the Yes or No button.

NOTE

When you click the Yes button, the prompt as provided in step (4) is displayed. Insert the optional module floppy disk labeled Disk#1, reenter the installation destination directory name, then select the Continue button. (The optional module must be installed in the same directory as for the basic module.) For details, see Section 1.2.2, "Setting up an optional module."

(9) Finally, a prompt appears asking whether you want to assemble the protectors of other softwares, such as FANUC PC FAPT, into the protector of the Symbolic CAP T basic module. Click Yes or No, as appropriate. If the protectors of other programs have not yet been attached, click No.

NOTE

This question does not appear in the open CNC.

Clicking Yes makes it possible to use other softwares simply by gathering their protector to the Symbolic CAP T basic module protector. After keying in the specification number of the software whose protector you want to gather into the Symbolic CAP T protector, click Gather two protect keys into one protector. (The specification number of the software appears on the floppy disk from which the software is installed.)

When No is clicked, the protectors must be attached individually.

- (10) When a message indicating the normal completion of setup appears, click OK.
- (11) Upon the completion of setup, confirm that FANUC Symbolic CAP T

group window has been added to Program Manager, together with an icon for Symbolic CAP T.

- (12) When a message indicating the normal completion of setup appears, click OK.
- (13) Upon the completion of setup, confirm that the FANUC Symbolic CAP T group window has been added to the Program Manager, together with an icon for Symbolic CAP T.

• Icon registered



Prompts and/or messages not described above may appear depending on the specific installation. In such cases, proceed as indicated on the window.

1.2.2 Setting up an optional module

When using an optional module, set it up once setup of the basic module has been completed. The procedure for setting up a typical optional module is described below. Note that some optional modules require the application of special procedures or precautions. Therefore, always refer to the corresponding operator's manual.

• Preparation

(1) Check the name of the directory where the basic module has been installed.

NOTE

Optional modules must be stored into the same directory as that where the basic module has been installed.

(2) After making sure that the computer and printer are switched off, attach the protectors for the basic module and optional module.

CAUTION

The computer and printer must be switched off while the protectors are being attached.

If the protector is attached or detached while the computer and printer are switched on, the computer, printer, or protector may be damaged.

NOTE

If two or more optional modules are to be set up, the protectors for all those modules can be attached at this time. Protectors can be connected in series, in any order.



(3) Switch on the computer and start Windows.

• Setup procedure

(1) Perform setup in the same way as described in steps (1) to (7) for setting up the basic module.

NOTE

Insert the floppy disk labeled Disk#1 for the module to be set up. As the installation destination directory, specify the directory where the basic module was installed.

(2) Upon the normal completion of setup, a prompt appears asking whether the protectors for the basic and optional modules are to be assembled into a single protector. Click Yes or No.

NOTE

Clicking Yes makes it possible to use Symbolic CAP T without attaching the protectors for the optional modules.

When No is clicked, the protectors for the basic module and each of the optional modules must be attached in series to enable the use of Symbolic CAP T.

Prompts and/or messages other than those described above may appear depending on the specific installation. In such cases, proceed as directed by the prompt.

1.2.3 Reinstalling Symbolic CAP T

When reinstalling Symbolic CAP T in a directory where Symbolic CAP T has already been set up, you can select whether to register the data files again.

When you want to use the existing data, do not register the corresponding data when you are prompted to specify the data file to be set up.

When you want to reregister a data file, if the corresponding data file already exists in the target directory, you will be prompted to specify whether the previous data file is to be backed up. Click Yes or No, as required.

NOTE

When Yes is selected, the existing data is backed up to the BAK subdirectory under the Symbolic CAP T directory. The Symbolic CAP T directory will be rewritten with the standard data. If necessary, however, the previous data can be restored from the BAK subdirectory. When No is selected, the existing data is overwritten with the standard data.

1.2.4 About the protector (Only for use with a personal computer)

The protector must be kept attached to enable the setup and use of Symbolic CAP T. If it is detached, Symbolic CAP T will fail to run normally.

CAUTION

Both the computer and printer must be switched off before attempting to either attach or disconnect the protector.

If the protector is attached or detached while the computer and printer are switched on, the computer, printer, or protector may be damaged.

The protectors for other softwares can be gathered into the protector of the Symbolic CAP T basic module. It is not possible to assemble the protector of the Symbolic CAP T basic module into that of any other software.

If the protectors of other softwares are gathered into the Symbolic CAP T protector, the user need no longer be concerned with the other protectors as part of daily operation. These softwares can be made usable simply by attaching the Symbolic CAP T protector. Note that the individual protectors are required when the corresponding softwares are upgraded. So, they should be kept in a safe place.

If the protectors are not gathered into the Symbolic CAP T protector, they must be attached individually. They can be attached in any sequence.

2 BASIC OPERATION

2.1 Fundamentals

2.1.1 Starting Symbolic CAP T

Double-click the Symbolic CAP T icon.



2.1.2 Terminating Symbolic CAP T

Select File => Exit from the Symbolic CAP T menu.

2.1.3 Reading the latest information about Symbolic CAP T

Double-click the Read Me file icon. The Read Me file contains information relating to Symbolic CAP T that users should be aware of, together with information specific to the installed version. The information in the Read Me file does not appear in the operator's manual. It is imperative, therefore, that the user read the contents of this file.



2.1.4 Other Symbolic CAP T icons

There are several other executable icons for Symbolic CAP T. These correspond to the File Transmission Tool, File Transmission Manager, NC Machine Setting Tool, and Auto Demo.



The File Transmission Tool enables NC data exchange with an external device via a serial port. The File Transmission Manager is an application that manages the transmission schedule and performs the actual communication. Usually, it is not used independently.

The NC Machine Setting Tool is used to edit the contents of the management file that governs the output format of NC data. Using these tools together enables Symbolic CAP T to support a range of CNC units.

Auto Demo is an application that introduces the major functions of Symbolic CAP T.

2.2 Contents of Symbolic CAP T Window

This section describes the Symbolic CAP T windows, using, as an example, the first window to appear after Symbolic CAP T becomes operable.



2.2.1 Title bar



The title bar usually displays the name of the file being processed. If the file has not yet been saved, Untitled is displayed.

2.2.2 Side menu



Side menus contain icons corresponding to the major commands related to programming and figure preparation.

The side menus are configured in a hierarchy.

• Primary menu

Enables the selection of either programming or figure preparation mode.

• Secondary menu

When programming mode is selected, the main menu for programming appears. When figure preparation mode is selected, the main menu for figure preparation appears.

• Tertiary menu

A submenu corresponding to each figure appears only in figure preparation mode.

2.2.3 Menu bar

•	<u>F</u> ile	<u>M</u> odify	<u>E</u> dit	⊻iew	<u>A</u> uxiliary	<u>S</u> etting	<u>H</u> elp	\$
					📶 🗌 💶	-1 - 1		

Commands to be executed can be selected from the menu bar. Command execution can be nested up to about 20 levels.

- File menu Provides the functions used to manipulate files.
- **Modify menu** Provides the functions used to modify individual figure entities.
- Edit menu

Provides the functions used to edit two or more figures at one time.

• View menu

Provides the functions used to display the drawing windows.

• Auxiliary menu

Provides auxiliary functions, such as those for setting drawing formats and confirming entity information.

• Setting menu

Provides functions for setting parameters, such as machining conditions, tool data, and standard values.

• Help menu

Provides functions that help explain the use of Symbolic CAP T.

2.2.4 Tool bar



Those functions that are used most frequently are also provided as icons, so that processing can be simplified. Moreover, the tool bar can be customized as necessary.

2.2.5 Drawing window

The drawing window displays information such as blank figure, parts figure, those figures prepared in figure preparation mode, and tool paths.

When using the mouse to specify a position or select an entity in this window, simply click the corresponding item directly.

2.2.6 Prompt area

Key entry panel									
Radius Mode switch b	Rotation Shorter path								
Position specification panel									
Entity selection panel									
Esc End									
Interrupt release button Prompt	back button								

In the prompt area, the user can enter responses to prompts output by the current processing. System messages also appear in this area.

The prompt area may contain the following entities, depending on the type of the prompt and the particular installation.

- (a) Prompt back button (button)
- (b) Interrupt release button (ESC button)
- (c) Mode switch box
- (d) Key entry panel
- (e) Position specification panel
- (f) Entity selection panel
- (g) Other panels as required

2.2.7 Status window



The status window displays the current status of an entity being prepared, such as color, line type, and layer No. The status of an entity can be modified directly using the status window.

The status window contains the following items.

- (a) Color setting button
- (b) Line type setting button
- (c) Figure search direction switch button
- (d) Machining plane depth entry field
- (e) Drawing layer No. entry field

When the Symbolic CAP T window is small, or if the display unit supports only a low resolution, the status window may not appear. In such a case, access the equivalent function on the menu bar (Auxiliary => Set Current Attributes).

2.3 Basic Operation

2.3.1 Using the Help function to become familiar with the operating procedure

Users can use the Help function to help them become familiar with the operating procedure of Symbolic CAP T. Whenever the user is unsure of how to proceed, accessing the Help function will provide guidance on the correct procedure. This function provides information that does not appear in the operator's manual.

Whenever the Help button appears in a window, double-clicking the button will provide help relating to that window. Whenever there is no Help button, help can be displayed by pressing the F1 key or by selecting the Help menu from the menu bar.

The help menu contains the following submenus.

• Contents

This submenu enables the table of contents to be searched for topics related to Symbolic CAP T.

• Search for Help on

This submenu enables a search to be made for topics related to Symbolic CAP T according to a specified keyword. Selecting this submenu causes a list of keywords that can be used for a search to appear.

• How to Use Help

This submenu explains how to use the help function.

2.3.2 Confirming the version

When it is necessary to confirm the installed version of Symbolic CAP T, such as when performing maintenance, select "Help => About Symbolic CAP T" from the menu bar. The version of Symbolic CAP T and information relating to the computer on which it is running will be displayed, as shown below.
	About Symbolic CAP T		
sanuc	Symbolic CAP T		
Edition 01.1			
Copyrig	ht(C) 1995 by FANUC LTD.		
All right	s reserved.		
Memoru:	45 820 224 hutes Free		
MCIIIUIY.	45,020,224 bytes i lee		
GDI Resource:	55%		
User Resource:	65%		
OS:	Windows 3.10 (Enhanced Mode)		
Disk Operating System 6.20			
CPU:	CPU: 80486 (with Math Coprocessor)		
Video:	Super VGA 1024x768 256 colors Small Fonts		

2.3.3 Modifying the operation flow

Symbolic CAP T commands are executed by selecting from side menus or menu bars. Usually, a conversational window or a string of prompts appear. Operation is performed by responding to the displayed messages or prompts.

This section explains how to abandon the execution of one command in favor of another, as well as how to correct operator errors.

• Abandoning one command and executing another

Using the side menu, it is always possible to switch the command being executed. Clicking an icon halts execution of the current command and starts execution of the command corresponding to the clicked icon.

Whenever it is not possible to click an icon in a side menu, click a button such as OK, Cancel, Close, or Exit in the prompt area or in any current conversational window, then try to click the desired icon again.

• Interrupt by another command

The commands available on a menu bar or tool bar are executed like interrupts. While a command is being executed, if another command is selected from a menu bar or tool bar, the current command is temporarily suspended while the newly selected command is executed. Upon completion of the newly selected command, execution of the original command is resumed from the point at which processing was suspended.

• Exiting an interrupt

Whenever a command is selected from a menu bar or tool bar, the ESC button appears in the prompt area. The command currently being executed can be terminated by clicking the ESC button or pressing the ESC key.

Example) This method can be used, for example, to terminate an edit or modify command that is repeatedly executed for two or more figures, such as figure copy or Trim/Extend.

Command interrupts can be nested up to about 20 levels. Within this limit, commands can be interrupted by others without terminating the original commands.

When an icon on a side menu is clicked, all interrupting commands are terminated automatically, and the newly selected command is executed, even if the user does not intend to terminate those commands.

• Returning to the previous prompt

If the current command displays more than one prompt area, clicking the $\Delta_{\underline{J}}$ button in a prompt area or pressing the CTRL-BS keys returns you back to the previous prompt area (if any).

Example) This function can be used with the command used to define a straight line passing through two points. When you are prompted to enter the second point after entering the first point, you may decide that you want to respecify the first point. In such a case, use this function.

2.4 Functions Provided by Side Menu

2.4.1 Programming mode

The programming mode side menu provides the following functions.

Icon	Function
	Blank figure creation
2	Parts figure creation
	Pre-machining setting
	Machining definition
	NC data preparation
	Tool path editing

2.4.2 Figure creation mode

The figure creation mode side menu enables the generation of the following figure entities.



Linear plural points (with intervals specified)
Linear plural points (with a number of
divisions specified)
Circular plural points (with intervals specified for the arc section)
Circular plural points (with the number of divisions specified for the arc section)
Circular plural points (with the number of
circumference)
Grid plural points
Random plural points
Determine random plural points
Select plural points as the target of operation
Plural points as nodes of a fitting curve

Icon		Function
\angle	Straight line creation	
		Horizontal line
		Vertical line
	+	Horizontal and vertical lines
	<u> </u>	Straight line passing through a specified point and having inclination of specified
		degree.
	<u>></u>	Straight line tangential to a specified circle and having inclination of specified degree.
	<u>/</u>	Straight line passing through two specified
	<u>*</u>	Straight line perpendicular bisector of a line
		segment between two specified points
	<u> </u>	Straight line passing through a specified point and parallel to a specified straight line
	¥	Straight line passing through a specified point and perpendicular to a specified
		straight line
	<u></u>	Straight line passing through a specified point and tangential to a specified circle
	1	Straight line tangential to two specified
	<i>K</i>	circles Straight line having a distance d from a
		specified straight line
	*	Straight line bisecting an angle between two specified straight lines

Icon		Function
	Circle creation	 Circle having a specified point as its center and radius r Circle having a specified point as its center and passing through a specified point Circle having a specified point as its center and tangential to a specified entity
		.Circle of radius r, passing through a specified point, and tangential to a specified entity
		Circle of radius r and passing through two specified points .Circle of radius r and tangential to two
		specified entities .Circle passing through three specified points
		Circle tangential to three specified entities Circle passing through a specified point and tangential to two specified entities
	0	Circle passing through two specified points and tangential to a specified entity
	<u> </u>	Circle for which the difference in radius between the circle and a specified circle/arc is specified value

Icon		Function
\bigcirc	Arc creation	
	<u>e</u>	Arc having a specified point as its center and specified radius
	C	Arc having a specified point as its center and passing through a specified point
	<u>`</u>	Arc having a specified point as its center and tangential to a specified entity
		Arc of specified radius, passing through a specified point, and tangential to a specified entity
		Arc of specified radius and passing through two specified points
		Arc of specified radius and tangential to two specified entities
		. Arc passing through three specified points
		. Arc tangential to three specified entities
	<u></u>	Arc tangential to a specified entity at the start point and another specified entity at the end point, and passing through a specified point
	<u>U</u>	Arc starting at a specified point and
	<u>ប</u>	Arc starting at a specified point, ending at another specified point, and tangential to a
	\bigcirc	specified entity .Arc starting at a specified point, passing through another specified point and
		tangential to a specified entity

Function	
circle by d	
Arc whose radius differs from the radi	ius of a
another arc by d	
Arc whose radius differs from the ra	dius of

Icon

Rectangle creation						
	Recta	angle defi	ned by di	agona	l two points	
	Recta	angle hav	ving the	speci	ified horizo	ntal
	and	vertical	widths,	and	inclination	of
	specit	fied degre	ee			

Icon

Function

占	Polyline/polygon creation
	Line segment starting at a specified point and ending at another specified point
	Arc tangential to a previous entity at the start point
	Arc passing through a specified point
	Arc angential to a specified entity
	counterclockwise
	Arc of specified radius and drawn clockwise
	Determine as a polyline
	Determine as a polygon (by closing the start
	and end points)
	Create a polyline/polygon along other
	entities (trace mode)

+	ut a line segment/arc at an intersection
W	vith another entity and determine as a
po	olyline
—	ut a line segment/arc at an arbitrary point
a	nd determine as a polyline
S	elect a polyline/polygon as a target of
0]	peration
	onnect a polyline/polygon to another entity

Icon

-

tion

2.5 Functions Provided by Menu Bar

2.5.1 File menu

Menu item	Function
• New	Clears the figure and machining specification data currently being created and creates a new figure and machining specification data. When the New command is chosen, the data of the figure and the machining definition data made till then can be preserved.
• Open	Reads a figure and machining data from a file.When the Open command is chosen, the data of the figure and the machining definition data made till then can be preserved.
• Save	Saves the figure and machining data currently being created to a file. Former file is overwritten.
• Save As	Specifies a file name and saves the figure and machining data currently being created to the specified file.
• Clear Machining Data	Clears the machining data currently defined and defines new machining data. When the Clear Machining data command is chosen, the data of the figure and the machining definition data made till then can be preserved.
• Exchange CAD Data	Actives the CAD data conversion.
• DXF-Input	The DXF file data is converted into the figure entity of Symbolic CAP T.
• DXF-Output	The figure element of Symbolic CAP T is converted and the DXF file is output.
• FAPT-Input	The figure definition statement of FAPT language is converted into the figure entity of Symbolic CAP T.
• Plot Out	Outputs a figure and tool path a plotter.
• Plotter Setup	Sets conditions for figure output to a plotter.
• Exit	Terminates Symbolic CAP T. It is possible to save current figure and machining instructions.

2.5.2 Modify menu

Μ	enu item	Function	
•	• Trim/Extend		
	• Intersection	Extends or trims an entity up to an intersection with a reference entity	
	• Intersection - Modal	Extends or trims multiple entities up to an intersection with a reference entity in succession.	
	Nearby Point	Extends or trims an entity up to a near by point of a selected point.	
	• Both Entities	Extends or trims both entities up to an intersection of two entities.	
	• Both Ends	Extends or trims both ends of an entity up to an intersection with two reference entities.	
	• Both Ends - Modal	Extends or trims both ends of multiple entities up to an intersection with two reference entities in succession.	
•	Erase Section		
	• 2 Positions	Erases a section specified by two points from an entity.	
	• 2 Positions - Modal	Erases a section specified by two points from multiple entities in succession.	
•	Break		
	• Intersection	Divides an entity at an intersection with a reference entity.	
	• Intersection - Modal	Divides multiple entities at an intersection with a reference entity in succession.	
	Nearby Point	Divides an entity at a near by point of a selected point.	
	• Both Entities	Divides both entities at an intersection of two entities.	
	• 2 Positions	Divides an entity by a section specified by two points	
	• 2 Positions - Modal	Divides multiple entities by a section specified by two points in succession	
•	Round	Rounds a corner.	
•	Rounds	Rounds all corners of a polygon or polyline	
		at one time.	
•	Chamfer	Chamfers a corner.	
•	Chamfers	Chamfers all corners of a polygon or polyline	
		at one time.	
•	Erase Corner Entity	Erases the specifications of corner round circle and chamfering.	

• Erase Corner Entities Erases the specifications of corner round circle and chamfering for all corners of a polygon or polyline at one time.

2.5.3 Edit menu

Μ	enu	ı item	Function
•	Uı	ndo	Restores the previously created, modified, or
			edited entities to its previous state.
•	Re	edo	Reproduces entities restored by the Undo
			command to its previous state.
•	Er	ase	Erases entities. Multiple entities can be
			selected and erased at one time.
•	Er	ase - Modal	Erases entities one by one in succession.
•	Co) DV	
	•	Parallel-Positions	Copies entities to specified positions in
			parallel mode.
	•	Parallel - Distance	Copies entities to multiple positions at
			uniformly spaced intervals in parallel mode.
	•	Parallel - Division No.	Copies entities in parallel mode to the
			multiple positions determined by equally
			dividing a specified maximum copy distance.
	•	Parallel - Position.	
		Division No.	Copies entities in parallel mode to the
			multiple positions determined by equaly
			dividing the distance to a specified final copy
			destination.
	•	Rotate - Distance	Copies entities to each position determined
			by rotating the entities repeatedly by the
			same angle.
	•	Rotate - Division No.	Rotates and copies entities to the positions
			determined by equaly dividing the angle to
			an end point.
	•	Mirror - Plane	Specifies a mirror plane perpendicular to the
			machining plane, then produces the mirror
			image of entities. The original entities are
			not erased.
	•	Mirror - Position	Specifies a mirror plane perpendicular to the
			machining plane by specifying a base
			position and copy destination. and produces
			the nirror image of entities. The original
			entities are not erased.

Arbitrary Specifies a basic axis for the source and copy destination, and copies a entities. Move Parallel - Position Moves entities to a specified position in . parallel mode. Parallel - Distance Moves entities by a specified distance in parallel mode. Rotate Rotates entities to a specified position. Mirror - Plane Produces the mirror image of entities by specifying a mirror plane perpendicular to the machining plane. The original entities are erased. Mirror - Position Produces the mirror image of entities by • specifying a mirror plane perpendicular to the machining plane and defining a reference position and move destination. The original entities are erased. Arbitrary Moves entities by specifying basic axes for a • source reference point and a move destination reference point. Reverse Keep Original Reverses and copies entities. • The original entities are not erased. Erase Original Reverses and moves entities. The original entities are erased. Assemble Polyline/Polygon Converts the separate entities into a polyline or polygon. Disassemble Polyline/Polygon Converts a polyline or polygon into the separate entities. Offset Keep Original Offsets the entire polyline or polygon. The original entities are not erased. **Erase** Original Offsets the entire polyline or polygon. The • original entities are erased. Offset of Part Keep Original Offsets part of a polyline or polygon. The • original entities are not erased. Erase Original Offsets part of a polyline or polygon. The • original entities are erased. Scale - Copy All Axes Copies entities while changing its size. The

same scaling factor is applied to the x-, y-,

-29-

• Each Axis	and z-axes. Copies entities while changing its size. Different scaling factors are applied to each of the x-, y-, and z-axes.
• Scale - Move	
• All Axes	Moves entities while changing its size. The
	same scaling factor is applied to the x-, y-,
	and z-axes.
• Each Axis	Moves entities while changing its size.
	Different scaling factors are applied to each
	of the x-, y-, and z-axes.

2.5.4 View menu

Menu item	Function
Set View Direction	Sets the direction of display or rotates the display.
• Redraw	Redraw a drawing window.
• Zoom	Enlarges a specified rectangle by mouse to fill the window.
Auto Scale	Redraws a drawing window with automatic scaling to enable the entire entities to be displayed.
• Double	Enlarges a drawing window display by a factor of 2.
• Half	Reduces a drawing window display by a factor of 2.
• Pan	Moves a drawing window display in parallel mode.
• Drawing Layer	Turns on or off drawing layer display.
• Layer for Tool path	Turns on or off tool path creation layer
Creation	display.
• Layer for NC data	Turns on or off NC data preparation layer
Preparation	display.

2.5.5 Auxiliary menu

Menu item	Function
Drawing Format	Sets up a coordinate system according to the format of a drawing.
• Entity Information	Displays information relating to entity (such as entity type, pen number, and coordinates).
• Set Individual Attributes	Changes the attributes (such as the color and pen number) of a entity.
• Set Current Attributes	Sets the attributes (such as the color and pen number) of a entity to be prepared.
• Measurement	
Coordinate Values	Entity start/end/middle point, intersection of two entities, center of a circle/arc, Existing point, and mouse pointer position are displayed.
Distance between Two Points	Distance between two points is measured.
• Distance of Point and Entity	Shortest distance between a point and entity is measured.
Radius of Circle/Arc	Radius of a circle/arc is displayed.
• Angle of Two Lines	Angle formed by two lines is displayed.
• Display Measured	When an entity for which dimension
Information	measurement is finished is selected, the
	information about a entity that contains the selected entity is displayed.
• Layer	
• Setting	Sets the attributes (such as name, drawing, view, and recognition) of layers.
• Move - Entities	Moves specified entities between layers.
• Move - Select Layer	Moves all entities between layers.
• Copy - Entities	Copies specified entities between layers.

• Copy - Select Layer Copies all entities between layers.

Menu item	Function	
Parameter Setting		
View Option	Sets parameters related to displays, such as Axis Mark, Origin Mark, Grid, and Cursor Tracking.	
• Figure Operation	Sets parameters related to figure preparation, modification, and editing.	
• Overall Setting for	Sets general information relating to Symbolic	
Symbolic CAP T	CAP T.	
• Blank Figure,	Sets the system behavior related to blank	
Parts Figure	figure preparation and parts figure preparation.	
• Setting of	Sets the system behavior related to the pre-	
Pre-machining	machining settings.	
• Machining Definition	Sets the length of arrow which indicates an area division point displayed in Machining Definition for Lathe	
Auto Process	Sets the system operations related to the	
Determination	automatic determination of a process, tool, or cutting condition.	
• Tool Path Creation	Sets the system behavior related to tool path creation.	
• NC Data Preparation	Sets the system behavior related to NC data preparation.	
Auto Saving	Sets the parameters needed to automatically save existing figures and machining definition data.	
Standard Setting		
• Blank Figure,	Sets the initial values for the conversational	
Parts Figure	window related to blank figure preparation and parts figure preparation.	
• NC Data Preparation	Sets the initial values for the conversational window related to NC data preparation.	

• Library Data Setting

•

Cutting Condition	Registers and modifies workpiece material,
	tool material, and cutting conditions.
• Tool, Tooling	Registers and modifies tool data and tooling
	data.
NC Machine File	Starts the NC machine setting tool.
Tool Bar Setting	Setting Customizes the tool bar.
Environment	Sets the storage location for each data file.

2.5.7 Help menu

Menu item	Function
• Contents	Finds an explanation from the table of contents.
• Search for Help on	Finds an explanation using keywords.
• How to Use Help	Explains how to use help information.
• About Symbolic CAP T	Displays Symbolic CAP T version information.

2.6 Functions Provided by the Tool Bar

The standard tool bar contains icons corresponding to the following functions.

Icon	Function
2	Reads data from a file.
	Saves the data currently being processed into a file.
2	Cancels the previous operation.
	Recovers a canceled operation.
	Erases figures.
X	Redraws figures.
#	Automatically enlarges or reduces the view
	area.
	Enlarges a specified range within the view
B	Doubles the size of the view area.
	Halves the size of the view area.
	Repositions the view area.
ø	Sets layers.

2.7 Flow of Operation



2.7.1 Overview of each function

• Blank figure creation

Standard blank figures (bars and tubes) can be created simply by responding to prompts displayed by the system. Special figures can also be created using the symbolic input method.

• Parts figure creation

Parts figures (turning figures, screws, grooves, and necking figures) can be created easily using the symbolic input method.

• Pre-machining settings

Using this function, the user can select tooling data and NC machine files, set a home position and index position, and set the chuck prior to machining definition.

• Machining definition

Cutting areas and machining conditions can be automatically determined from previously created blank figures and parts figures, machining types (hole machining, roughing, semifinish machining, finishing, grooving, threading, and so forth), many different cutting condition libraries, and tool data. Moreover, fully automatic process determination is possible.

• NC data preparation

From the specified processes, NC data can be prepared for each NC machine. NC data for a single given process can also be created. Moreover, a check can be made using animated simulation, and NC data and machining process lists can be printed out.

• Tool path editing function

The tool path editing function enables the editing of tool paths generated on Symbolic CAP T as graphic images, using the mouse. It is possible to re-prepare NC data from edited tool paths.

2.7.2 Overview of data

• Blank figure data(*)

Blank figure data includes blank-related data such as blank materials and dimensions.

Blank figure data is created using the blank figure creation function, and is subsequently referenced by the machining definition function and NC data preparation function.

• Parts figure data(*)

Parts figure data represents the figure of a part to be machined. Parts figure data includes figure parameters that have been entered using the symbolic input method, and surface roughness information.

Parts figure data is created by the parts figure creation function, and is referenced by the machining definition function and NC data preparation function.

If parts figure data is deleted, all process data is deleted at the same time.

• Pre-machining setting data(*)

Pre-machining setting data includes selected tooling data, NC machine file names, and set home position and index position data.

Pre-machining setting data is created by the pre-machining setting function, and is subsequently referenced by the turning machining definition function.

• Chuck data(*)

Chuck data includes data such as chuck figures.

Chuck data is created by the pre-machining setting function, and is referenced by the NC data preparation function.

• Process data(*)

Process data includes data specified during machining definition. Process data is referenced by the NC data preparation function.

NC data file

The NC data file holds the NC programs that are ultimately created. An NC data file is created by the NC data preparation function.

-36-

• Tool path data

Tool path data includes data related to tool operation, which is referenced during the creation of an NC program. Tool path data is created by the NC data preparation function, and can be edited by the tool path editing function.

• Execution list

The execution list consists of information relating to errors that occurred during NC data preparation, and the alarms that were issued.

• Machining process list

A machining process list contains the tools used, feedrates, and cutting distances of each process. A machining process list is created with the NC data creation function.

Data marked with (*) is saved to a file by selecting File => Save or File => Save As from the menu, and is read by selecting File => Open from the menu.

2.8 Layers

2.8.1 About layers

Layers are used to make complicated machining areas and figures easy to read, arrange, and check. Layers are not normally required for simple figures, but are useful when handling complicated figures.

Layers can be compared to overlaid sheets of paper: one layer corresponds to one sheet of paper. By modifying the settings for a given layer (sheet of paper) to use for drawing creation, the layer (sheet of paper) to display, and so forth, the entire machining figure can be understood more easily, or the details of a desired machining area can be viewed more closely.

2.8.2 Major classification of layers

Layers can be classified into three types according to their purposes.

• Figure creation layers

For figure creation, up to 256 layers (sheets of paper) can be used. Blank figures, part figures, and machining profiles are automatically assigned to layer 129 and up. Figure entities (such as dots, straight lines, circles, and arcs) created in the figure creation mode of the side menu are created in layer 1 as standard. For each layer, display, recognition, and layer name can be set. From the total of 256 layers, only one layer can be set for figure creation.

• Tool path creation layer

On the tool path creation layer, tool paths are displayed for tool path editing. This layer does not allow figures to be created. The user can switch between display and hide.

• NC data creation layer

On the NC data creation layer, tool paths are displayed when NC data is created. This layer does not allow figures to be created. The user can switch between display and hide.

2.8.3 Figure creation layer setting

By selecting Auxiliary => Layer => Setting from the menu bar or clicking the [Set Layer] button () on the tool bar, the settings below can be made for 256 figure creation layers.

• Figure creation

Specify a layer number for registering figure entities created in the figure creation mode of the side menu. Only one of 256 layers can be specified.

• Display

Specify a layer to be displayed on the window. The figures of specified layers, overlaid over each other, are displayed on the window. The figure entities of layers that are not displayed cannot be selected.

Recognition

Specify whether to select from the figure entities displayed on the window. If non-recognition is specified, the figure entities cannot be selected even if they are displayed.

• Layer name

An arbitrary name can be assigned.

2.9 Drawing Format Setting

- (1) Select Auxiliary => Drawing Format from the menu.
- (2) Select the icon corresponding to the desired drawing format.



When using a horizontal lathe, select one of the four drawing formats above. To facilitate dimension data input at the time of programming, select \overrightarrow{z} or \overrightarrow{z} when the dimensions of a drawing are described with respect to the left side; select \overrightarrow{z} or \overrightarrow{z} when the dimensions of a drawing are described with respect to the right side.

When using a vertical lathe, select one of the four drawing formats listed below.

PROGRAMMING EXAMPLE

3.1 Flow of Operation

This chapter describes the procedure for creating NC data with Symbolic CAP T.

3.1.1 Procedure

Symbolic CAP T is used according to the following procedure:

Figure definition



The user can switch between figure definition and machining definition at any time. So, if the user notices a missing figure during machining definition, it is not necessary to restart from the beginning, thus improving work efficiency.

3.1.2 Explanation of each step



..... Blank Figure Creation button

Clicking this button allows the user to specify routine blank figures simply by responding to displayed prompts, and also enter special figures by applying the symbolic input method. Blank figures can thus be created very easily.



..... Part Figure Creation button

Clicking this button allows the user to create part figures quickly and easily by means of the symbolic input method.



..... Pre-machining Setting button

Clicking this button allows the user to enter a home position, index position, and so forth.



..... Machining Definition button

Clicking this button automatically sets a cutting area and machining condition according to the type of machining to be performed and the contents of the cutting condition libraries. It is possible for the entire process to be set automatically.



..... NC Data Preparation button

Clicking this button allows the user to create NC data for each NC machine, based on the cutting area and machining conditions determined by machining definition.

3.1.3 Example of operation



This section presents an example of the machining performed according to the above drawing. This example does not use fully automatic process determination, but assumes that the machining conditions need not be modified.

NOTE

This example assumes that the values of the cutting condition data, tool/tooling data, NC machine setting, standard value setting, and operation setting items are the initial values immediately after setup.

WARNING

Before starting machining based on NC data prepared according to this example, thoroughly check the contents of the NC data to ensure that the tool path and machining processes are set correctly, and that the tool will not collide with the workpiece or machine (including the chuck and tailstock). Before starting a production run, perform a dry run to ensure that the tool will not collide with the workpiece or machine (including the chuck and tailstock). For example, start the NC data without mounting a workpiece on the machine. Any collision can cause damage to the tool, machine, and/or workpiece, or injury to the user.

(1) Blank figure creation

Click the Blank Figure Creation button, then enter data for each item as follows:

Material A L Blank figure Tube Blank size Diameter : 120.0 Inner dia. : 20.0 Length : 65.0 Base point : 5.0 Standard SR : SR-Mark 2

(2) Part figure creation

Click the Part Figure Creation button to display the symbolic input window. Then, enter part figure data by using the symbolic figure input buttons, as follows:

-		
-6	Start point	Start DX : 30.0
		Start Z : 60.0
	Straight line	End Z: 30.0
		Surface: SR-Mark 2
<u> </u>	Chamfer	Chamfer: 2.0
		Surface: SR-Mark 2
1 0	Straight line	End DX: 37.402
		Surface: SR-Mark 2
L _G	Grooving	Type: Trapezoid
	Grooving	Surface: SR-Mark 2
		Last or Next: On Next element
		Direction:Un
		Width: 15.0
		Depth: 5.0
		Bottom1: Angle
		Angle: (Input not required)
		Bottom2: Angle
		Angle: 60.0
		1st corner: None
		2nd corner: Chamfer
		2nd value: 1.0
		3rd corner: None
		4th corner: None
-6	Straight line	End Z: 0.0
		Surface: SR-Mark 2

T	Threading	Type: General thread
		Last or Next: On Last
		Length: 18.0
		Internal
		Lead: 2.0
		Thread N: 1
		Depth: 1.299
		Angle: 60.0
<u> </u>	Chamfer	Chamfer: 2.0
		Surface: SR-Mark 2
1	C	
	Straight line	End DX: (Input not required)
		Surface: SR-Mark 2
R	Corner R	Corner R: 5.0
		Surface: SR-Mark 2
<mark>. ~</mark> 7	Straight line	End DX: 90.0
		End Z: 20.0
		Angle A: 15.0
		Surface: SR-Mark 2
► _R	Corner R	Corner R: 50
_	conter it	Surface: SR-Mark 2
←,		
4	Straight line	End Z: 40.0
		Surface: SR-Mark 2
_ - G	Grooving	Type: Normal
		Surface: SR-Mark 2
		Last or Next: On Last element
		Direction:Down
		Width: 13.0
		Depth: 5.0
		1st corner: Corner R
		1st value: 1.0
		2nd corner: Corner R
		2nd value: 1.0
		3rd corner: Corner R
		3rd value: 1.0
		4th corner: None
1 8	Straight line	End DX: 110.0
	č	Surface: SR-Mark 2
	Chamfer	Chamfer: 20
	Channel	Surface: SR-Mark 2

←4	Straight line	End Z: 50.0
⊾ _R	Corner R	Surface: SR-Mark 2 Corner R: 1.0 Surface: SR-Mark 2
1 ₽	Straight line	End DX: 120.0 Surface: SR-Mark 2
	Chamfer	Chamfer: 2.0 Surface: SR-Mark 2
←4	Straight line	End Z: 55.0 Surface: SR-Mark 2

(3) Pre-machining settings

Click the Pre-machining Setting button to display the Setting of Premachining input window. Then, select the tooling and NC machine, and specify a home position and index position as follows:

Select Tooling: Tooling 1 Select NC machine: FS16T/18T (Geometry) Home position: DXH = 200.0, ZH = 200.0 Index position: DXI = 150.0, ZI = 150.0



(4) Machining definition

Click the Machining Definition button to display the Machining Definition window. Then, select a machining type by clicking the Machining Type button in the Addition group.

• Hole machining

Add => Hole => Drilling

When a machining type is selected, the machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button to display the Machining Condition window. To modify the cutting area, first modify the value of Depth, then click the Auto button.

To change the cutting area, first change the [Bottom Pos.], then click the [Auto] button.



• Outer surface rough machining (rough end facing)



=> Rough => Outer

When a machining type is selected, the machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button to display the Machining Condition window. To modify the cutting area, select the first and second division points, move the division points with the cursor keys, then specify directions with the numeric keypad.



• Outer surface rough machining

In the above state, click the separate area specification button \checkmark . Then, the machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button \checkmark to display the Machining Condition window. To modify the cutting area, select the first and second division points, move the division points with the cursor keys, then specify directions with the numeric keypad.



• Inner surface rough machining



 \Rightarrow Rough \Rightarrow Inner

When a machining type is selected, the machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button to display the Machining Condition window. To modify the cutting area, select the first and second division points, move the with points the cursor keys, then specify directions with the numeric keypad.



• Outer surface finish machining



The machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button to display the Machining Condition window. To modify the cutting area, select the first and second division points, move the division points with the cursor keys, then specify directions with the numeric keypad.



• Inner surface finish machining



 \Rightarrow Finish => Inner

The machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button to display the Machining Condition window. To modify the cutting area, select the first and second division points, move the division points with the cursor keys, then specify directions with the numeric keypad.



• Outer grooving



 \Rightarrow Rough Finish Groove \Rightarrow Outer

The machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button to display the Machining Condition window.



• Inner grooving



 \Rightarrow Rough Finish Groove \Rightarrow Inner

The machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button to display the Machining Condition window.



• Inner threading



The machining conditions and cutting area are determined automatically. To modify the machining conditions, click the Machining Condition button to display the Machining Condition window.


(5) NC data preparation

Click the NC Data Preparation button to display the NC Data Preparation window. Next, click the start button **D**. Then, the NC data is created. After the NC data has been created, click the execution list display button **D** to display the NC data.

— Syn	nbolic CAP T Cutting	Animation]	
X 🔁 🞞 😆	🚺 🗾 📄 🛛 <u>S</u> ett	ing		
Rotation	Speed			
		50		
—				
Decement No. 1	1924		! 🔄 🕡 🗌	
Program No.:	1234		<u>المع الم</u>	a a
Program Name:	FANUC1234		<u> </u>	
Output File Name:	01234.DAT	Setting	lose Wa	rning- O

😑 Log List - [D:\CAPT.E\TEMP.LOG] 🛛 🔽 🗖	•
<u>F</u> ile <u>E</u> dit <u>O</u> utput	
*** Start of NC data preparation *** 1	ŀ
01234(FANUC1234)	
G28U0.W0.	
G30U0.W0.	
(DRILLING)	
M08	
G97S682M03	
T101	
G99G0028.	
X0.	
G01Z-23.F0.23	
G00Z-20.	
G01Z-51.	
G00Z-48.	
G01Z-71.412	
G 04P 1 000	
G0028.	
M09	
M05	
G30U0.	٢

BLANK FIGURE CREATION

WARNING

Enter correct blank figure data. Failure to enter correct data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user when machining is performed based on the NC data.

4.1 Blank Figure Creation

Click on the side menu to display the following conversational window:

_	Blan	k figure
<u>M</u> aterial AL	Ŧ	
figure	Blank Size	+X
• <u>B</u> ar	<u>D</u> iameter	
○ <u>T</u> ube		+Z D
○ Sp <u>e</u> cial		
	Length	
	Base <u>P</u> oint	
	Standard SR SR-N	Mark 2 🛃 🛛 OK 🛛 Cancel

Material

Select a material name from the displayed list. The material names registered in the machining condition data are displayed.

Blank figure (Bar, Tube, Special figure)

Select a type of blank figure.

Standard SR

The value specified here serves as the initial surface roughness for a turning figure.

Blank size (when Bar or Tube is selected)

Specify the diameter, inner diameter (when Tube is selected), length, and base point of the blank.

As the base point, specify the distance from the face to the origin of the program coordinate system.

4.2 Special Figure

When Special figure is selected, the following conversational window appears:

-	Blank figure	
<u>M</u> aterial AL	_ _	
figure		+X
0 <u>B</u> ar		
O <u>T</u> ube		+Z
• Special		
	Standard SR SR-Mark 2 ±	OK Cancel

Click OK. Then, the special figure creation window appears.

	<u>D</u> elete <u>E</u> x	it
<u>k</u> .2	Alter	

A special figure can be created by means of the symbolic input method. For details of the symbolic input method, see Chapter 5.

NOTES

- When Special figure is selected, a threading figure, grooving figure, nor necking figure cannot be specified.
- In special figure creation, use the positive area (including 0) along the X-axis. If a negative X coordinate is specified, machining definition and animated simulation may not be performed normally.



 For special figure creation, the start point must be the same as the end point. Otherwise, the warning message shown below is output when the End button is clicked. If OK is clicked in response to this message, a connection is automatically established between the start and end points.



4.3 Blank Figure Color/Line Type Modification

When "Setting => Standard Setting => Blank Figure, Parts Figure" is selected from the menu bar, the following conversational window appears to enable blank figure color/line type modification:

— Standard setting co	oncerning blank figure and	part figure
Blank type: Bar	Color/line of blank	OK Cancel
<u>S</u> urface roughness:	Part position for path • Left • Right	

The results of a modification made using this conversational window are reflected when a blank figure is next modified, or a new blank figure is created. A light green, thin dotted line is initially set.

5

PARTS FIGURE CREATION

WARNING

Enter correct parts figure data. Failure to enter correct data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user when machining is performed based on the NC data.

5.1 Turning Figure Creation

Click the button on the side menu to display the first symbolic input window. Note, however, that when a new figure is to be created, the first symbolic input window is skipped, the second symbolic input window being displayed immediately.



[First Symbolic Input Window]

New Figure button

This button is used to delete a previously created figure, and create a new figure.

Modify Figure button

This button is used to edit a part of an existing figure.

Figure Exit button

This button is used to terminate figure creation using the symbolic input method.

Clicking the New Figure button or Modify Figure button causes the second symbolic input window to be displayed. On this window, a figure can be created using the arrow keys.



Symbolic buttons

The symbolic buttons are used to select the element used to constitute a figure.

The user can select Straight line, Arc, Round, Chamfer, Grooving, Threading, and Necking entities. Clicking a symbolic button causes a conversational window to be displayed, thus enabling the user to enter an end point and so forth.

Symbolic list

The buttons corresponding to the entities are displayed as a row, called the symbolic list. Up to 10 buttons can be displayed at any one time. The hidden buttons can be displayed by using the scroll bar.

Scroll bar

The scroll bar is used to scroll the symbolic list. The symbolic list can be scrolled by clicking the left-hand or right-hand arrow button. A desired figure entity on the symbolic list can be displayed by clicking and dragging the corresponding square button on the scroll bar.

The display of the symbolic list can also be scrolled by pressing the Tab key to move the focus to the scroll bar and then pressing the " \leftarrow " or " \rightarrow " key.

Delete button

Clicking this button deletes selected entity from the symbolic list.

Alter button

Clicking this button displays a conversational window for entering data for selected entities in the symbolic list.

Exit button

Clicking this button terminates figure creation. When the Exit button is clicked, the end point of each figure entity is automatically calculated. If an error is detected in the entered data during end point calculation, an error message is displayed.

Figure Reference button

Clicking this button registers figures created using the figure creation function or those entered using the CAD data input function, as entities for creating a figure.

5.2 Symbolic Input Method

5.2.1 Entity selection

Using the symbolic buttons shown below, select Straight line, Arc, Round, Chamfer, Grooving, Threading, or Necking entities.



Entities can be selected in either of the following ways:

- Click the desired buttons on the conversational window.
- By using the keyboard, enter the number or alphabet that appears in the lower-right corner of each desired button on the conversational window.



For the figure shown above, click the following symbolic buttons in the order shown around the outline of the figure.





is represented the end of a figure, and cannot be deleted.

5.2.2 Symbolic figure data input

When a symbolic button is clicked, a conversational window appears to enable the user to enter an end point and so forth. On this window, enter only those items that appear on the drawing. Any items that do not appear on the drawing need not be entered. Click OK or press the Enter key to create the figure. A created figure is displayed in the symbolic list. The selected entities can be invalidated by clicking the Cancel button or by pressing the Esc key.

If sufficient data is not entered to create a figure, a warning message is displayed.



To enter an increment, key in a value followed by the letter I in the edit box. The direction of increment is indicated using the algebraic sign of the value entered rather than an arrow. The value entered in the edit box is converted to absolute coordinates instantly when the letter I is entered.

Example: 25I

A value corresponding to the current position plus 25 is displayed in the edit box.

(1) Start point (Symbolic input)

When the first entity is selected, the following conversational window appears to prompt the user to specify the start point of the figure:

-	Start point	
Start D <u>X</u>		OK
Start <u>Z</u>		Cancel
Parts posit	tion for path	
● <u>L</u> eft		
⊖ <u>R</u> ight		

Parts position for path

Specify the side where the figure to be entered is located, as viewed in the direction of the arrow for the selected entity.

(2) Straight line (Symbolic input)



clicked, the following conversational window appears to enable straight line entity input.

_	Straight li	ne	
~ ?	 		-
End D <u>X</u>	□ Tangent	<u>L</u> ast Nevt	
End <u>Z</u>	- Tangent	<u>H</u> evi	Cancer
Angle <u>A</u>	<u>S</u> urface	SR-Mark 2	:

End DX

Enter the X coordinate of the end point of the straight line, using a diameter value.

End Z

Enter the Z coordinate of the end point of the straight line.

Angle A

Enter the angle between the Z-axis and the straight line.

NOTE

Some items may not be displayed, depending on the type of straight line and the figure data for the previous entity.

(3) Arc (Symbolic input)





is clicked, the following When any of symbolic buttons conversational window appears to enable arc entity input

	Circle		
S			
End D <u>X</u>	□ Tangent I	_ast Next	OK
End <u>Z</u>	- rangent		
Radius <u>R</u>	<u>S</u> urface	SR-Mark 2	Ŧ
Center CDX(<u>I</u>)			
Center CZ <u>(K</u>)			

End DX

Enter the X coordinate of the end point of the arc, using a diameter value.

End Z

Enter the Z coordinate of the end point of the arc.

Radius R

Enter the radius of the arc.

Center CDX

Enter the X coordinate of the arc center, using a diameter value.

Center CZ

Enter the Z coordinate of the arc center.

Tangent Last

Specify whether the arc is tangent to the previous entity.

Tangent Next

Specify whether the arc is tangent to the next entity.

When multiple intersections occur

The following conversational window is displayed if multiple intersections (contact points) occur when a straight line intersects (or touches) an arc, or if an arc intersects (or touches) another arc:



Select the position of an intersection, as viewed from the arc center, with the following buttons:



(4) Chamfer (Symbolic input)

When symbolic button is clicked, the following conversational window appears to enable chamfer entity input.

Chamfer	
	OK Cancel
SR-Mark 2 🛓	
	SR-Mark 2

Chamfer

Enter a chamfer value.



Surface

The standard SR value, set on the conversational window for blank figure creation, is displayed as the initial value. Change the value according to the drawing. The feedrate for finish machining is automatically determined according to the surface roughness set here.

(5) Round (Symbolic input)

When symbolic button is clicked, the following conversational window appears to enable round (corner R) entity input.

-	Corner R	
⊾ _R		
Corner <u>R</u>		OK Cancel
<u>S</u> urface	SR-Mark 2	
<u>S</u> urface	SR-Mark 2 🛓	

Corner R

Enter a radius for corner R.



Surface

The standard SR value, set using the conversational window for blank figure creation, is displayed as the initial value. Change the value according to the drawing. The feedrate for finish machining is automatically determined according to the surface roughness set here.

(6) Threading (Symbolic input)

When symbolic button is clicked, the following conversational window appears to enable threading entity input.

	Threading
Type(<u>G</u>) Gener	al thread 🛃
Last or Next ○ On <u>L</u> ast ● On <u>N</u> ext	Length(<u>W</u>) OK External(<u>I</u>) O Internal(<u>J</u>) O Face(<u>K</u>) Lead(<u>P</u>)
	Thread N(M) 1 Depth(D)

Type

As the type, select General thread, Metric thread, Unified screw, PT/PF thread, or Trapezoidal screw. Once a type has been selected, the fields for setting data for that type appear. If PT/PF screw is selected, the taper angle for threading is not determined. Instead, the angle of the immediately preceding or following straight line entity is adopted as the taper angle for threading.

Last or Next

Specify whether to create a threading entity on the next entity or on the previous entity. Note that this specification determines the direction of threading.



are clicked in the order shown, and the On Next

entity is selected





are clicked in the order shown, and the On Last

entity is selected



Length

Specify the length of the threading. When chamfering is used, the length of the threading must be calculated starting at the intersection of the straight line entities before chamfering.



External/Internal/Face

Select a threading type. The constant applied to automatic thread depth calculation varies with the type of the threading. This item is valid only when automatic thread depth calculation is specified. For some types of threading, this item need not be selected. The initial values for the constants are as follows:

External/Face: 0.7578

Internal: 0.6495

When Setting => Parameter Setting => Blank figure, Parts figure is selected from the menu bar, the conversational window for constant modification appears.

Lead and Thread N

The lead value is the distance traveled along the Z-axis for each rotation of the thread. For a single thread, the lead is equal to the pitch. For an n-start thread, the lead multiplied by 1/n equals the pitch. For a taper thread, the lead calculation is made along the Z-axis.

Depth

For a general thread, the thread depth is calculated using the following formula:

(Thread depth) = ((lead)/(number of starts)) \times (thread constant)

When a lead value and a number of starts (Thread N) are set, the thread depth is calculated automatically. If an automatically calculated thread depth is modified, however, thread depth calculation is not repeated if the lead and the number of starts (Thread N) are changed again.

NOTE

For a unified screw or PT/PF screw, an automatic thread depth calculation is performed if the number of crests per inch is specified.

Angle

Specify a thread angle between 0 and 90.

When the straight line entity on which a threading entity is placed is tapered

When the straight line entity is tilted by up to 45 degrees relative to the Z-axis, the thread is an external thread or internal thread. When the straight line entity is tilted by more than 45 degrees relative to the Z-axis, the thread is a face thread.



When the straight line entity immediately following a threading entity is tapered

Threading starts at the intersection of the straight line connecting the thread roots and the subsequent straight line entity.

In this case, note that the length of the thread actually produced will differ from the specified length.



(7) Grooving (Symbolic input)

When symbolic button **use** is clicked, the following conversational window appears to enable grooving entity input:

—	Grooving
Type Normal 🛓	Surface SR-Mark 2 ±
O On <u>L</u> ast element ● On <u>N</u> ext element	1st corner None ± OK Cancel
Direction(P) Down ±	2nd corner None 🛓
Depth(<u>D</u>)	<u>3</u> rd corner None ±
	4th corner None ±

Туре

Select Normal, Slant, or Trapezoid as the type of groove.

a. Normal grooving (Symbolic input)

Surface

The standard SR value, set using the conversational window for blank figure creation, is displayed as the initial value. Change the value according to the drawing. The feedrate for finish machining is automatically determined according to the surface roughness set here.

Last or Next

Specify whether to create a groove on the next entity or on the previous entity.



Width

Specify the width of a groove. When a trapezoidal groove is to be created, specify the width of the opening. Note that if the groove width extends over two or more entities, a groove figure cannot be created.

Depth

Specify a groove depth.

Direction

Specify the direction of the groove to be created, by selecting one of the arrows, as shown below.



1st corner/2nd corner/3rd corner/4th corner

For the corners of a groove, chamfer or corner R can be specified. When chamfer or corner R is selected, specify a chamfer value or a radius for corner R in response to the corner value input prompt.

Corner numbers are assigned in the order of the figure advance direction.





b. Slanted grooving (Symbolic input)

In addition to the data entry fields for Normal, a field for specifying the angle is also displayed.

Angle

Specify the angle of the wall of the groove to be created.





c. Trapezoid grooving (Symbolic input)

In addition to the data entry fields for Normal, a field for bottom specification and another for width specification are displayed.

Bottom1, Bottom2

Select Angle or Width.

Angle

When Angle is selected in bottom specification, specify the angles shown below. The order in which these angles are specified depends on the advance direction of the figure.



Width

When Width is selected in bottom specification, set the widths shown below. The order in which these widths are specified depends on the advance direction of the figure.



NOTE

No grooving entity is drawn until the next entity has been determined.

(8) Necking (Symbolic input)

When symbolic button is clicked, the following conversational window appears to enable necking entity input.

-	Necking
Type(<u>N</u>) General 🛃	
Width(<u>W</u>)	1st corner None Cancel
Depth(<u>D</u>)	
approach angle(<u>A</u>)	2nd corner None 🛓
Relief value(<u>R</u>)	3rd corner None
Relief height(<u>H</u>)	
Relief angle(<u>B</u>)	4th corner None ±
Surface SR-Mark 2 ±	5th corner None 🛨

Туре

Select General, Grinding1, Grinding2, or Threading necking as the necking type.

When Grinding1 or Grinding2 is selected, a necking width, necking depth, and corner value are determined automatically (according to the DIN standard), provided the previous figure has already been defined. When Threading necking is selected, a necking width, necking depth, and corner value are determined provided the previous figure has

already been defined, and that that figure is subsequent to a threading figure.

Surface

The standard SR value, set using the conversational window for blank figure creation, is displayed as the initial value. Change the value according to the drawing. The feedrate for finish machining is automatically determined according to the surface roughness set here.

a. General necking (Symbolic input)

When General is selected as the type of necking, the data entry fields shown below appear.

Width/Depth/Approach angle/Relief value/Relief height/ Relief angle



1st corner/2nd corner/3rd corner/4th corner/5th corner

For necking angle setting, chamfer or corner R can be specified. When chamfer or corner R is selected, specify a chamfer value or a radius for corner R in response to the prompt for corner value input.



Numbers are assigned to the corners of a necking figure as shown below.

b. Grinding necking 1 (Symbolic input)

When Grinding1 is selected as the type of necking, the data entry fields shown below appear.

Width/Depth/Approach angle/1st corner/2nd corner

The initial width, depth, and corner value indicated in the table shown below are set. An approach angle of 15 degrees is initially set.



c. Grinding necking 2 (Symbolic input)

When Grinding2 is selected as the type of necking, the data entry fields shown below appear.

Width/Depth/Approach angle/Relief angle/Relief value/1st corner/2nd corner

The initial width, depth, relief value, and corner value indicated in the table shown below are set. An approach angle of 15 degrees, and a relief angle of 8 degrees are initially set.



The initial values indicated in the table shown below are set in the fields for Grinding1 and Grinding2, according to the X coordinate of the previous straight line entity:

Diameter	Width	Depth	Corner	Relief
(X coordinate)			value	value
Less than 18 mm	2.0 mm	0.25 mm	0.6 mm	0.1 mm
Less than 0.7087 inch	0.0787 inch	0.0098 inch	0.0236 inch	0.0039 inch
18 mm to 80 mm	2.5 mm	0.35 mm	0.6 mm	0.2 mm
0.7087 to 3.15 inch	0.0984 inch	0.0138 inch	0.0236 inch	0.0079 inch
80 mm or more	4.0 mm	0.45 mm	1.0 mm	0.3 mm
3.15 inch or more	0.1575 inch	0.0177 inch	0.0394 inch	0.0118 inch

d. Threading necking (Symbolic input)

When Threading necking is selected as the type of necking, the prompts shown below appear.

Width/Depth/Approach angle/1st corner/2nd corner

The width, depth, and corner value calculated using the formulas given below are initially set. The pitch and thread depth values entered for the subsequent threading entity are used. If the subsequent entity is other than a threading entity, no initial value calculation is performed.

Width = $3.5 \times$ pitch Depth = $1.2 \times$ thread depth Corner value = $0.5 \times$ pitch Width 2nd corner 1st corner Depth

An approach angle of 30 degrees is initially set.

NOTE

- A necking entity is not drawn until the next entity has been determined.
- Necking cannot extend over three or more successive entities.
- A necking entity must be created between a horizontal straight line and a vertical straight line.

5.2.3 Symbolic Figure data editing

(1) Selecting entities (Symbolic Figure data editing)

The symbolic list button for the currently selected entity is displayed as though it has been physically pressed. To select another entity, click the corresponding button in the symbolic list.

The currently selected entity can be shifted one entity to right by pressing the + key on the keyboard. Similarly, the currently selected entity can be shifted one entity to left by pressing the - key.

(2) Deleting entities (Symbolic Figure data editing)

To delete an entity, select the entity from the symbolic list, then click the Delete button. When an attempt is made to delete the first entity, the conversational window for setting a start point is displayed. Specify an appropriate start point.

(3) Inserting entities (Symbolic Figure data editing)

To insert an entity between another two entities, select the latter of the two entities in the symbolic list. Then, from the symbolic buttons, select the entity to be inserted. The conversational window for the selected entity is displayed. Then, enter the required data.

(4) Modifying figure data (Symbolic Figure data editing)

To modify figure data select, from the symbolic list, the entity to be modified, then click the Alter button. The conversational window for the selected entity appears. Then, modify the data as required. When an attempt is made to modify the first entity, the conversational window for setting a start point is displayed. Specify an appropriate start point.

NOTE

If the figure data is edited, the drawing may not be updated immediately. This is because intersection calculations will not yet have been made for the entities subsequent to that currently selected in the symbolic list. By advancing from the currently selected entity to the next, the intersection calculation required to update the drawing can be instigated. Note that the drawing of an entity such as a chamfer entity, round entity, threading entity, grooving entity, and necking entity is not updated until the next entity is selected.

(5) Redo and undo (Symbolic figure input)

During symbolic input, the redo and undo operations are disabled. If a redo operation is performed after the completion of symbolic input, the state existing before the start of symbolic input is restored. Subsequently performing an undo operation would restore the state existing at the termination of symbolic input. So, to abandon the editing of a figure for whatever reason, terminate symbolic input, then execute the undo operation.

(6) Color/line of parts figure

When Setting => Parameter Setting => Blank Figure, Parts Figure is Changeselected from the menu bar, the conversational window enabling part figure color/line modification appears.

Setting concerning blank figure and part figure	
Color,Line	ОК
Lype : Part figure 🛨	Cancel
Coefficient of a thread	
<u>O</u> uter: 0.7578 <u>Inner:</u> 0.6000	

Select the item to be modified, then modify its color and line width. Multiple figure types can be selected together for color and line modification. If modification is performed during symbolic input, the modification is not immediately reflected in the drawing; the modification is not reflected in the drawing until after symbolic input has terminated, and the next symbolic input operation is started.

The initial settings are as follows:	
Parts figure	: Light red
Undefined	: Pale yellow
Threading/grooving/necking	: Light blue
Selected figure	: Pale yellow
Selected point	: Light blue

5.2.4 Symbolic figure creation by figure reference

(1) Overview (Symbolic figure creation by figure reference)

By means of the symbolic input function, the user can create a symbolic figure by entering figure-related data on the input window from the keyboard. With the symbolic figure creation function based on figure reference, however, the user creates a symbolic figure by selecting a figure that was previously created by using the drawing function, rather than by entering figure data from the keyboard.

Thus, the user can create a symbolic figure as if the selected figure data had been automatically entered with the data input window.

The symbolic figure creation function based on figure reference can be used to create parts figures and special figures.

Figures that can be referenced

- (a) Figures created with the figure creation function
- (b) Figures entered with the CAD data input function
- (c) Figures created with the figure creation function of the FANUC PC FAPT series

Types of figures that can be referenced

- (a) Straight lines parallel to the machining plane
- (b) Circles parallel to the machining plane
- (c) Arcs parallel to the machining plane
- (d) Polylines/polygons parallel to the machining plane
- (e) Fitting curves parallel to the machining plane

(2) Operation (Symbolic figure creation by figure reference)

The creation of a symbolic figure by means of figure reference is outlined below.



- ① Display the second symbolic input window, then click button (a). A symbolic figure is created immediately before the selected button.
- ② Click button (b). The symbolic figure creation function based on figure reference is started.
- ③ If selected button (a) is located at the start entity, the prompts related to the start point and parts position for the path are displayed as described below.
 - When an end point of a figure to be selected is used as the start point, never select check box (c).
 - When an arbitrary point is selected as the start point, select check box (c), then enter a desired start point.
 - When a part figure is to be created, select Parts position for path.

∠ (C)	
Reference to figu	ıre
□ Specify start-Point Start DX Start Z	OK Cancel
Parts position for path Left <u>R</u> ight	

④ To enable the user to select a reference figure, the following prompts appear:



- Select an entity from the window.
- To perform trimming between the currently selected entity and the next entity to be selected, click YES in box (d).



- Click the End button or the Esc button to terminate the symbolic figure creation function based on figure reference, and return to the symbolic figure input window.
- When the button is clicked, the symbolic figure entity registered last is deleted. Note that it is not possible to delete symbolic figures that were already registered before the symbolic figure creation function based on figure reference was started.
- (5) Depending on the type of a selected figure, the prompt shown below may appear to enable the user to specify the direction of the entity.
- (i) When the selected figure is other than a circle In response to the prompt shown below, select a desired end side for the entity.



- When the button is clicked, the currently selected entity is canceled, and the prompts for entity selection (4) reappear.
- (ii) When the selected figure is a circle

The prompt shown below appears. When the Yes or No button is clicked, a symbolic figure is created. When the Cancel button is chosen, no symbolic figure is created, and the prompts for entity selection ④ reappear.



- When symbolic figure creation terminates normally, the window displays an X at the end point position. Then, the prompts for entity selection ④ reappear.
- ⑦ Repeat steps ④ through ⑤ to complete symbolic figure creation.

NOTE

When a symbolic figure is created by figure reference, not all figure data is referenced, instead only that data necessary for symbolic figure creation being referenced.

For a symbolic figure created by figure reference, the standard SR is already set.

Referenced figures are neither deleted nor deformed as a result of being referenced.

If you want to change the end point of a symbolic figure entity fetched through figure reference, turn on the [Specify End Point] check box.

A figure to be fetched through figure reference must be continuous.

6

PRE-MACHINING SETTING

Pre-machining settings consist of the following:

- Selecting tooling data
- Selecting an NC machine file
- Setting the home position
- Setting the index position
- Setting and selecting a chuck



[Example of the Pre-Machining Setting Window]

6.1 Selecting Tooling Data

Clicking of Select Tooling displays a list of available tooling data. Select the tooling data to be used.

Select Tooling	
Tooling 1	*
Select Tooling	
Tooling 1	
Tooling 1	
Tooling 2	

6.2 Selecting an NC Machine File

Clicking of Select NC machine displays a list of available NC machines. Select the NC machine to be used.

Select NC machine	
F\$15T	, t
Select NC machine	
FS15T	<u> </u>
FS15T	†
F\$16T/18T	
FS15T(Geometry)	¥

After the user selects the NC machine, the home position and index position are automatically changed to the values in the selected NC machine file.

WARNING

Select an NC machine file that matches the CNC machine tool to be used. If a mismatching NC machining file is selected, NC data that does not match the machine tool to be used is output. This may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

6.3 Setting the Home Position

Set the home position. As the initial home position, the values set in the currently selected NC machine file are displayed.



NOTE

This setting item can be suppressed by setting of Setting => Parameter Setting => Setting of Pre-machining.

WARNING

Set correct home position data. Failure to set correct data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

6.4 Setting the Index Position

Set the index position. As the initial index position, the values set in the currently selected NC machine file are displayed.



When Setting of Pre-machining is terminated after the index position has been set, the machining start position for the previously defined machining is calculated automatically.

NOTE

This setting item can be suppressed by setting of Setting => Parameter Setting => Setting of Pre-machining.

WARNING

Set correct index position data. Failure to set correct data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.
6.5 Setting the Chuck

Clicking the button displays the window shown below. Using this window, chuck data can be set.

The chuck selected using this window will be displayed during animated simulation when the NC data is prepared.



WARNING

Be sure to set correct chuck figure data when making a tool and check interference check with the animated simulation function. If incorrect chuck figure data is entered, the animated simulation function cannot make an interference check correctly. This may result in the tool colliding with the chuck, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

6.5.1 Setting items of the chuck

On the window, set the following items:

(1) Currently Selected Chuck

Set the chuck figure to be used for animated simulation, as well as the data for positioning the chuck. The values set here are stored in a part file which is prepared for each program. Animated simulation is based on the values stored in this file.

Chucking Depth ZC

To prevent a workpiece from being held by the chuck at the innermost position, set this item. If this item is not set, the workpiece is held by the chuck at the innermost position.



Holding Diameter XC

When a blank having an special figure is to be used, specify the workpiece holding diameter for the chuck. If the blank is a bar or tube, this item need not be set. If no value is specified, the diameter presented to the end face near the chuck is assumed to be the holding diameter.



Current Selected Chuck

Data for the currently selected chuck is displayed. Double-clicking the chuck displays the registration/edit window, with which the user can edit figure data directly.

To select a chuck from the Registered Chuck list, click the desired chuck, then click the following:



To store the currently selected chuck figure by adding it to the Registered Chuck list, click the following:



(2) Registered Chuck

In Registered Chuck, the chuck figure data to be used for animated simulation is registered or edited. The displayed chuck data is stored in the chcktail.ctf file. This file is stored under the directory containing Tool, Tooling as set in Environment.

To register chuck figure data, click the following:

Register
<u> </u>

Then, the registration/edit window appears.

To edit chuck figure data, double-click the desired chuck. Alternatively, select the desired chuck, then click the following:



Then, the registration/edit window appears.

To delete a registered chuck, select the chuck, then click the following:



6.5.2 Chuck figure registration/edit window

Clicking the Edit button on the Selection /Register of chuck data window causes the following window to appear:



Data set using this window is stored in the chcktail.ctf file. This file is stored under the directory containing Tool, Tooling as set in Environment.

Set the following items using this window:

Name

Specify a chuck name using up to 15 characters or up to 7 double-byte characters. Each name must be unique within the list.

Туре

Select outer claw or inner claw type.

L, L1, W, W1

Set the figure data indicated in the guidance drawing.

7

MACHINING DEFINITION

Symbolic CAP T supports the following machining operations:

- Hole Center drilling Drilling Reaming Tapping
- Rough
- Semi-finish
- Finish

Outer	
Face	
Inner	
Outer(residual)
Face(r	esidual)
Inner(r	residual)

- Rough Groove
- Finish Groove
- Rough Finish Groove

Outer
Face
Inner

Threading Outer



NOTE

When the blank figure and part figure have not yet been defined, the machining definition window is not displayed.

again.

7.1 **Machining Definition Window**

ł button on the side menu displays the machining Clicking the definition window. Using this window, the user can define the machining to be performed.

The machining definition window is shown below.

Machi	ning Definition - Turning						×
No.	Machining Type	T Co	de Tool	Dim.	Machining	Start	Pos.
** *	**************	***********	*******	*******	*********	*****	*******
Edit	t				Automa	tic	
<u>A</u> 0	id Modify Delete	Au <u>×</u> .			Decide	Proc.	<u>C</u> lose
Add	I machining process	s. (Ahead of s	elected it	tem)			

The functions of the machining definition work buttons are as follows:

	Add : Add the machining definition.
	Modify : Changes the machining contents.
	Delete : Deletes the machining definition.
	Decide Proc. : Specifies automatic process determination.
	<u>Close</u> : Ends machining definition, and displays the side menu
1	
	NOTE
	The machining definition work buttons may be enabled or

disabled as follows, according to the machining definition status:



7.2 Selecting the Machining Type

Clicking the **Add** button displays the following Machining Type major classification menu:

Hole	►
Rough	►
Semi finish	►
Finish	►
Rough groove	►
Finish groove	►
Rough finish groove	►
Thread	►

Position the cursor to a desired item. Then, the detail menu appears. Select the machining type to be defined.

Hole	►
Rough	Outer
Semi finish	Face
Finish	Inner
Rough groove	Outer(residual)
Finish groove	Face(residual)
Rough finish groove	Inner(residual)
Thread	

[Example of Detail Items for Rough]

After the machining type has been selected, the cutting area, tool, and machining conditions are determined automatically, then the cutting area definition window, shown below, appears in the prompt area.

When the cutting area cannot be determined automatically, the OK button is disabled OK.

Cutting direct specifi	on cation - Di	vision poin dire	Condit t and d ction s	ion setting - ivision pecification	
Rough (Outer) Progress Pos.1: Pos.2:	 Z:[Z:[0.000] -90.000]	DX : [DX : [0.000] 👯 80.000] 🙀) OK Cancel
				1	

Specification of another area J

[Example of the Cutting Area Definition Window for Rough (Outer)]

NOTE

The cutting area definition window varies depending on the selected machining type.

Clicking the OK button defines the machining data and returns the display to the machining definition window.

Clicking the **Cancel** button does not define the machining data and returns the display to the machining definition window.

The user can manually modify the automatically set data. When the area and conditions for the selected machining type cannot be determined automatically, the cutting area definition window appears, but with the ok button disabled. Enter the required data manually.

7.3 Specifying the Cutting Area and Condition Data

The cutting area definition display supports the following three functions:

- Cutting area definition function
- Condition setting function (B button)
- Function for defining another area (

7.3.1 Specifying the Cutting Area

(1) Hole (center drilling, drilling, reaming, tapping)



Set the hole diameter and bottom position. (For tapping, also set a lead value.) After setting the necessary data, click the <u>Auto</u> button. Then, the tool and machining conditions are determined automatically.

NOTE

For Bottom position, specify the tool tip position as an absolute

(2) Rough, Semi-finish, Finish (Normal) (Outer, Face, Inner)

Rough (0	uter)					
Progress	Pos.1:	-	Z : [0.000]	DX : [0.000]
	Pos.2:	1	Z : [-90.000]	DX : [80.000]

Specify the cutting direction, first division point, second division point, and the dividing direction at each division point to determine the cutting area for rough machining, semi-finish machining, and finish machining with a normal tool.

(a) Specifying the cutting direction

Specify the cutting direction as one of the following four directions:



(b) Specifying the dividing direction

First division point:Indicated by a white arrowSecond division point:Indicated by a green arrow

• Specify the location of a division point.

Pressing a cursor key causes a division point to move together with an arrow. Set the arrow at one end of the cutting area.

If you want to move the division point to an arbitrary location, key in "0" from the numeric key pad. You will be prompted to specify the desired location. When you specify the location of the division point, a point on the parts outline that has the specified location as its neighborhood point is calculated, and the division point is shifted to the calculated location. At the same time, the cutting area is recreated.



• Specifying the direction of an arrow for the division point Select the direction of division from the following eight arrows using the numeric key pad. Specifying the direction of division causes an area that can be cut with the currently selected tool and under the currently selected cutting conditions to be drawn.

Direction of C		Corresponding			5		
division			num	eric l	keys		
~	1	~		7	8	9	
+		-		4	5	6	
1	-	S		1	2	3	

If you want to specify an arbitrary angle, key in "5" from the numeric key pad. You will be prompted to specify the desired angle. The specification of the angle does not depend on the drawing format.

The counterclockwise rotation is represented as a positive number, and the clockwise rotation, as a negative number. The direction of the arrow is set to the specified angle. At the same time, the cutting area is re-created.

	Input Angle of Divided Point Allow.
Lat	

NOTE

Only one direction is determined automatically for the area dividing arrow, depending on the machining type: Outer-surface cutting, inner-surface cutting, or end facing. So, it is not necessarily optimal.

(3) Rough, Semi-finish, Finish (Reverse) (Outer, Face, Inner)

Prev. 🛛 <u>N</u>ext

When an area is left uncut after machining with a normal tool, it can be cut with a reverse tool. Whenever such an uncut area exists, the selected part of the area is drawn. When more than one such area exists, an area can be selected by using the area select buttons (Prev. Next).

An area can also be defined by specifying the cutting direction and division points.

Rough (Outer)			
Progress Pos	.1: 📑 Z:[0.000]	DX : [0.000]
Pos	.2: 🚹 Z:[-90.000]	DX : [80.000]

NOTE

When the division points are specified, they can be moved to an end point of a linear entity, end point of an arc entity, and quadrant division point. To return the cutting area to the previous area, use the area select button.

NOTE

The commands for specifying the cutting direction and division point for this machining type are the same as those for roughing, semifinishing, finishing (forward) (outer-surface, inner-surface cutting, and end facing).

(4) Groove (Rough, Finish, Rough/Finish)

Normal 🛨	Prev.	<u>N</u> ext

Clicking **Normal** displays the following, allowing the user to select one of four groove types:

Normal	
Trapezoid	
Slant	
Free	
Normal	Ŧ

When more than one area exists when Normal, Trapezoid, or Slant is selected, an area can be selected by using the area select buttons $(\underline{Prev. Next})$. The selected groove area is drawn in a different color.

When Free groove is selected, specify two division points to specify an area in the same way as when Rough, Semi-finish, or Finish is selected.



NOTE

When Normal, Trapezoid, or Slant is selected, a groove area created with figure definition "G" must be specified for machining. When Free groove is selected, any figures including figures created with "G" can be machined.

NOTE

The commands for specifying the division point for this machining type are the same as those for roughing, semifinishing, finishing (forward) (outer-surface, inner-surface cutting, and end facing).

(5) Thread (Outer, Front, Inner)



When more than one area exists for threading, use the area select buttons $(\underline{\text{Prev. Next}})$ to select an area. The selected threading area is drawn in a different color.

WARNING

Set correct cutting area data. Failure to set correct data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

7.3.2 Location of the division point and the cutting area

The cutting area is a portion enclosed using lines indicating the direction of the arrow at the division point, the blank figure, and parts figure. A line indicating the direction of an arrow is referred to as a boundary entity. The way a boundary entity is handled changes with the location of the division point. So, the resulting cutting area takes a different figure depending on the location of the division point.



The boundary entity has either of the following two attributes depending on the location of the division point.

- ① Portion of a parts outline
- 2 Boundary line on the tool-nose radius center

In a boundary entity with attribute ①, the boundary entity is provided with the same finishing allowance as for a parts outline (even for machining type other than finishing).



In a boundary entity with attribute 2, the actual cutting area extends beyond the boundary entity by the tool-nose radius, because the tool-nose radius center lies on the boundary entity.



There are two boundary entities for one cutting area. Each boundary entity may have a different attribute depending on the location of each division point. The relationships between the division point and the related attribute are described below.

(1) If a division point is at one end of a parts entity and the angle formed by parts at the division point is smaller than 180 degrees:

If a division point is at one end point of an entity (line segment or arc) forming a parts figure, and the angle formed by parts entities at the division point is smaller than 180 degrees, the boundary entity has the "portion of a parts outline" attribute.



(2) If a division point is at one end of a parts entity and the angle formed by the parts entity and an adjacent part is larger than 180 degrees:

If a division point is at one end of a parts entity and the angle formed by the parts entity and an adjacent part included in the cutting area side is larger than 180 degrees, the boundary entity has the "portion of a parts outline" attribute.



(3) If two division points coincide:

If two division points coincide, the boundary entity has the "portion of a parts outline" attribute.



(4) Other than cases (1), (2), and (3)

If a division point is not in any of cases (1), (2), and (3), the boundary entity has the "boundary line on the tool-nose radius center" attribute.



7.3.3 Condition setting

Clicking the button displays a condition setting data window. On this window, the following condition data can be changed and modified:

- Machining condition data
- Tool data
- Start position/pass position data

🗕 Setti	ing of cutting conditions -	Rough(outer)
<u>C</u> utting data	<u>T</u> ool data	Start/ <u>P</u> ass pos.
Cut type In-feed O offset Clearance O Z axis : 3 X axis : 3 Einish amount D Z axis : 0.2 X axis : 3 X axis : 3 X axis : 3	ption Machining pocket Machining overhang epth of cut epth of cut by once : 2.5 Cut depth rate : 10 Cut depth clamp : 2 st override(%) : 10 ool <u>a</u> nt Non coolant © Oil O M	OK Cancel Save lib. Next page

[Example of Condition Setting Window (Outer Rough Machining)]

Clicking a button displays the corresponding setting window.

📥 Setti	ng of cutting conditions - F	Rough(outer)	
<u>C</u> utting data	<u>T</u> ool data	Start/ <u>P</u> ass pos.	

	Setting of cutting c	onditions -	Rough(outer)	
<u>C</u> utting data	<u>T</u> ool d	lata	Start/ <u>P</u> ase	s pos.
Tool name. : Gen Tool kind. : Gen Tool type : Lat Material name : Can	neral(Out <u>F</u> ace Rou neral tool he 'bide	gh)1		OK Cancel Browse
T <u>o</u> ol code Select no. : 1	Tool <u>d</u> imension Nose R : Cut edge angle : Nose angle :	0.8 1 95 80 I	Fool width : Protect angle :	30 ?
Tool Information Use-1 : Use-2 : Spindle direction :	Roughing Outer and Face Normal	Setting Setting o Setting p Setting p	lirction : Lef position (Z) : 0 position (X) : 0	t

[Example of Tool Data Setting Window]

	ng of cutting conditions - F	Rough(outer)
<u>C</u> utting data	<u>T</u> ool data	Start/ <u>P</u> ass pos.
X (XE2, ZE2) (XE1, ZE1) Start position XSP: 50 ZSP: 1 Approach pass position XA1: ZA1: XA2: ZA2:	(XSP, ZSP) (XA1, ZA1) (XA2, ZA2)	N ZE1: ZE2:

[Example of the Machining Start Pass Position Setting Window]

On the condition setting window, the button is provided for each item. Clicking this button displays an explanatory drawing that describes the data items to be set. Then, the user can set that data.



[Example When the Button for Clearance Amount is Clicked]

The button is provided for each item on each window. For an explanation of the data to be specified, refer to the corresponding window.

WARNING

Set correct machining condition data. Failure to set correct data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

7.3.4 Specifying Another Area

A different area can be machined using the same tool by specifying that area with the button.

First, move the division points to the area to be machined. Specify a tool and the machining conditions as necessary. Then, click the button. This defines the first area.



② Then, move the division points to the area to be machined next. The dividing direction and machining conditions can all be changed. The tool, however, cannot be changed.



③ When no further cutting areas remain to be specified, click the OK button to terminate the specification.

7.4 Changing and Modifying the Machining Process

After machining definition, the machining definition window is displayed, as follows:

		Ма	achining Defi	niti	on - Turnir	īg				
No.	Kind of Machining		T Code	То	ol Dim.		Machining	Sta	nrt Po	s.
01	Hole (Center Drilling)									
	[Area-01	1	T0101	D	18.000	Х	150.000	z	150.0	00
02	Hole (Drilling)									
	[Area-01]	T0202	D	28.000	х	150.000	z	150.0	000
03	Rough (Face)									
	[Area-01]	T0303	R	0.800	Х	150.000	z	150.0	000
04	Rough (Outer)									
	[Area-01]	T0404	R	0.800	Х	150.000	z	150.0	000
05	Finish (Outer)									
	[Area-01	1	T0707	R	0.400	Х	150.000	z	150.0	000
06	Rough Finish Groove (Ou	ter)							
	[Area-01]	T0909	R	0.200	Х	150.000	z	150.0	000
	[Area-02]	T0909	R	0.200	Х	150.000	z	150.0	000
07	Thread (Outer)									
	[Area-01	1	T1010	R	0.250	Х	150.000	z	150.0	000
	[Area-02]	T1010	R	0.250	Х	150.000	z	150.0	00
××	*****	×××	*****	××	****	××>	*****	жжжэ	кжжжжж	жжж
	dd Data differi Data da									(Free all
	aa Moany Delete							AUT	0	End

[Example of List Window Displayed after Machining Definition]

(1) Modifying the process

Modify the process in either of the following ways:

• Position the cursor to the area line to be modified in the process list, then click the **Modify** button. Then, a prompt for selecting the item to be modified is displayed as shown below. Select the desired item.



• Position the cursor to the area line to be modified in the process list, then double-click that line. The cutting area definition window appears. Using this window, the user can make the necessary modifications.

(2) Adding a machining process

To append a new machining process to the end of the process, perform the following procedure.

• Position the cursor to the end of the process list, click the <u>Add</u> button, and select the machining type to be added.

[Area-02]	T0909	R	0.200	Х	150.000	Ζ	150.000
07 Thread (Outer)								
[Area-01]	T1010	R	0.250	х	150.000	z	150.000
[Area-02	1	T1010	R	0.250	х	150.000	z	150.000

[Positioning of Cursor to Add a Machining Process]

To insert a machining process at a point in the middle of a process, perform the following procedure.

Position the cursor to the point immediately after the machining number line in the list into which a machining process is to be inserted. Click the Add button and select a machining type to be added.

NOTE

In the example shown below, a new machining process is added between machining process 3 and 4.

	[Area-01]	T0202	D	28.000	Х	150.000	z	150.000	
6	93 Rough (Face)									
L.	[Area-01]	T0303	R	0.800	Х	150.000	z	150.000	
G	04 Rough (Outer)									
	[Area-01]	T0404	R	0.800	х	150.000	z	150.000	
e	95 Finish (Outer)									
	[Area-01]	T0707	R	0.400	Х	150.000	z	150.000	

[Positioning of Cursor to Insert a Machining Process]

(3) Deleting a machining process

To delete a machining process, position the cursor to that machining process, and click the **Delete** button. Then, deletion is performed according to the cursor position, as follows:

- When the cursor is positioned to the machining name All areas subject to that machining process are deleted.
- When the cursor is positioned to an area line Only the specified area is deleted.

7.5 Defining New Machining Process

To delete all defined processes and define a new process, apply either of the following two methods:

- Delete all machining process on the machining definition window.
- Select "File => Clear Machining data" from the menu bar.

<u>F</u> ile	<u>M</u> odify	<u>E</u> dit	<u>V</u> iew	<u>A</u> u>
<u>N</u> ew	/		Ctr	I+N
<u>0</u> pe	en		Ctr	I+0
<u>S</u> av	е		Ctr	I+S
Sav	e <u>A</u> s			
<u>C</u> lea	ar Machin	ing dat	ta	
<u>E</u> xcl	hange CA	D data	ι	Þ
<u>P</u> lot	Out			
Plot	ter Set <u>u</u> p			
Exit				

7.6 Fully Automatic Determination

The fully automatic determination function automatically performs machining definition by using blank figure, part figure, and automatic determination data.

Click the **Auto** button on the machining definition window. Then, the machining process will be determined automatically.

Automatic determination uses the tooling data and machining condition data. The processing is performed according to the machining procedure specified in a machining procedure group. Therefore, these data items must be set in advance.

Clicking the <u>Auto</u> button displays the following prompt area.



(1) Setting a machining procedure group

Clicking the <u>Set...</u> button displays the window for setting automatic determination data.

-	-		Setting Data	of Automatic Pr	ocess Specification	
I	Machining	Group GRO	UP0 🛓			
	Process	Kinds of Mac	hining			
	001	Turning	Drilling		Contor-Drilling	
	001	Turning	UFIIIIng		Center Drilling	<u>↑</u>
	002	furning	Urilling		Urilling	
	003	Turning	Roughing		Face	
	004	Turning	Roughing		Outer	
	005	Turning	Roughing		Inner	
1	006	Turning	Finishing		Face	
	007	Turnino	Finishina		Outer	
	008	Turnina	Finishina		Inner	
	009	Turning	Rough and	Fin Groovina	Outer	
	010	Turning	Rough and	Fin Grooving	Inner	
1	011	Turning	Threading	-	Outer	
	012	Turning	Threading		Inner	+
ļ			_		,	
1	Group Nar	ne: GROUF	0	Halt		
1	Insert	Delete	Register Res	store Settin	End End	
l						



(2) Selecting and renaming a machining procedure group

Clicking GROUPO displays the following, allowing the user to select a desired machining procedure group. Selecting a group causes the display to change from the list to the contents of the selected group. When group "**********" is selected, a new group can be added.

GROUP0	2	F
GROUP0		
GR0UP1		
*******	******	

To rename a group name, enter a new name in response to the prompt, as follows:

Group Name : GROUP0

(3) Adding or inserting a machining process

- ① To append a machining process to a list, position the cursor to the last line of the list.
- ② To insert a machining process, position the cursor to the line immediately after the point where the machining is to be inserted.
- ③ Click the <u>Insert</u> button. Then, a two-level menu appears. Select the machining process to be added or inserted.

Drilling	•
Roughing	Outer
Semi-finishing	Face
Finishing	Inner
Rough Grooving	Outer (Residual)
Finish Grooving	Face (Residual)
Rough and Fin. Grooving	Inner (Residual)
Threading	Þ

[Example of Selecting Outer Rough Machining]

WARNING

When using the fully automatic process determination function, correctly set all the data required for automatic determination including blank figure data, part figure data, tooling data, cutting condition library data, machining method group data, standard value setting data, and operation setting data. If any incorrect data is set, an incorrect machining process is automatically determined. This may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

8

NC DATA PREPARATION

8.1 Preparing and Displaying NC Data

Click the NC data preparation button. On the NC data preparation window, shown below, specify a program number, program name, and output file name, then click the NC data preparation start button. Then, NC data preparation starts. Once NC data preparation has been completed, click the execution list display button. Then, the NC data appears.



NC data preparation button

NC data preparation start button

Execution list display button

8.2 NC Data Preparation Window

Program No. :	1234	
Program Name :	FANUC1234	
Output File Name :	01234.DAT	Setting Close Warning- 0

Clicking the NC data preparation button causes the above NC data preparation window to appear. On this window, operation and setting related to NC data preparation can be performed.



NC data preparation button

- **Program No.** Specify an O number for the NC data.
- **Program Name** Specify a program name (comment) for the NC data.
- Output File Name

Set the name of the file to which NC data will be output. The directory specified by "Setting => Parameter Setting" is used. When a directory has been specified in the Output File Name field, that directory is used.

-114

Output file name reference button

Clicking this button causes the file select window to appear. Using this window, a search can be made for an NC data output file or such a file can be specified.

Delete Tool Path button

Clicking this button causes the Delete Tool Path dialog box to appear. In this dialog box, specify the cutting area for a tool path to be deleted.

NOTE

Tool paths for standard machining are prepared based on machining definition data during NC data preparation. If there is already a tool path, changing cutting conditions using machining definition does not re-prepare the tool path.

To enable machining definition changes for standard machining, it is necessary to delete tool paths before NC data preparation.

To use this function, it is necessary to previously enable the tool path editing function using the following commands:

Setting => Parameter Setting => Overall Setting for Symbolic CAP T ... on the menu bar

Use for Tool Path Editing Function in the dialog box for setting the overall attribute of Symbolic CAP T.



Execution list display button

Clicking this button causes an execution list to appear. This allows the user to edit, print, and transfer NC data.



Machining process list display button

Clicking this button causes a machining process list to appear.



Line drawing button

Clicking this button causes a line drawing to be produced.



Animated simulation button

Clicking this button starts animated simulation.



Redraw button

Line drawing

- The tool path is erased, and other entities are redrawn.
- Animation
- The blank is initialized.



NC data preparation start button This button is used to start NC data preparation.

III: NC data preparation pause button

Clicking this button pauses NC data preparation. To continue the preparation processing, click the NC data preparation start button or NC data preparation step execution button.



NC data preparation step execution button

Each time this button is clicked, one step of the NC data preparation processing is executed. Note, however, that each click of this button does not necessarily correspond to the output of one block of data.



NC data preparation stop button

Clicking this button terminates NC data preparation.

Setting...: NC data preparation setting window display button Clicking this button causes the window for setting the data required for NC data preparation to appear. For details, see the description of the NC data preparation setting window.

<u>C</u>lose

NC data preparation end button

Clicking this button terminates NC data preparation.

8.3 Process Order Editing Window

-	Edit Machining Process Order					
No.	Machining Type/Figure Name		T Code	Plane	Axis	Interfer.
01 02	Rough (Outer) [Area-01 Finish (Outer)]	T 01 01	Turn		
03	[Area-01 Hole (Center Drilling)]	T 02 02	Turn		
64	[ThroughDrill Hole (Drilling)]	T5 05 0	Face(F)	Y-Axis	NO
05	[ThroughDrill Hole (Chamfering)]	T5151	Face(F)	Y-Axis	NO
	[ThroughDrill	1	T5252	Face(F)	Y-Axis	NO
**	*****	***	****	*****	******	****
Edit Interference Check Move Default Ord. Select All						
Select machining process you want to output NC data OK Cancel						

The process order editing window supports the following functions:

• Changing the order in which NC data is prepared

The order in which NC data is prepared can be changed by moving processes as required. Specify the range of the processes to be moved in the list, then click the Move button. The window changes to that shown below. Position the cursor to the move destination process number (the specified processes will be moved to a point immediately above the destination process number), then click the OK button.

_	N	love Ma	chining Pr	ocess		
No.	Machining Type/Figure N	ame	T Code	Plane	Axis	Interfer
01	Rough (Outer)					
	[Area-01]	T 01 01	Turn		
03	Hole (Center Drilling)	-				
	[ThroughDrill	1	T 5 0 5 0	Face(F)	Y-Axis	NO
04	Hole (Drilling)	-				
	[ThroughDrill	1	T5151	Face(F)	Y-Axis	NO
05	Hole (Chamfering)	-				
	[ThroughDrill	1	T5252	Face(F)	Y-Axis	NO
**	*******************	*****	********	********	******	*******
Se	lect moving destination.	(Abea	d of sele	cted item)	ОК	Cance
36.	iece moving descination.	(nnca		CCCU ICENY		

• Selecting the processes for which NC data will be prepared By selecting processes from the list, NC data will be prepared only for those processes.

Clicking the Select All button selects all the processes.

• Specifying whether interference check will be performed during C/Yaxis machining

It is possible to specify whether an interference check will be performed during C/Y-axis machining. Specify the range of processes for which an interference check will be performed (or will not be performed) in the process list, then click the Yes or No button.

Interference Check		
Yes	No	

NOTE

This field is displayed only when the optional C/Y-axis module is installed.

• Initializing the order in which NC data is prepared Clicking the Default Order button cancels the results of editing performed on this window and restores the default order.

WARNING

After process sequence editing, ensure that correct machining sequence data is set. If incorrect machining sequence data is set, the tool may collide with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

8.4 NC Data Preparation Data Setting Window

Setting Data		
Working Animation		ОК
Output Mode		
Absolute		
○ Incremental		
Sequence No. Output		
🗵 🖬 at Every Process 🛛 🗖 at Every E	Block	
Initial Value	: 100	
Incremental Value(at Every Proc.)	: 100	
Incremental Value(at Every Block)	: 1	
at Intervals of	: 1	

• Working Animation

Select this item to enable animated simulation.

• Output Mode

Specify the NC data output mode.

• Sequence No. Output

Specify whether to output sequence numbers, and also specify the number output pattern.

8.5 **Animation Window**

Symbolic CAP T enables animated simulation for NC data to be performed while the NC data is being prepared. If the tool interferes with anything, it is drawn in purple. However, the color of an inner surface drawn in gradation is not changed even if it interferes with the tool. For animated simulation, the following settings can be performed:

💳 🛛 Symbolic CAP T Cutting Anima	ation
Rotation	
• 0	Setting
Speed	
• 50	

Rotation

The blank rotates. A value from 0° (side face) to 90° (end face) can be set.

Speed •

The tool movement speed can be changed. A value of between 1 (minimum) and 100 (maximum), or MAX. (only the point of each block is drawn) can be set.

Tool bar

X Redraw button

The blank is initialized.



Automatic Scale button

The display scale is automatically adjusted so that the entire blank fits into the animated simulation window.

: Pa

: Partial Enlarge button

Part of the displayed data is enlarged.

E Double Enlarge button

The display scale is enlarged by a factor of 2, with the center of the animated simulation window used as the reference.

Half Reduction button

The display scale is reduced by a factor of 2, with the center of the animated simulation window used as the reference.



Display Area Move button

The display area is moved in parallel.

F

Cutting Surface Display button

When a C/Y-axis machining surface is displayed on the animated simulation window, the previous cutting surface is displayed.

Setting...

Animated simulation data setting window display

button

Clicking this button causes the window for setting the data required for animated simulation to appear.

8.6 Data Setting Window for Animated Simulation

<u>G</u> radation			
© DN			
O OFF	Cancel		
<u>R</u> otation	<u>H</u> older		
• Full Gradation	• ON		
○ 3/4 Gradation	O OFF		

This window is used to set the drawing pattern for the blank.

• Full Graduation

Full graduation is applied. Note that the cutting status and the inner surface, as viewed from the side face, cannot be shown.

• 3/4 Graduation

Drawing is produced with one-fourth graduation omitted. During cutting, the drawing is always displayed in this state.

• Holder ON

Both the tip and tip holder are drawn.

Holder OFF

Only the tip is drawn. Drawing requires less time than when Holder ON is selected.
8.7 Writing NC Data to a Floppy Disk and Transferring NC Data to a Peripheral Device

When the execution list is displayed, and the NC data output command is selected, the file transmission tool is activated. The file transmission tool can be used to transfer data to floppy disks and peripheral devices. For details, see the description of the File Transmission Tool.

8.8 Printing NC Data and a Process Table

An execution list and process table can be printed by selecting the File => Print command from the execution list and process table.

9

TOOL PATH EDITING

This section explains how to edit tool paths you prepared.

9.1 Tool Path Editing

The tool path editing function enables editing tool paths generated on Symbolic CAP T as figure images, using a mouse. It is possible to reprepare NC data from edited tool paths.

WARNING

After tool path editing, ensure that the correct tool path data is set. If incorrect tool path data is set, the tool may collide with the workpiece and/or machine, or forced machining may occur, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

9.1.1 Tool paths subjected to editing

The following three types of tool paths can be edited.

- Tool paths generated for standard machining on Symbolic CAP T
- Tool paths generated using the machining macro execution function of Symbolic CAP T
- Tool paths generated using the user customization function of Symbolic CAP T

Tool paths for standard machining are prepared based on machining definition data during NC data preparation. If there is already a tool path, changing cutting conditions using machining definition does not re-prepare the tool path.

To enable machining definition changes for standard machining, it is necessary to delete tool paths before NC data preparation.

To use this function, enable the tool path editing function using the following commands:

Setting => Parameter Setting=> Overall Setting for Symbolic CAP T
 ... on the menu bar

(2) Use (U) for Tool Path Editing Function in the dialog box for setting the overall attribute of Symbolic CAP T.

Tool Path Editing Function			
● <u>U</u> se	O <u>N</u> ot Use		

9.1.2 Unit of editing

Tool paths are edited in machining process units. If there is a machining subprocess, however, they are edited in subprocess units.

9.1.3 Coordinate system used for tool path editing

When a tool path is edited, the machining surface on which the tool path lies is caused to face the front. The drawing coordinate system is set on the machining surface. If a machining surface is cylindrical, however, no tool path is drawn for it.

9.1.4 Editing

A tool path consists of two types of entities: Movement entity (line/arc) and auxiliary entity (POST statement). Commands related canned cycle, C-axis rotation, and feedrate are categorized as auxiliary entities. Only movement entities can be edited using the tool path editing function. They can be submitted to the following types of editing:

- Erase movement entities.
- Add movement entities.
- Trim/Extend movement entities.
- Round or chamfer a corner between movement entities.

9.2 Execution of Tool Path Editing

Clicking the button in the side menu causes the Tool Path Editing dialog box to appear. In this dialog box, you can edit tool paths. If there is no tool path, the following message appears, and the Tool Path Editing dialog box does not appear.



NOTE

Editing executed using this function cannot be undone.

9.3 Tool Path Editing Dialog Box

If there is a tool path, the following Tool Path Editing dialog box appears.

-		Tool Path Ed	liting	
No.	Edit	Kind of Machining		Machining Plane
01		Hole(Center Drilling)		
	[**]	[Area-01]	Turning
02		Rough(Face)	_	_
	Гокј	[Area-01]	Turning
03		Rough(Outer)		
	[**]	[Area-01]	Turning
	[OK]	[Area-02	Ī	Turning
04		Finish(Face)	_	_
	[NG]	[Area-01	1	Turning
05		Finish(Outer)	-	-
	[**]	[Area-01]	Turning
T.PathSelectEditDeleteElose				

9.3.1 Machining process list

Previously defined processes are listed in the Tool Path Editing dialog box. The Kind of Machining, Machining Plane, and status of editing are displayed for each machining process. The following symbols are used to indicate the status of editing, and appear in the Edit column.

- NG : There is no tool path.
- OK : There is a tool path, which has not been edited.
- ** : There is a tool path, which has been edited.

9.3.2 Selecting a tool path for editing

Clicking the **Edit** button selects a tool path for a process specified in the Tool Path Editing dialog box as a target of editing. If you do not know the process to which the desired tool path belongs, click the **Select** button to select the desired tool path in the box, and the corresponding process will be selected in the process list.

9.3.3 Deleting tool paths

Clicking the **Delete** button deletes a tool path for a process selected in the Tool Path Editing dialog box. Erasure executed using this function cannot be undone.

9.3.4 Terminating tool path editing

Clicking the **Llose** button terminates tool path editing.

9.4 Editing Movement Entities

Clicking the **Edit** button in the Edit Tool Path dialog box changes the display in the prompt area as follows:

Rough(Face) [Area-01]		Check Continuance.	ОК
	★ ↓ ↑ Start Ent. ¬ ↓ ↓ ↓ ↓	Entities List	Cancel

9.4.1 Adding movement entities

Shown below are icons used to add movement entities.



This function inserts a movement entity in such a manner that it will continue to a specified entity (start entity). If no start entity is specified before one of the icons shown above is clicked, you will be prompted to specify one. From this point on, no prompt is made to specify a start entity. The attributes, such as color, of an entity to be added are the same as those on a tool path being edited.

If an entity to be added is a line segment, the following combo box appears in the prompt area, enabling specifying a feedrate attribute.

Feedrate	
Cut Feed	Ŧ
Cut Feed	
Rapid	

If cutting feed is specified for an entity to be added, the feedrate for the entity is inherited from the previous cutting feed entity.

9.4.2 Erasing movement entities

Shown below is an icon used to erase movement entities.



This function erases movement entities.

9.4.3 Trimming/Extending movement entities

Shown below are icons used to Trim/Extend movement entities.



9.4.4 Undoing/redoing editing

Shown below are icons used to undo/redo.



This function is valid only when a movement entity is being edited. Once movement entity editing is terminated, the previous undo information is cleared, disabling the previous editing from being undone.

9.4.5 Specifying start entities

This function adds a movement entity in such a way that it will link to a start entity. To change the position where a movement entity is to be added, it is necessary to specify a new start entity. A start entity can be specified by picking an entity of a tool path on the window. An entity of a tool path on the window can be picked using the following icon. This way, a start entity is specified.



If a start entity is specified, the start position is circled as shown below.



The color of the circle enclosing a start position can be changed as follows:

- (1) Setting => Parameter Setting => Figure Operation ... on the menu bar
- (2) The color of the circle enclosing a start position is the one specified using Attribute 1 in Display Attribute for Plural Possibilities of Figure Creation in a dialog box related to figure manipulation.

Display Attributes for Plural Possibilities of Figure Creation				
Attributes 1	Attributes 2			

9.4.6 Checking tool paths for continuity

Clicking the **Check Continuance.** button enables checking a tool path being edited for continuity. If a discontinuity is found, its entities beside it are drawn in a color different from that of the tool path. If there is no discontinuity, the following message appears.



The color of the entities beside a discontinuity can be changed as follows:

- (1) Setting => Parameter Setting => Figure Operation ... on the menu bar
- (2) The color of the entities beside a discontinuity is the one specified using Attribute 1 in Display Attribute for Plural Possibilities of Figure Creation in a dialog box related to figure manipulation.

Display Attributes for Plural Possibilities of Figure Creation				
Attributes 1	Attributes 2			

button causes the following

9.4.7 **Listing entities**

Clicking the Entities List dialog box to appear. **Entities List** No. Entity Type Close 53 * Line ŧ Visible 54 ----Auxiliary 55 Line Invisible ----Auxiliary 56 57 Line <u>S</u>el. All 58 --Auxiliary 59 Line 60 Arc 61 Line

Entities List ...

In this dialog box, entities (such as auxiliary, line segment, and arc entities) forming a tool path are listed in the order in which they are arranged on the tool path. The list indicates an entity serial number, display status, and type. Whether an entity is displayed is indicated using letter * at the left of the entity.

Letter * appears The entity is being displayed. :

The entity is not being displayed. Letter * does not appears :

---Auxiliary

* Line

62 63

⊻isible

Whether to display an entity can be switched using the and

Invisible buttons. Clicking these buttons can cause an entity selected in the list to appear or disappear. This does not work for auxiliary entities, however.

Sel. All button causes all entities in the list to be selected. Clicking the <u>Close</u> Pressing the button closes the dialog box.

9.4.8 Canceling movement entity editing

Cancel cancels all editing operations performed on Pressing the movement entities. The effect of this function cannot be undone.

9.4.9 Terminating movement entity editing

Clicking the **OK** button causes a tool path to be automatically checked for continuity. If there is no discontinuity in the tool path, the Tool Path Editing dialog box is opened again. If there is one, another dialog box appears to prompt you to specify whether to make a connection at the discontinuity automatically. If an automatic connection is not specified, it is impossible to terminate the dialog box for editing movement entities.

To make a manual connection, click the **Check Continuance.** button to check for continuity previously. It is impossible to terminate the dialog box for editing movement entities as long as there is a discontinuity in the tool path.



If cutting feed is specified as a feedrate attribute for an entity, the feedrate for the entity is inherited from the previous cutting feed entity.

10 SAVING AND LOADING FILE

10.1 Saving a File

Once figures and machining definitions have been prepared, they can be saved. To do this, select the File => Save As command. Alternatively, click the Save As button on the tool bar, and specify a destination drive as

necessary.

Save As			
File <u>N</u> ame: fanuc04.pf	<u>D</u> irectories: f:\capt	ОК	
fanuc01.pf fanuc02.pf fanuc03.pf fanuc04.pf	★ Capt	2 Cancel	
Save File as <u>T</u> ype:	Drives:	*	
CAP T Files(*.pf)	➡ f: ms-dos_6	<u>+</u>	

10.2 Loading a File

A saved file can be loaded. To do this, select the File \Rightarrow Open command. Alternatively, click the File Open button on the tool bar, and select the required file.

		Open		
File <u>N</u> ame: fanuc01.pf		<u>D</u> irectories: f:\capt		ОК
fanuc01.pf fanuc02.pf fanuc03.pf fanuc04.pf	+	<mark>≥</mark> f:\ ► capt	+	Cancel
List Files of <u>Type:</u> CAP T Files(*.pf)	Ŧ	Dri <u>v</u> es: E f: ms-dos_6	Ŧ	

CREATING NEW PROGRAM

When Symbolic CAP T is activated, a new file is created automatically. This new file contains no blank figure, part figure, or machining definition data. The file is assigned a temporary name (untitled). Thus, when creating a new program, the user can start programming immediately, using this file.

To create a new program at any other time, select File => New command from the menu bar. To create machining definition data using only existing blank figure and part figure data, select File => Clear Machining Data.

When the contents of a program to be created closely approximate those of an existing file, the user can read the original file, then assign it a new name and store it under that name.

NOTE

When New or Clear Machining Data is executed, the file being manipulated at that instant is discarded. If the program being manipulated has not yet been saved, a prompt appears asking the user to specify whether the program should be stored before initialization.

12 FIGURE CREATION

12.1 Types of Figures

This section explains the types of figures that can be handled.

(1) Figure entities

The smallest graphics unit is called a figure entity. Figure entities include:

- Point
- Straight line
- Circle
- Arc
- Polyline

An open figure entity consisting of multiple connected arcs and straight lines with both ends of the entity not connected.

Polygon

A closed figure entity consisting of multiple connected arcs and straight lines.

• Fitting curve

A figure entity formed by connecting a series of points with arcs and straight lines smoothly.

(2) Group

A group consists of multiple figure entities. At present, only a point group is defined as a group.

12.2 Specifying a Radius, Distance, and Angle

Enter values such as a radius, distance, and angle from the keyboard. An angle is positive in the counterclockwise direction and negative in the clockwise direction, relative to the positive Z-axis.

12.3 Specifying Positions

When figures are created, modified, and edited, the positions of figure entities, the positions of figure intersections, or positions in a plane may have to be selected. Before specifying positions, select the position specification mode by using the following buttons:



(1) Specifying a coordinate system and dimensions

(a) Coordinate system and dimension

Specify one coordinate system and dimension from among the 2dimensional machining plane coordinate system (2D), 3dimensional machining plane coordinate system (3D), and 3dimensional world coordinate system.

- 2D: Z and X coordinates are specified in the machining plane coordinate system.
- 3D: X, Y, and Z coordinates are specified in the machining plane coordinate system.
- 3DW: X, Y, and Z coordinates are specified in the world coordinate system.

For details of the coordinate systems, see Section Coordinate system.

(2) Coordinate specification method

(b) Cartesian coordinates

Specify a position by inputting the X and Y coordinates.

(c) Polar coordinates (reference point is fixed)

Specify a position by means of the distance from the fixed reference point and the angle, which indicates the direction.

(d) Polar coordinates (reference point is updated automatically) Specify a position by means of the distance from the position input immediately before and the angle, which indicates the direction.

(e) Origin of the polar coordinate system

Set the origin of the polar coordinates system.

(3) Specifying a location with the mouse

(f) Start point or end point of an entity

Click on a point near the start point or end point of an entity to select the start point or end point.

(g) Intersection of a figure

Click on a point near an intersection to select it.

(h) Intersection of two figure entities

Click on a point near each of two figure entities to select the intersection of the two figure entities. The intersection of the extended lines of two entities can also be selected.

(i) Midpoint of an entity

Click on a point near an entity to select its middle point. The middle point is determined as follows:

- Line segment: Midpoint of the line segment
- Circle: Center of the circle
- Arc: Middle point on the circumference of the arc
- Others: Middle point of one of the entities constituting the target entity

(j) Center of a circle or arc

Click on a point near a circle or arc to select the center of the circle or arc.

(k) Already entered point or point group

Click on a point near an already entered point or point group to select it.

(l) Grid display point

Click on a point near a grid display point to select it.

(m) Position of the mouse pointer in the window

Click on a desired point in the drawing window to select the mouse pointer position.

12.4 Selecting Entities

To select an entity, click the corresponding position.

When an entity is selected, figure search processing is performed in figure type order (point to straight line to circle to arc ...). When entities are close together, a desired entity sometimes cannot be selected. In such a case, the figure search direction switch button displayed in the status window can be clicked to reverse the search order.

To select more than one entity, the user can select them one-by-one, by clicking them with the mouse. Alternatively, the user can specify a rectangle and select all the entities in that rectangle. Or, a particular entity in the rectangle can be excluded.

The method of selecting entities can be set using the entity select mode, as follows.



(a) Select/Deselect

Whether an entity is to be selected or deselected is indicated.

(b) Entities inside a rectangle

All entities inside a rectangle are selected. select entities by dragging diagonal corners of the rectangle to indicate the range of selection.

(c) Entities inside a rectangle and entities touching the rectangle frame

All entities inside a rectangle and all entities that touch the rectangle frame are selected. Select entities by dragging diagonal corners of the rectangle to indicate the range of selection.

(d) Entities outside a rectangle

All figures that are located entities outside a rectangle are selected. Select elements by dragging diagonal corners of the rectangle to indicate the range of selection.

(e) Entities outside a rectangle and entities touching the rectangle frame

All entities that are located outside a rectangle and entities that touch the rectangle frame are selected. Select elements by dragging diagonal corners of the rectangle to indicate the range of selection.

(f) All entities displayed in a window

All entities displayed in a window are selected. Click on an arbitrary point in a target window to select all elements in the window.

(g) All entities displayed in a window

All the entities displayed in a window are selected. Click any point within the window.

12.5 Coordinate System

This section explains two coordinate systems: the world coordinate system and machining plane coordinate system.

12.5.1 World coordinate system

Created figures and tool paths are stored in a dedicated three-dimensional database. These data items are managed using a unique coordinate system which is used as a base coordinate system to indicate all positions. This absolute coordinate system is called the world coordinate system. Normally, parts or other structures have one point that is used as a reference point for the entire structure. Placing this reference point at the origin of the world coordinate system streamlines figure creation and other processing.

12.5.2 Machining plane coordinate system

A machining plane coordinate system tailored to a drawing format is provided to facilitate coordinate input. The machining plane coordinate system is automatically determined according to the drawing format.

12.6 Modifying Figures by Extension/Trimming

A figure can be modified by trimming unnecessary parts of figure entities such as unnecessary draft lines and by extending parts of figure entities to specified position.



Using the above figures as examples, modifying figures by selecting Modify=>Trim/Extend from the menu is described below. The operations for extension and trimming are the same in the examples below.

(a) Extending/trimming a figure entities up to an intersection

- 1) Select Modify => Trim/Extend => Intersection from the menu.
- 2) Select point 1 and point 2 in this order.

(b) Extending/trimming a figure entities up to a nearby point

- 1) Select Modify => Trim/Extend => Nearby Point from the menu.
- 2) Select point 1 and point 2 in this order.
- (c) Extending/trimming two figure entities up to the intersection of the entities
 - 1) Select Modify => Trim/Extend => Both Entities from the menu.
 - 2) Select point 1 and point 2 in this order.

(d) Extending/trimming both ends of figure entities

- 1) Select Modify => Trim/Extend => Both Ends from the menu
- 2) Select point 1, point 2, and point 3 in this order.

12.7 Copying Figures

A figure can be copied in many different ways.

(a) Parallel-Positions

A figure can be copied to specified multiple positions in parallel mode.

- 1) Select Edit => Copy => Parallel-Positions from the menu.
- 2) Select point 1 to specify the figure to be copied, then select the End button.
- 3) Select point 1 to specify the reference point of the figure to be copied.
- 4) Select point 2 and point 3 as copy destinations, then select the End button.
- 5) Press the ESC button.



(b) Parallel-Distance

A figure can be copied to the multiple positions at uniformly spaced intervals in parallel mode.

- 1) Select Edit => Copy => Parallel-Distance from the menu.
- 2) Select point 1 to specify the figure to be copied, then select the End button.
- Enter 15 and -15 to specify space interval values DZ and DY, respectively.
- 4) Enter 2 to specify the number of copy destinations.
- 5) Press the ESC button.



(c) Parallel-Division No.

A figure can be copied in parallel mode to the multiple positions determined by equally dividing a specified maximum copy distance.

- 1) Select Edit => Copy => Parallel-Division No. from the menu
- 2) Select point 1 to specify the figure to be copied, then select the End button.
- Enter 30 and -30 to specify maximum copy distance values DZ and DX, respectively.
- 4) Enter 2 to specify the number of copy destinations.
- 5) Press the ESC button.



(d) Parallel-Position, Division No.

A figure can be copied in parallel mode to multiple positions determined by equally dividing the distance to a specified final copy destination.

- Select Edit => Copy => Parallel-Position, Division No. from the menu.
- 2) Select point 1 to specify the figure to be copied, then select the End button.
- 3) Select point 1 to specify the reference point of the figure to be copied.
- 4) Select point 2 to specify the final copy destination.
- 5) Enter 2 to specify the number of copy destinations.
- 6) Press the ESC button.



(e) Rotate-Distance

A figure can be copied to positions determined by rotating the figure repeatedly by the same angle.

- 1) Select Edit => Copy => Rotate-Distance from the menu.
- 2) Select point 1 to specify the figure to be copied, then select the End button.
- 3) Select point 2 to specify the center of rotation.
- 4) Enter 60 to specify the angle of rotation.
- 5) Enter 2 to specify the number of copy destinations.
- 6) Press the ESC button.



(f) Rotate-Division No.

A figure can be copied to positions determined by equally dividing a specified total angle of rotation.

- 1) Select Edit => Copy => Rotate-Division No. from the menu.
- 2) Select point 1 to specify the figure to be copied, then select the End button.
- 3) Select point 2 to specify the center of rotation.
- 4) Enter 120 as the total angle of rotation.
- 5) Enter 2 to specify the number of copy destinations.
- 6) Press the ESC button.



(g) Mirror-Plane

The mirror image of a figure can be produced by specifying two points to define a mirror plane perpendicular to the machining plane.

- 1) Select Edit => Copy => Mirror-Plane from the menu.
- 2) Select point 1 to specify the figure to be copied, then select the end button.
- 3) Select point 2 and point 3 to specify the mirror plane.
- 4) Select the ESC button.



(h) Mirror-Position

The mirror images of a figure can be produced by defining a reference point and copy destination to specify a mirror plane perpendicular to the machining plane.

- 1) Select Edit => Copy => Mirror-Position from the menu.
- 2) Select point 1 to specify the figure to be copied, then select the End button.
- 3) Select point 1 to specify the reference point of the figure to be copied.
- 4) Select point 2 to specify the copy destination.
- 5) Select the ESC button.



(i) Arbitrary

A figure can be copied by specifying a copy source reference point and the directions of two associated orthogonal axes and a copy destination reference point and the directions of two associated orthogonal axes.

- 1) Select Edit => Copy => Arbitrary from the menu.
- Select point 1 to specify the figure to be copied, then select the End button.
- 3) Select point 1 to specify the reference point of the figure to be copied.
- 4) Select point 3 and point 4 to define the directions of the first axis and second axis.

- 5) Select point 5 to specify the reference point of the copy destination.
- 6) Select point 6 and 7 to define the directions of the first axis and second axis.
- 7) Press the ESC button



12.8 Creating Polygons



A polygon can be created by combining multiple entities.

This section explains how to create the polygon in (a) above from the figure in (b) above.

- 1) Create figure in (b) with a rectangle and two lines.
- 2) Select the icon for polygon from the side menu.
- 3) Select the icon for Trace from the side menu.
- 4) Select start point P1.
- 5) Select the polygon entities P2, P3, P4, P5, P6, P7, and P2 in this order.
- 6) Select Edit => Erase from the menu.
- 7) Select **P8**, **P9**, and **P10**. Then, select the End button.
- 8) Select View => Redraw from the menu.

12.9 Fitting Curve

12.9.1 Creation method of a fitting curve

A smooth curve that passes through a series of points is called a fitting curve. A fitting curve consists of a series of tangent arcs. When three consecutive points lie on the same straight line, they can be connected by a straight line. Generally, one section (between two adjacent points) is connected by two arcs. When there is an inflection point within a section, the section is connected with four arcs.



12.9.2 Adjusting the creation method of a fitting curve

Normally, the method of creating a fitting curve need not be adjusted. When necessary, however, the method of creating a fitting curve can be adjusted by changing the three parameters explained below. To adjust the creation method, modify the Fig_FcvCvdg parameter registered in the initialization file (the standard initialization file is CAPT.INI) in the same directory as that in which Windows is installed, then restart the system. By changing this parameter setting, the window for setting the parameters is displayed each time a fitting curve is created:

Fig_FcvCvdg=FALSE => Fig_FcvCvdg=TRUE

(1) Dumping constant

The dumping constant parameter is used for adjusting a section which has no inflection point and which is connected by two arcs.

When the arcs for such a section are obtained, a tangent vector is obtained at each end point first. Then, an arc tangent to each tangent vector is obtained. The fitting curve can be adjusted by using the dumping constant to correct the tangent vector at the end point, as shown in the figure below.

When the dumping constant is set to 100, no adjustment is made.

When the dumping constant is set to a value other than 100, a different fitting curve will be obtained from the same series of points, depending on whether the creation order is reversed. Therefore, the dumping constant should be used carefully.

A value between 0 and 100 can be set as the dumping constant. The initial value is 100.



(2) Movement rate of inflection point

The movement rate of inflection point parameter is used to adjust the position of the inflection point when a section contains an inflection point.

An inflection point is the intersection of the inflection point vector and the perpendicular bisector of the straight line connecting the start and end points.

As the value of this parameter is reduced, the position of the inflection point approaches nearer to the midpoint of the start and end points.

The inflection point vector is obtained as follows:



- A1: Angle between the start point tangent vector and the straight line connecting the start and end points
- A2: Angle between the end point tangent vector and the straight line connecting the start and end points
- Ap: Angle between the inflection point vector and the straight line connecting the start and end points

$$Ap = \frac{A1 - A2}{2} \times (Movement rate of inflection point) / 100$$

A value between 0 and 100 can be set. The initial value is 75.

(3) Tolerance for section judgment

This parameter is used to determine a straight line section. When the angle formed by three consecutive points is equal to or less than the value specified for this parameter, the sections corresponding to those three points are recognized as being a straight line. The units are radians. The formula for converting radians to degrees is as follows.

Angle (degrees) =
$$\frac{180}{\pi}$$
 × Angle (radians)

12.10 Offset

A figure can be created by shifting a polyline/polygon, either entirely or partly, through a specified distance. This function is called the offset function. To execute this function, select Edit => Offset, or Edit => Offset of part. The following sections explain the use of the offset function to shift a corner both out and in.

12.10.1 Outward offset for a corner

When outward offset is applied to a corner, the corner entities will be separated from each other. Each entity must be extended in order to reconnect them. This processing is called corner interpolation. There are four modes of corner interpolation. The desired mode is selected by responding to the Target corner prompt which appears in the prompt area when Edit => Offset or Edit => Offset of part is selected.

(1) Segment/OFF

When this mode is selected, entities are extended by a straight line to connect them. When the original entity is a straight line, it is simply extended directly. When the original entity is an arc, a straight line tangent to the arc is extended.



(2) Segment/Tolerance

In this mode, the Length to Chamfering position, set when Setting => Parameter Setting => Figure Operation is selected, is referenced.

Basically, the operation performed in this mode is almost the same as that performed in Segment/OFF mode. In this mode, however, when the distance from the vertex of the original corner to the vertex of the new corner obtained by the offset operation exceeds the value set for Length to Chamfering position, each entity is extended with a straight line by an amount equal to the offset. Then, both ends are connected with a straight line. This operation is called chamfering.



(3) Segment/Auto

In this mode, the corner angle is referenced.

When the corner angle is smaller than 90, chamfering is performed in the same way as explained above.

When the corner angle is equal to or greater than 90, the operation performed is the same as that performed in Segment/OFF mode.

(4) Arc

In this mode, the entities are connected by an arc having a radius equal to the specified offset.



12.10.2 Inward offset for a corner

When inward offset is performed for a corner, the figure entities obtained by performing offset may cross other entities, or the positional relationship between entities may differ from the original figure. Such conditions are referred to as interference.


The operation performed in the case of interference is determined according to the setting of the When Interfering item, which appears when Setting => Parameter Setting => Figure Operation is selected.

If The interfered element is not removed. is selected, the offset figure is created with the interference left as is.

If The interfered element is removed, is selected, the interference is first removed, then the figure is created. If removing the interference causes the figure to disappear, as in the case of Example 1 above, an error occurs.

By default, The interfered element is removed. is set.



13 CUTTING CONDITION DATA

This section describes how to edit the cutting condition data that is used when automatically determining the machining conditions during machining definition.

13.1 Overview

Symbolic CAP T allows cutting speed, feedrate, and other cutting condition data to be classified according to the workpiece material, such that it can be registered in a cutting condition library. Cutting condition data can be registered as combinations of machining type, tool type, and tool material for a given workpiece material.

Multiple materials can be registered. When a range of materials is registered, the material selected for a blank figure can be referenced and used in machining definition.

In machining definition, the cutting condition data corresponding to a desired machining type, as well as the type and material of the tool to be used are determined from the cutting condition data registered for the current workpiece material. Based on this data, the spindle speed, feedrate, and other data are determined automatically. As part of machining definition, the cutting condition library can be updated by feeding back the cutting condition data to the library.



NOTE

1. Searching for a tool to be used

For both manual and automatic process determination for machining definition, the tooling data is searched for a tool of a type that can be used for the machining type specified in the cutting condition data for the current workpiece material. An appropriate tool is selected as the tool to be used.

When there are two or more tools that are judged as being suitable for the machining type, a search of the data is made in the order in which the tool types are registered.

2. Cutting conditions and machining conditions

In this chapter, the cutting speed, feed amount, and other data that can be specified for each workpiece material are referred to as the cutting conditions. The spindle speed, feedrate, and other data that are obtained from the cutting conditions are referred to as machining conditions.

WARNING

Before creating a machining program using the conversational function, ensure that correct cutting condition data is set. Failure to set correct cutting condition data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

13.2 Functions

The following functions can be used to edit the cutting conditions:

• Adding or deleting a workpiece material name

This function adds or deletes the name of the material of the workpiece to be machined. The material registered using this function can be selected on the blank figure window.

• Adding modifying, or deleting the cutting conditions

This function adds, modifies, or deletes the cutting condition data corresponding to the combination of machining type, tool type, and tool material specified for a workpiece material.

The following cutting condition data can be set.

- (1) Data set for a machining type
 - Finishing allowance
- (2) Data set for a machining type, tool type, or tool material
 - Coolant type
 - Cutting speed
 - Feedrate or feed amount for a single tooth or for all teeth
 - Cutting depth
 - Activation of the cutting condition data utility

13.3 Starting of Editing Cutting Condition Data

The function to edit cutting conditions is started by selecting it from the menu bar.

From the menu bar, select Setting => Library Data Setting => Cutting Condition.

Edit Cutting Condition Data window appears, as shown below:

Edit Cutting Condition Data					×		
	Material	Name	NC M	achine	Name		
AL S4 FC	5C 25					<u>C</u> lose	
						Utility	
	<u>A</u> dd	Modify	<u>D</u> elet	e			

13.4 Adding a Material Name

Click the Addition button on the Edit Cutting Condition Data window. Add Material Name dialog appears, as shown below:

Add Material Name	Add Material Name 🔀				
© New Making of Cutting Condition File					
Material Name :		ОК			
NC Machine :		Cancel			
File Name :					
(No extension,	Max. 8 characters)				
CReading of Cutti	ng Condition File				
Directory :					
	<u>L</u> ist Display				
Material Name	NC Machine Name				

A cutting condition file must be placed in the directory specified in Setting of Environment dialog of the Setting menu.

The cutting condition file can be added in either of the following two ways:

- Creating a new cutting condition file
 - A cutting condition file is created and placed in the directory specified on Setting of Environment dialog of the Setting menu.
- Reading an existing cutting condition file
 An existing cutting condition file is copied into the directory specified in Setting of Environment dialog of the Setting menu.

NOTE

Cutting condition file and cutting condition data In this chapter, a file containing cutting conditions is referred to as a cutting condition file. The cutting condition data includes the contents of the cutting condition file and information registered for the system.

(1) Creating a new cutting condition file

On the Add Material Name window, select the New Making of Cutting Condition File option. Specify a workpiece material name, NC machine name, and file name. Then, click the OK button.

The NC machine name is merely a comment and can be omitted. The window for selecting the machining type to be added as a cutting condition is displayed. For an explanation of adding the cutting conditions, see the section describing the editing of cutting condition data.

(2) Reading an existing cutting condition file

On the Add Material Name window, select the Reading of Cutting Condition File option. Specify the directory containing the cutting condition file, then click the List Display button. Select a target cutting condition file from the list, then click the OK button. The Addition of Cutting Condition Data window remains displayed. The specified cutting condition file can be read. To return to the Edit Cutting Condition Data window, click the Cancel button.

13.5 Deleting a Material Name

On the Edit Cutting Condition Data window, select the material to be deleted, then click the Delete button. This operation cannot be used to delete the cutting condition data for a material referenced in the machining definition.

The window for deleting material name appears, as shown below:

Delete Material Name	×		
• Delete from Material List			
C Delete Cutting Condition File			
Cutting Condition Data			
Material Name : S45C			
NC Machine :			
Delete Ok ?			
OK Cancel			

The material name can be deleted in either of the following two ways:

• Deleting the material name from the material list only

The material name is deleted from the material list such that it cannot be referenced in machining definition. The corresponding cutting condition file is not deleted from the cutting condition file directory specified in the Setting of Environment.

• Deleting both the material name and the cutting condition file The material name is deleted from the material list. The corresponding cutting condition file is also deleted from the cutting condition file directory specified in the Setting of Environment. To delete a material name, select the Delete from Material List option or Delete Cutting Condition File option, then click the OK button.

13.6 Editing Cutting Condition Data

On the Edit Cutting Condition Data window, select the material for which the corresponding data is to be edited, then click the Modify button. The Machining Type List window appears, as shown below:

-	Machining	g Type List				
Material	Material Name : AL					
NC Machir	NC Machine :					
Mac	hining Name	Data Number				
Hole	Center drilling	1 1	F I			
Hole	Drilling	1				
Hole	Tapping	1				
Hole	Reaming	1				
Turning	Outer Rough	3				
Turning	Inner Rough	3				
Turning	Facing Rough	3				
Turning	Outer Semi Finish	3				
Turning	Inner Semi Finish	3				
Turning	Facing Semi Finish	3 •	F			
Modify	<u>D</u> elete					

Clicking the OK button after editing the cutting condition data makes the edited data effective.

Clicking the Cancel button after editing the data discards all changes that have been made.

13.6.1 Modifying cutting condition data on a machining type basis

To modify the cutting condition data, select a target cutting condition on the Machining Type List window, then click the Modify button.

The Tool Type List window appears, as shown below:

		Tool Type List		
Material Name : Machining Type : Finishing Allowanc X Axis Dir. :	AL Turning e 0.200 Z Axis	Nuter Rough	C Machine :	OK
Tool Typ	pe T	ool Material		
General Straight Button	Cai Cai Cai	rbide rbide rbide		<u>B</u> ack <u>N</u> ext
Insert	<u>M</u> odify	<u>D</u> elete		<u>C</u> hange Order

• Finishing Allowance

Specify the finishing allowance to be referenced during automatic process determination.

A finishing allowance can be specified for the following machining types:

- Rough turning (outer, inner, face)
- Turning semi-finish (outer, inner, face)
- Rough grooving (outer, inner, face)

Tool type list

A tool type that appears in the tool type list can be used for the selected machining type. In manual or automatic process determination of the machining definition, the tooling data is searched for a tool of the tool type registered in this list.

If two or more tool types are registered, data corresponding to those tool types is searched in the order in which the types appear in the list.

Back button

Clicking this button causes the tool type list for the machining type preceding the currently selected type to be displayed.

Next button

Clicking this button causes the tool type list for the machining type following the currently selected type to be displayed.

13.6.2 Adding cutting condition data

To add cutting condition data, select the tool type at a desired insertion position on the Tool Type List window, then click the Insert button. The window for adding a tool type appears, as shown below:

Addition of Tool Type			
Tool Type	Tool Material		
General Straight Button Multiple	Carbide	OK Cancel	
Select Tool Type and Tool Material			

Tool Type

The kind of tool types that can be added for a given machining type is determined by the system. This kind cannot be changed.

• Tool Material

To change the tool material of a tool type , click Setting => Library Data Setting => Tool, Tooling.

Select the tool type or tool material of the cutting condition data to be added, then click the OK button.

The window for adding and modifying the cutting conditions data for the selected machining type appears.

Modify Cutting Con	dition Data				×	
Machining Typ	Machining Tupe : Hole - Center drilling					
Tool Type :	Center	Drill	Mater	rial: SKH9)	
Coolant / Cuttin	ng Speed / Fe	ed Rate				
Coolant Type	Not lise	-				
	Hot Osc				Cancel	
Tool Dia Cu	tting Speed -	Feed Amo	ount			
Tool D	ia. – Cutt:	ing Spee	d - Feed Amount	Add		
1	60.		0.03			
3.	60.		0.03 0.06	<u>M</u> odify		
6	60.		0.1			
10	60.		0.1	Delete		

The following data can be specified. For some data items, two units are specified: The first is for metric specification while the second is for inch specification.

• Coolant Type

Select one of the following three coolant type settings:

- Not used
- Oil
- Mist
- Cutting Speed

The following data can be specified for each machining type, in units of meters per minute or feet per minute.

• Hole machining

Cutting speed for tool diameter

Other machining

Cutting speed for turning

• Feed Amount

•

The following data can be specified for each machining type:

• Hole machining (center drilling)

Specify the feed amount per revolution, corresponding to the tool diameter, as the feed amount for all teeth, in units of

millimeters per revolution or inches per revolution.

• Hole machining (drilling)

Specify the feed amount per revolution, corresponding to the tool diameter, as the feed amount for all teeth, in units of millimeters per revolution or inches per revolution. From this value, the cutting feedrate is calculated.

- Specify the feedrates at start, at end, at return, and at repeated cut-in as a percentage of the cutting feedrate.
- Hole machining (reaming)

Specify the feed amount per revolution, corresponding to the tool diameter, as the feed amount per tooth, in units of millimeters per revolution or inches per revolution. From this value, the cutting feedrate is calculated.

Specify also the feedrate at return as a percentage of the cutting feedrate.

• Other machining

Specify the feed amount per revolution for a machining type, in units of millimeters per revolution or inches per revolution.





NOTE

Calculating the cutting speed and feed amount corresponding to the tool diameter

If the cutting speed or feed amount corresponding to the tool diameter is registered in the cutting condition data, the registered cutting speed or feed amount is used.

If not, the data is determined by linear interpolation from the registered tool diameter and the cutting speed or feed amount, as shown below:



• Cutting depth, cutting ratio

The following data can be specified for a machining type:

• Hole machining (drilling)

Specify the change ratio (%) and cramp amount (maximum or minimum value, in units of millimeters or inches) for the pecking amount in drilling.

Specify also the ratio of the hole depth to the diameter so that a drilling method can be determined.

Example

Ratio of hole depth to diameter (depth/diameter) for drilling method 1 = A (%)

Ratio of hole depth to diameter (depth/diameter) for drilling method 2 = B (%)

Ratio of hole depth to diameter of hole to be machined = R

Under the above conditions, the drilling method is determined as follows:

R < A	:	Drilling cycle
$A \leqq R < B$:	High-speed deep drilling cycle
$B \leqq R$:	Deep drilling cycle

• Rough turning (outer, inner, face) Specify the cutting depth (in units of millimeters or inches) in the direction of the diameter, the change ratio (%) for the cutting depth, and the cramp value (maximum or minimum value, in

units of millimeters or inches) for the cutting depth.

• Rough grooving (outer, inner, face)

Specify the change ratio (%) for the cutting depth in the direction of the groove depth and the cramp value (maximum or minimum value, in units of millimeters or inches) for the cutting depth.

Specify also the cutting depth in the direction of the groove width as a ratio to the available chip width (%) and the first cutting depth in the direction of the groove depth by a ratio to the chip width (%).



Threading (outer, inner, front) Specify the first cutting depth and cramp value for the cutting depth (maximum or minimum value, in units of millimeters or inches).

Back button

When two or more tool types are registered, clicking this button causes the cutting conditions for the tool type preceding the currently selected tool type to appear.

Next button

When two or more tool types are registered, clicking this button causes the

cutting conditions for the tool type following the currently selected tool type to appear.

Specify the desired cutting condition data, then click the OK button. The cutting conditions are registered.

13.6.3 Modifying cutting condition data

To modify previously registered cutting condition data, select the target tool type on the Tool Type List window, then click the Modify button.

The window for registering and modifying the cutting conditions for the selected machining type is displayed.

Modify the cutting condition data, then click the OK button. The cutting conditions are modified.

13.6.4 Deleting cutting condition data

To delete registered cutting condition data, select a target tool type on the Tool Type List window, then click the Delete button.

A prompt window asking the user to confirm the deletion appears. To delete the data, click the Yes button.

Two or more cutting conditions can be deleted at one time.

13.6.5 Changing the order of the tool types

When two or more tool types are registered for a machining type, the order of those tool types can be changed.

To change the order, select a target tool type on the Tool Type List window, then click the Change Order button.

At the bottom of the conversational window, the following message appears:

Select Position of Top Reversed List after Changed Order.

Select a new position from among those that are not displayed in reverse video. The order is changed.

The positions of two or more tool types can be changed at one time.

13.6.6 Deleting cutting condition data on a machining type basis

To delete a cutting condition on a machining-type basis, first select a target machining type on the Machining Type List window, then click the Erase button.

A prompt window asking the user to confirm the deletion appears. To delete the data, click the Yes button.

The cutting condition data for two or more machining types can be deleted at one time.

13.7 Calculating Method of the Machining Condition Data

- Spindle speed (S)
 - Hole machining

 $S = (1000 \times V)/(\pi \times D) \dots$ Metric machine

 $S = (12 \times V)/(\pi \times D) \dots$ Inch machine

V: Cutting speed

- D: Tool diameter
- Other machining

S = (Smax + Smin)/2

Smax = $(1000 \times V)/(\pi \times Dmax)$ Smin = $(1000 \times V)/(\pi \times Dmin)$... Metric machine

Smax = $(12 \times V)/(\pi \times Dmax)$

Smin = $(12 \times V)/(\pi \times Dmin)$... Inch machine

V: Cutting speed

Dmax: Maximum workpiece diameter in the corresponding tool path output

- Dmin: Minimum workpiece diameter in the corresponding tool path output
- Feedrate (F)
 - Hole machining (center drilling) Feedrate for cutting

 $\mathbf{F} = \mathbf{f}$

f: Feed amount for all teeth

• Hole machining (drilling) Feedrate for cutting

F = f

Feedrate at start, at end, at return, and at repeated cut-in Fn = F $\times\,\alpha$ /100

 α : Compensation ratio (%)

• Hole machining (tapping) Feedrate for cutting

F = P

= **r**

P: Pitch

• Hole machining (reaming)

Feedrate for cutting

 $F = f \times n$

- f: Feed amount per tooth
- n: Number of teeth

Feedrate for return

 $Fn=F\times\alpha\,/100$

 α : Compensation ratio (%)

• Other machining

The feed amount per revolution specified for the corresponding machining type is used as is.

- Cutting depth (C)
 - Drilling

Primary drilling depth (D1, first cutting depth) and secondary drilling depth (D2) of a high-speed deep drilling cycle or deep drilling cycle are calculated as follows:

- D1 = ((Ratio of hole depth to diameter for drilling method1)/100) × (Tool diameter)
- D2 = ((Ratio of hole depth to diameter for drilling method2)/100) × (Tool diameter)





- Rough turning, threading The cutting depth specified as part of the cutting condition data is used as is.
- Rough grooving

(First) cutting depth in the Z-axis direction for outer or inner machining, or in the X-axis direction for face machining (D1)

 $D1 = ((Ratio to available chip width)/100) \times (Available chip width)$

(First) cutting depth in the X-axis direction for outer or inner machining, or in the Z-axis direction for face machining (D2)

 $D2 = ((Ratio to chip width)/100) \times (Chip width)$

For all machining types, if the cutting depth varies in a cutting cycle, the cutting depth is calculated as follows:

$C=Cn\times\alpha\,/100$

Cn: Previous cutting depth

α: Change ratio for cutting depth(%)

14 CUTTING CONDITION DATA UTILITY

This section describes the cutting condition data utility used to input/output cutting condition files.

14.1 Functions

The cutting condition data utility has the following functions.

• Output of cutting condition file

A cutting condition file is output from a cutting condition file directory, specified using Setting of Environment, to a specified directory.

• Input of cutting condition file

A cutting condition file is input from a specified directory to a cutting condition file directory, specified using Setting of Environment.

• Change of cutting condition file

The workpiece material, target NC machine, and file names related to a cutting condition file in a specified directory are changed. This function can also be used as a copy tool by preserving the original cutting condition file.

• Uniting of cutting condition file

Two cutting condition files in a specified directory can be united into a single cutting condition file.

14.2 Starting the Cutting Condition Data Utility

The cutting condition data utility is started from cutting condition registration.

To start the utility, select Setting => Library Data Setting => Cutting Condition from the menu bar.

The following Edit Cutting Condition Data dialog box appears.

Edit Cutting Condition Data 🛛 🛛 🗙			
Material Name NC Machine Name			
AL S45C FC25	<u>C</u> lose		
Add Modify Delete	<u>U</u> tility		

Click the Utility button. Cutting condition data utility dialog box appears.

14.3 Selecting Utility Functions

Cutting condition data utility
Qutput
Input
C <u>h</u> ange
<u>U</u> niting

Clicking a function button displays the dialog box that corresponds to the selected function.

Clicking the Close button returns you to Cutting Condition Data dialog box.

14.4 Outputting Cutting Condition Files

Clicking the Output button in Cutting condition data utility dialog box displays the following dialog box, used to specify cutting condition file output.

💳 Outpu	it of cutting condition file
Output origin Directory: G:\SYMCAPT\PCFAPT.E	Output destination Directory:
Work Material Name(NC Machine Name) S45C AL FC25	Vork Material Name(NC Machine Name)
<pre>*:System unregistration <u>A</u>II</pre>	<pre>*:Unregisters in managment file </pre> <u>Execution</u>

- Output origin
 - Cutting condition file list

This is a list of the cutting condition files in a cutting condition file directory, specified as the output origin using Setting of Environment.

If a cutting condition file name in the cutting condition file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the corresponding cutting condition file exists in the directory.

Cutting condition files, selected from the cutting condition file list, are output.

• All/Nothing button

This button either selects or deselects all the cutting condition files in the cutting condition file list. Clicking the button toggles between selection and deselection of all the cutting condition files.

- Output destination
 - Directory

This box is used to specify a directory to which cutting condition files are to be output.

 Cutting condition file list Clicking the List button displays a list of cutting condition files in a specified directory. If a cutting condition file name in the cutting condition file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the corresponding condition file exists in the directory.

• Execution button

Cutting condition files are output by selecting them from the output origin list, specifying the output destination directory, then clicking the Execution button.

The cutting condition file list at the output destination is updated. Cutting condition file output can continue after the update.

It is possible to output more than one cutting condition file at a time.

Close button

Clicking the Close button returns you to Cutting condition data utility dialog box.

14.5 Inputting Cutting Condition Files

Clicking the Input button in Cutting condition data utility dialog box displays the following dialog box, used to specify cutting condition file input.

-	Input of cutting condition file				
ſ	Input destination	Input origin			
	Directory: G:\SYMCAPT\PCFAPT.E	Directory:			
		L <u>i</u> st			
	Work Material Name(NC Machine Name)	Work Material Name(NC Machine Name)			
	S45C AL				
	FC25				
	*:System unregistration	*:Unregisters in managment file			
ſ	Execution				

- Input destination
 - Cutting condition file list

This is a list of cutting condition files in a cutting condition file directory, specified as an input destination using Setting of Environment.

If a cutting condition file name in the cutting condition file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the cutting condition file exists in the directory.

- Input origin
 - Directory

This box is used to specify a directory from which cutting condition files are to be supplied.

• Cutting condition file list

Clicking the List button displays a list of cutting condition files in a specified directory.

If a cutting condition file name in the cutting condition file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the cutting condition file exists in the directory.

Cutting condition files, selected from the cutting condition file list, are input.

All/Nothing button

This button either selects or deselects all the cutting condition

-184-

files in the cutting condition list. Clicking the button toggles between selection and deselection of all the cutting condition files.

Execution button

Cutting condition files are input by selecting them from the input origin list and clicking the Execution button.

The cutting condition file list at the input destination is updated. Cutting condition file input can continue after the update.

It is possible to input more than one cutting condition file at a time.

Close button

Clicking the Close button returns you to Cutting condition data utility selection dialog box.

14.6 Changing Cutting Condition Files

Clicking the Change button in Cutting condition data utility dialog box displays the following dialog box, used for changing the cutting condition files.

Change of cutting condition file	
Directory: D:\CAPT.E List Work Material Name(NC Machine Name) S45C AL FC25	Close
*:System unregistration	
<u>E</u> xecution	

• Directory

This box enables the specification of a directory in which cutting condition files are to be changed. Initially, the box indicates the cutting condition file directory specified using Setting of Environment.

• Cutting condition file list

Clicking the List button displays a list of cutting condition files in a specified directory.

If a cutting condition file name in the cutting condition file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the corresponding condition file exists in the directory.

Cutting conditions selected from the cutting condition list are subjected to change.

• All/Nothing button

This button either selects or deselects all the cutting condition files in the cutting condition list. Clicking the button toggles between selection and deselection of all the cutting condition files.

• Execution button

After the target cutting condition file has been selected from the cutting condition file list, clicking the Execution button displays the following dialog box for specifying the name data to be used after the change.

Specification of the data after changing		
🗖 Oliginal Data is Left		
Material Name:	AL	
NC Machine:		
File Name :	WKMATALM	
(No extension, Max. 8 characters)		
OK Cancel		

• Original Data is Left

If the check box for this item is not selected, the original cutting condition file is replaced with a new cutting condition file (resulting from the change); the original is deleted.

If the check box is selected, the original cutting condition file is not deleted. Instead, it is duplicated, using different names for the original and new cutting condition files.

Material Name

This box is used to specify the work material name to be used in the new cutting condition file.

NC Machine

This box is used to specify the NC machine name to be used in the new cutting condition file.

• File Name

This box is used to specify the file name to be used for the new cutting condition file.

If the Original Data is Left check box is not selected, there is no need to change the file name. If the check box is selected, an error message is displayed unless the new file name differs from the original file name.

• Next/OK button

When there are other cutting condition files for which new data is to be specified, the Next button is displayed. After the new data has been specified, clicking the Next button displays a dialog box for changing the data of another cutting condition file.

-187-

If new data for the last cutting condition file is specified, the OK button is displayed. After the new data has been specified, clicking the OK button applies the specified changes, then returns you to Changes of cutting condition file dialog box.

It is possible to change more than one cutting condition file at a time.

Cancel button

Clicking the Cancel button returns you to Change of cutting condition file dialog box, without applying the specified changes.

Close button

Clicking the Close button returns you to Cutting condition data utility dialog box.

14.7 Uniting Cutting Condition Files

Clicking the Uniting button in Cutting condition data utility dialog box displays the following dialog box, used to unite cutting condition files.

- Uniting of cutting condition files				
First condition file Directory: G:\SYMCAPT\PCFAPT.E List Work Material Name(NC Machine Name) S45C AL FC25	Second condition file Directory: G:\SYMCAPT\PCFAPT.E List Work Material Name(NC Machine Name) S45C AL FC25			
*:System unregistration	*:System unregistration			

- First condition file
 - Directory

This box is used to specify the directory that contains one of the cutting condition files to be united. Initially, the box indicates the cutting condition file directory specified using Setting of Environment.

• Cutting condition file list

Clicking the List button displays a list of the cutting condition files in a specified directory.

If a cutting condition file name in the cutting condition file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the cutting condition file exists in the directory.

A cutting condition file selected from the cutting condition file list is subjected to file uniting. Only one cutting condition file can be selected from this list.

• Second condition file

-189-

Directory

This box is used to specify the directory that contains the second cutting condition file to be united. Initially, the box indicates the cutting condition file directory specified using Setting of Environment.

• Cutting condition file list Clicking the List button displays a list of the cutting condition files in a specified directory.

If a cutting condition file name in the cutting condition file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the cutting condition file exists in the directory.

The cutting condition file selected from the cutting condition file list is subjected to file uniting. Only one cutting condition file can be selected from this list.

• Execution button

After both the first and second cutting condition files have been selected, clicking the Execution button displays the following dialog box, used to specify the conditions for file uniting.

-	Specifica	tion of uniting condition
Method of preserving the same data		
Preserve data of first condition file		
O Preserve data of second condition file		
Mate	rial Name:	S45C
NC M	achine:	
Method of preserving uniting file		
O Replacement first condition file		
O Replacement second condition file		
Preserve new condition file		
Dire	ctory: D	:\CAPT.E
File	Name :	
(No extension, Max. 8 characters)		
OK Cancel		

- Method of preserving the same data
 - Preserve data of first condition file
 - Preserve data of second condition file

If two files to be united have different data for the same item, the Preserve data of first condition file and Preserve data of second condition file items enable the specification of which file is to supply data to the new file. For example, if both the first and second cutting condition files have cutting condition data for roughing preceding contouring, end-milling, and tool material HSS, the cutting condition data from the specified file is saved in the new file.

Material Name

This box is used to specify the work material name to be used in the new cutting condition file (resulting from uniting).

- NC Machine This box is used to specify the NC machine name used in the new cutting condition file (resulting from uniting).
- Method of Preserving Uniting File
 - Replacement First Condition File
 - Replacement Second Condition File
 - Preserve New Condition File

These items enable the specification of the way in which the new cutting condition file is to be saved. If the Preserve New Condition File item is selected, the user must specify the directory in which the new file is to be saved, as well as the file name.

Directory

This box is used to specify the directory in which the new cutting condition file is to be saved. Initially, the box indicates the cutting condition file directory specified using Operating Environment.

• File Name

This box is used to specify a name for the new cutting condition file.

OK button

After the conditions for file uniting have been specified, clicking the OK button applies the specified uniting, then returns you to Uniting of cutting condition files dialog.

• Cancel button

Clicking the Cancel button returns you to Uniting of cutting condition files dialog without applying the specified uniting.

 Close button Clicking the Close button returns you to Cutting condition data utility dialog box.

15 TOOL DATA AND TOOLING DATA

This section describes how to edit the tool data and the tooling data.

15.1 Overview

Symbolic CAP T defines tool data and tooling data as described below:

Tool data

Tool data consists of data relating to all the tools in a manufacturing facility, for example, organized according to tool type. The data for all the available tools can be registered as tool data. Data that has been registered only as tool data cannot be referenced or used in machining definition.

• Tooling data

Tooling data is registered data for those tools selected from all tools that are to be mounted on a machine to enable the machining of a workpiece. A tool for which data is registered in the tooling data can be referenced as part of machining definition.

Two or more toolings can be registered. In this case, the tooling data to be referenced or used in machining definition is selected using Setting of pre-machining dialog.

Usually, tooling data is created as follows: Data relating to all available tools is registered as tool data. The data for a tool to be used in machining definition is copied into the tooling data.



WARNING

Before programming using the conversational function, ensure that correct tool data and tooling data are set. Failure to set correct tool data and tooling data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

15.2 Functions

For tool data and tooling data, the following functions are available:

- Addition and deleting of a tooling data This function adds, or deletes a tooling data. Tooling data add using this function can be referenced in machining definition.
- Modification of tooling data

This function is used to modify add tooling data.

• Editing the tool data

This function is used to modify the tool data for a given tool type.

• Activation of tool/tooling data utility

Activate the utility function used for tool/tooling data input/output and modification. For details of the tool/tooling data utility, see Tool/Tooling Data Utility.

15.3 Starting of Tool Data/Tooling Data Editing

From the menu bar, select Setting => Library Data Setting => Tool, Tooling.

Edit Tool Data, Tooling Data window appears, as shown below:

Edit Tool Data, Tooling Data	×
Tooling Name	
Tooling 1	Close
	<u>U</u> tility
<u>A</u> dd <u>M</u> odify	

15.4 Addition of Tooling data

Click the Addition button on Edit Tool Data, Tooling Data window. Addition of Tooling Data dialog appears, as shown below:

	Addition of Tooling Data			
● New Making of Tooling File				
Tooling Name :		ОК		
File Name :		Cancel		
(No extension,	Max. 8 characters)			
• Reading of Tooling File				
Directry :				
	<u>L</u> ist Display			
	Tooling Name			

A tooling file(Note 1) must be placed in the directory specified in Setting of Environment Dialog of the Setting menu.

The tooling file can be added in either of the following two ways:

• Creating a tooling file

A new tooling file is created and placed in the Toolin/Tool File directory specified in Setting of Environment Dialog of the Setting menu.

• Reading an existing tooling file

An existing tooling file is copied into the directory specified in Setting of Environment dialog of the Setting menu.

NOTE 1

Tool file, tooling file, tool data, and tooling data:

This chapter refers to a file containing data related to tools as a tool file or tooling file. The data in the tool file or tooling file and the information registered in the system are referred to as tool data or tooling data.
(1) Creating a tooling file

Select the New Making of Tooling File option on Addition of Tooling Data window. Specify a tooling name and file name, then click the OK button.

Edit Tool Data, Tooling Data window appears. For an explanation of how to edit the tooling, see the section describing modification of the tooling data.

(2) Reading an existing tooling file

Select the Reading of Tooling File option on Addition of tooling data window. Specify the directory in which tooling files are stored, then click the List Display button. Select a target tooling file from the list, then click the OK button. Addition of Tooling Data window remains displayed and another tooling file can be read. To return to Edit Tool Data, Tooling Data dialog, click the Cancel button.

15.5 Deleting a Tooling

On Edit Tool Data, Tooling Data window, select the tooling name to be deleted, then click the Delete button. Tooling data referenced during machining definition cannot be deleted.

The window for deleting a tooling data appears, as shown below:

	Deletion of Tooling Data		
• Delete	from Tooling Name List		
O Delete	Tooling File		
Tooli	ng Data		
То	oling Name : Tooling 2		
Delet	e Ok ?		
	OK Cancel		

The tooling data can be deleted in either of the following two ways:

- Deleting the tooling data only from the tooling name list The tooling is deleted from the tooling name list and cannot be referenced in machining definition. The tooling file, however, is not deleted from the Tooling/Tool File directory specified with Settings of Environment.
- Deleting both the tooling name list entry and the tooling file The tooling is deleted from the tooling name list. In addition, the tooling file is deleted from the Tooling/Tool File directory specified with Setting of Environment.

To delete the tooling, select the Delete from Tooling Name List option or the Delete Tooling File option, then click the OK button.

15.6 Modification of the Tooling Data

On Edit Tool Data, Tooling Data window, select the tooling name to be edited, then click the Modify button.

The window for modifying tooling data and tool data appears, as shown below:

Modify Tool Data, Tooling Data				×
- Tool Data				
Tool Type : Center Drill		© Add	OInsert	
Tool Name	Dia. T	ooth Length	Tooth Num. Tool	,Offset No.
Center Drill-1	4.000	1.790		
Center Drill-2	5.000	2.430		
Center Drill-3	6.000	3.580		
Center Drill-4	8.000	4.220		
Center Drill-5	10.000	4.870		•
New Make Modify	<u>D</u> elete		<u>С</u> ору	to Tooling Data
Tooling Data				
Tooling Name : Tooling 1			🖲 Add	
rooming have a rooming r			O Insert	
Tool Name			Tool Data	
Center Drill-1	4.000	1.790		
Center Drill-2	5.000	2.430		
Center Drill-3	6.000	3.580		
Center Drill-4	8.000	4.220		
Center Drill-5	10.000	4.870		_
New Make Modify	Delete			
	Delete			

Clicking the OK button after editing the tooling data or tool data makes the edited data effective.

Clicking the Cancel button after editing the data discards all changes that have been made.

15.6.1 Adding or inserting the tool data for a tool into the tooling data

To add or insert the registered tool data for a tool into the tooling data, select a target tool from the tool data list on Modify Tool Data, Tooling Data window, then click the Copy to Tooling Data button. The tool data for two or more tools can be copied into the tooling data at one time.

Selecting the Add option of the tooling data causes the tool data to be added to the end of the tooling data.

Selecting the Insert option causes the tool data to be inserted before the tool currently selected in the tooling data list.

To display the tool data for another tool type, click Tool Type, display the tool type list, then select the desired tool type name.

15.6.2 Registering a new tool and adding or inserting the data into the tooling data

To add or insert the data for a new tool into the tooling data, click the New Make button in the tooling data field on Modify Tool Data, Tooling Data window.

The conversational window for registering and modifying tool appears.

Modify Tool					×
Tool Type:	Center Drill 💌	Tool	Name:	Center Drill-	
Туре :	Milling, Turning 🔻	Mate	rial:	SKH9	-
*Tool No. :				<u>M</u> aterial Edit	
*Offset No.:					
Tool Dimension I Dia.(DN) :	Data 6.0	100	- Auxiliary Rot. Dir	v, Setting Data —	Normal 🔽
DT :	6.0	100			
DS :	3.0	100			
LT :	3.0	100	*Set Pos	i.(ZS):	0.000
AT :	60.	000	*Set Pos	i.(XS):	0.000
<u>E</u> xplanation	∗ : Possible U	ndefine	20	OK	Cancel

Tool Name

Specify a unique tool name.

Material Name

Select a tool material from the tool material name list.

To add or delete a tool material name, do so after clicking the Material Edit button. Tool materials can be registered for individual tool types.

• Tool No., Offset No.

Specify a tool number and tool offset number for NC data output. These numbers need not always be specified on this window. If not specified, the numbers must be specified in the machining definition.

• Dimension information

Specify the dimensions of a tool. Specify all dimensions, including the angle, as a positive value.

To display an explanatory drawing, click the Explanation button.

Rotate Direction

For a milling tool, specify the direction of rotation for the tool. For a turning tool, specify the direction of rotation for the workpiece.

-200-

• Blade Protect Angle (turning tool only)

For pocketing or turning, specify a clearance angle between the cutting edge and material.

This angle need not always be specified on this window. If not specified on this window, however, the angle must be specified in the machining definition.



• Number of blades

Specify the number of blades of a tool such as a reamer.

• Holder type

Specify the holder type of a tapping tool.

• Mounting position (ZS,XS)

Specify the relative distance between the tip of a tool to the reference point (usually, the turnet turning position) when the tool is mounted.

Clicking the OK button after entering the tool causes the data to be added to, or inserted into, the tooling data.

Selecting the Add option of the tooling data on Modify Tool Data, Tooling Data window causes the tool to be added to the end of the tooling data. Selecting the Insert option of the tooling data causes the tool data to be inserted before the tool currently selected in the tooling data list.

15.6.3 Modifying tool registered in the tooling data

To modify tool that has been registered in the tooling data, first select the target tool from the tooling data list, then click the Modify button for the tooling data on Modify Tool Data, Tooling Data dialog.

The conversational window for registering and modifying tool data appears. Modify the tool, then click the OK button.

Multiple tools can be modified in succession by selecting the tools from the tooling data list and clicking the Modify button.

15.6.4 Duplicating a tool in tooling data

This function is used to modify tooling data only. Any tool registered in the tooling data can be duplicated by selecting the tool from the tooling data list and clicking the Copy button for the tooling data.

A dialog box for tool registration and modification appears. After modifying the data in the dialog box, click the OK button.

If the Add to End option has been selected for the tooling data, the copy of the specified tool is added to the end of the tooling data.

If the Insert before Highlight option has been selected for the tooling data, the copy of the specified tool is inserted before the selected tool in the tooling data list.

When more than one tool has been selected from the tooling data list, clicking the Copy button enables the selected tools to be duplicated in succession.

15.6.5 Changing the order of the tools in the tooling data

This function is used to modify tooling data only. The order in which tools registered in the tooling data will appear can be changed by selecting a tool from the tooling data list and clicking the Change Order button for the tooling data. The following message appears at the bottom of the dialog box:

Specify the new position for the highlighted (top) line

Selecting a position from the non-highlight list moves the selected tool to that position.

The positions of two or more tools can be changed at the same time.

15.6.6 Deleting tooling from the tooling data

To delete tool from the tooling data, select a target tool from the tooling data list, then click the Delete button for the tooling data on Modify Tool Data, Tool Data dialog.

A prompt window requesting the user to confirm the deletion appears. To delete the data, click the Yes button.

Two or more tools can be deleted at one time.

15.6.7 Registering a new tool and adding or inserting tool data

To add or insert new tool, click the New Make button for the tool data on Modify Tool Data, Tooling Data dialog.

Modify Tool dialog appears. Enter the tool, then click the OK button. The tool is added or inserted.

Selecting the Addition option for the tool data on Modify Tool Data, Tooling Data dialog causes the tool to be added to the end of the tool data.

Selecting the Insert option for the tool data causes the tool to be inserted before the tool currently selected in the tool data list.

15.6.8 Modifying tool registered in the tool data

To modify registered tool, select the target tool from the tool data list, then click the Modify button for the tool data on Modify Tool Data, Tooling Data window.

Modify Tool dialog appears. After modifying the data, click the OK button. Multiple tools can be modified in succession by selecting the tools from the tool data list and clicking the Modify button.

15.6.9 Duplicating the tool in the tool data

This function is used to modify tool data only. A specific registered tool can be duplicated by selecting the tool from the tool data list and clicking the Copy button for the tool data. Modify Tool dialog appears. After modifying the data in the dialog box, click the OK button.

If the Add option has been selected, the specified tool is added to the end of the existing tool data.

If the Insert option has been selected, the new tool is inserted before the tool data for the selected tool in the tool data list.

When more than one tool has been selected from the tool data list, clicking the Copy button causes the tool data for all the selected tools to be duplicated.

15.6.10 Changing the order of the tools in the tool data

This function is used to modify tool data only. The order in which tools registered in the tool data will be displayed can be changed by selecting a tool from the tool data list and clicking the Change Order button for the tool data. The following message appears at the bottom of the dialog box:

Select Position of Top Reversed List after Changed Order

Selecting a line from the unhighlighted list moves the selected tool to that line.

The order of two or more tools can be changed at the same time.

15.6.11 Deleting tool from the tool data

To delete registered tool, select the target tool from the tool data list, then click the Delete button for the tool data.

A prompt window asking the user to confirm the deletion appears. To delete the, tool click the Yes button.

The tool for two or more tools can be deleted at the same time.

15.6.12 Adding or deleting a tool material name only

To add or delete a tool material name only, click the New Make button for the tooling data on Modify Tool Data, Tool Data.

Modify Tool dialog appears. Tool material names are specified for each tool type. Select the target tool type from the tool type list, click the Material Edit button, then add or delete the material name.

After adding or deleting a tool material name, close the Modify Tool dialog box by clicking the Cancel button. Only a tool material name modification is made.

16 TOOL/TOOLING DATA UTILITY

This section describes the tool/tooling data utility for the input/output of tool files and tooling files.

16.1 Functions

The tool/tooling data utility has the following functions.

• Tool/tooling file output

Tool and tooling files are output from "Tooling/Tool File" directory, specified using Settings of Environment, to a specified directory.

• Tool/tooling file input

Tool and tooling files are input from a specified directory to Tooling/Tool File directory, specified using Setting of Environment.

• Tooling file change

Tooling names used in, and the name of, a tooling file in a specified directory are changed. This function can also be used to copy a tooling file while leaving the original tooling file as is.

16.2 Starting the Tool/Tooling Data Utility

The tool/tooling data utility is started from Tool Data, Tooling Data dialog. To start the utility, select Setting => Library Data Setting => Tool, Tooling from the menu bar.

The following Tool Data, Tooling Data dialog box appears.

💳 Tool Data, Tooling Data		
Tooling Name		
Tooling 1	Close	
	Utility	
Addtion Modify		

Click the Utility button. Tool/Tooling data utility dialog box appears.

16.3 Selecting Utility Functions

Tool / Tooling data utility			
<u>O</u> utput	Close		
<u>I</u> nput			
C <u>h</u> ange			

Clicking a function button displays the dialog box corresponding to the selected function.

Clicking the Close button returns you to Tool Data, Tooling Data dialog box.

16.4 Outputting Tool/Tooling Files

Clicking the Output button in Tool/Tooling data utility dialog box displays the following dialog box for tool/tooling file output.

🗢 Outpu	ıt of Tool / Tooling file
Output origin	Output destination
Directory: G:\SYMCAPT\PCFAPT.E	Directory:
	List
Tool File	Tool File
Center Drill Drill Counter Sink Reamer Bore1	
Tooling	Tooling
Tooling 1	
*:System unregistration All([iool] All(Tooling)	*:Unregisters in managment file Execution

- Output origin
 - Tool file list

This is a list of tool files in Tooling/Tool File directory, specified as the output origin using Setting of Environment. Tool files selected from the tool file list are output.

• All (Tool)/Nothing (Tool) button

This button either selects or deselects all the tool files in the tool list. Clicking the button toggles between the selection and deselection of all the tool files.

• Tooling file list

This is a list of tooling files in Tooling/Tool File directory, specified as an output origin using Setting of Environment.

If a tooling file name in the tooling file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the tooling file exists in the directory.

The tooling files selected from the tooling file list are output.

• All (Tooling)/Nothing (Tooling) button

This button either selects or deselects all the tooling files in the tooling list. Clicking the button toggles between the selection and deselection of all the tooling files.

- Output destination
 - Directory

This box is used to specify a directory to which the tool/tooling files are to be output.

-208-

• Tool file list

Clicking the List button displays a list of the tool files in a specified directory.

• Tooling file list

Clicking the List button displays a list of the tooling files in a specified directory.

If a tooling file name in the tooling file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the tooling file exists in the directory.

• Execution button

Tool and tooling files are output by selecting them from the respective output origin lists, specifying the output destination directory, then clicking the Execution button.

The tool and tooling file lists at the output destination are updated. Tool/tooling file output can continue after the completion of update.

It is possible to output more than one tool or tooling file at a time.

• Close button

Clicking the Close button returns you to Tool/Tooling data utility dialog box.

16.5 Inputting Tool/Tooling Files

Clicking the Input button in Tool/Tooling Data Utility dialog box displays the following dialog box for tool/tooling file input.

📼 Input of Tool / Tooling file					
Input destination	Input origin				
Directory: G:\SYMCAPT\PCFAPT.E	Directory:				
	List				
Tool File	Tool File				
Center Drill Drill Counter Sink Reamer Bore1					
Tooling	Tooling				
Tooling 1					
*:System unregistration	*:Unregisters in managment file				
Execution	All(<u>Tool</u>) All(T <u>o</u> oling)				

- Input destination
 - Tool file list

This is a list of the tool files in Tooling/Tool File directory, specified as an input destination using Setting of Environment.

• Tooling file list

This is a list of the tooling files in Tooling/Tool File directory, specified as an input destination using Setting of Environment. If a tooling file name in the tooling file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the tooling file exists in the directory.

Input origin

.

Directory

This box is used to specify the directory from which tool/tooling files are to be supplied.

Tool file list Clicking the List button displays a list of the tool files in a specified directory.

The tool files selected from the tool file list are input.

• All (Tool)/Nothing (Tool) button

This button either selects or deselects all the tool files in the tool list. Clicking the button toggles between the selection and deselection of all the tool files.

 Tooling file list Clicking the List button displays a list of the tooling files in a -210specified directory.

If a tooling file name in the tooling file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the tooling file exists in the directory.

The tooling files selected from the tooling file list are input.

• All (Tooling)/Nothing (Tooling) button

This button either selects or deselects all the tooling files in the tooling file list. Clicking the button toggles between the selection and deselection of all the tooling files.

• Execution button

Tool and tooling files are input by selecting them from the input origin tool and tooling file list, then clicking the Execution button.

The tool and tooling file lists at the input destination are updated. Tool/tooling file input can continue after the update.

It is possible to input more than one tool or tooling file at a time.

Close button

Clicking the Close button returns you to Tool/Tooling data utility dialog box.

16.6 Changing Tooling Files

Clicking the Change button in Tool/Tooling data utility dialog box displays the following dialog box for tooling file change.

-	Change of Tool / Tooling file			
		Close		
Directory:	D:\CAPT.E			
	List			
	Tooling			
Tooling 1				
*:System un	registration			
	All(T <u>o</u> oling)			
Execution				

• Directory

This box is used to specify the directory containing the tooling files to be changed. Initially, the box indicates the Tooling/Tool File directory specified using Setting of Environment.

• Tooling file list

Clicking the List button displays a list of the tooling files in a specified directory.

If a tooling file name in the tooling file list is suffixed with an asterisk, the relevant information has not been registered as system management information, even though the tooling file exists in the directory.

The tooling files selected from the tooling file list are subjected to change.

• All (Tooling)/Nothing (Tooling) button

This button either selects or deselects all the tooling files in the tooling list. Clicking the button toggles between the selection and deselection of all the tooling files.

• Execution button

After the target tooling list file has been selected from the list of tooling list files, clicking the Execution button displays a dialog box for specifying the name to be used after the change.

-212-

Specification of the data after changing			
🗖 Oliginal Data is Left			
Tooling Name:	Tooling 1		
File Name:	TLSET1		
(No extension, Max. 8 characters)			
OK C	Cancel		

• Original Data is Left

If the check box for this item is not selected, the original tooling file is replaced with the new tooling file; the original is deleted. If the check box is selected, the original tooling file is not deleted. Different names are used for the original and new tooling files.

Tooling Name

This box is used to specify the tooling name for the new tooling file.

• File Name

This box is used to specify the file name for the new tooling file. If the Original Data is Left check box is not selected, there is no need to change the file name. If the check box is selected, an error message is displayed unless the original file name differs from the new file name.

• Next/OK button

If there are other tooling files for which new data is to be specified, the Next button is displayed. Once new data has been specified, clicking the Next button displays a dialog box for changing the data of the next tooling file.

Once new data for the last tooling file has been specified, the OK button is displayed. Clicking the OK button applies the specified changes, then returns you to the tooling file change dialog box.

It is possible to change more than one tooling file at a time.

• Cancel button

Clicking the Cancel button returns you to Change of Tool/Tooling file dialog box without applying the specified changes.

Close button

Clicking the Close button returns you to Tool/Tooling data utility dialog.

17 NC MACHINE SETTING

With Symbolic CAP T, NC data can be prepared by means of blank figure creation, part figure creation, and machining definition. However, the format of the NC data will vary from one NC machine to another.

To enable these different NC data formats to be handled, Symbolic CAP T provides NC machine files that are used for NC data output format management. Editing an NC machine file to tailor it to a particular NC machine is referred to as NC machine setting.

This chapter describes NC machine setting.

17.1 NC Machine File

When NC data is prepared, an NC machine file is referenced to determine the NC data output format for the NC machine.



17.2 Overview of NC Data Preparation Procedure

With Symbolic CAP T, tool path data is created from figure data and machining data, after which the created tool path data is converted to NC data.

- (1) Tool path data is created from figure data and machining data.
- (2) The created tool path data is converted to NC data by referencing data such as parameters in the NC machine file.

17.3 Modifying the NC Data Output Format

The output format of the NC data can be tailored to a given NC machine by editing the NC machine file to suit the NC machine. Each NC machine file consists of parameters and machining pattern macros, as shown below.

• Parameters

Parameters are used to determine NC data formats. The output formats include those of individual functions such as an address word and G code (such as character string and numeric value formats), and those based on CNC specifications (such as the circular interpolation and threading formats).

• Machining pattern macros

Macro programs are used to determine the output formats of machining patterns (such as tool change, spindle rotation, and coolant control) other than those for cutting. (For details of the macro language, refer to the appendix.)

By means of parameter setting, NC data output formats can be modified. If the output formats cannot be modified as required by parameter setting alone, the machining pattern macros must be edited.



Machine	 Home position and index position setting 		
specifications	Number of simultaneously controlled axes		
	 Spindle speeds (maximum and minimum) 		
	Rapid traverse rate		
CNC	Modification of address characters (such as move		
specifications	axes, F, S, and T)		
	 Total number of numeric value digits 		
	Whether to output a decimal point		
	Number of decimal places		
	Minimum input increment		
	Output character string modification of G code		
	and M code		
	G code group number modification		
	Setting of arbitrary output character strings		
	Circular interpolation method		
	 Whether to use a threading cycle 		
	Whether to use a hole machining cycle		
	Whether to move the home position and index		
	position		
NC data format	 Start and end sections of NC data 		
	 Move block (approach, retraction, normal) 		
	Threading cycle		
	Hole machining cycle		
	Others		

• Parameters (machine setting/CNC setting)

Start of NC data	 Program number and program name
(POST_INT)	Sequence number
	ABS/INCR
Tool registration	 Storing of tool numbers and offset numbers
(POST_TOOL)	(only when the use of the ATC function is
	specified in the machine specifications)
Coordinate system	Setting of workpiece coordinate systems
registration	(only when the use of the ATC function is
(POST_CORD)	specified in the machine specifications)
Preprocessing	Reference position return (only when this item is
before start of	specified in the CNC specifications)
machining	• Coordinate system setting (only when this item is
(POST_HOME)	specified in the CNC specifications)
Start of main	Coordinate system setting (Whether to output
machining(POST_	this information can be specified.)
PROCINT)	Coolant command
	Maximum speed command
	 Constant surface speed control command
	Spindle rotation command
	Tool change command
Start of sub-	 Coolant command (not output by default)
machining(POST_	 Maximum speed command (not output by
PROCSUB)	default)
	Constant surface speed control command (not
	output by default)
	• Spindle rotation command (not output by default)
	 Tool change command (not output by default)
Preprocessing	Coolant OFF
before retraction	Spindle stop
(POST_PROCESC	
)	
Termination of	Offset cancel (only when this item is set)
main machining	
(POST_PROCEND	
)	
Canned	Output of each cycle command
cycle(POST_DRIL	
LCYCL)	
End of NC	Reference position return (only when this item is
data(POST_END)	set in the CNC specifications)
	Program end output

• Machining pattern macros

17.4 Machine Specifications

17.4.1 NC machine name

Assign a name to each NC machine. After the NC machine files have been stored, an NC machine file can be selected simply by specifying the NC machine name.

[Restrictions]

- Maximum number of characters: 24
- Duplicate machine names must not be specified.

17.4.2 Home position/index position

Specify a home position and index position. Specified data is also displayed on the Setting of Pre-machining window.

- Position in machine coordinate system
- Index center
- Use a radius value for input.
- Start position for tool path drawing
- Set -1.0 for an unused item.

17.4.3 Total number of axes/number of simultaneously controlled axes

Specify the number of machine axes.

- Total number of axes
- Number of axes that are simultaneously controlled in rapid traverse
- Number of axes that are simultaneously controlled in cutting feed

If the specified number of simultaneously controlled axes is less than the total number of axes, tool operation is divided into multiple operations so that the tool can operate within the number of simultaneously controlled axes.

[Restrictions]

- The number of simultaneously controlled axes must not exceed the total number of axes.
- A value of 1 or above must be set.

17.4.4 Feedrate

Set the feedrates as follows:

- Specify a rapid traverse rate for each axis.
- Specified rapid traverse rates are used for machining time calculation, and are indicated in the machining list.

17.4.5 Spindle speed

Specify the maximum and minimum spindle speeds.

- Specify a maximum and minimum spindle speed for each stage.
- A stage switching command is output if the spindle speed moves outside the range specified with the maximum and minimum values.
- A minimum or maximum speed is output if no stage is applicable.
- Specify -1 for an unused item.

Example:

Minimum speed for first stage 100					
Maximum speed for first stage 4500					
Minimum speed for second stage 3000					
Maximum speed for sec	cond stage	9000			
Minimum speed for this	rd stage -1	.0			
Maximum speed for thi	rd stage -1	.0			
Speed command	2000	\rightarrow	S2000		
			:		
Speed command	4000	\rightarrow	S4000		
			:		
Speed command	6000	\rightarrow	M?? (M code for 2nd		
range switching)					
			:		
			S6000		
			:		
Speed command	12000	\rightarrow	S9000		

17.4.6 Whether to use the ATC function

Specify whether the automatic tool changer (ATC) function can be used with the NC machine. When the ATC function is supported, register all tools and the coordinate systems for every process, using the following machining pattern macros:

- POST_TOOL(): Tool registration
- POST_CORD(): Coordinate system registration

17.5 CNC Specifications

17.5.1 Address setting

(1) Address word

Set an address word for the NC data for each function. The characters set here are output as NC data.

- Address word (such as a move axis) for real number output
- Address word (such as a T code) for integer output
- Modal management is performed for each function.
- Some address words are set separately for absolute and incremental programming.

Example:

First linear axis "X" \rightarrow "A" Second linear axis "Y" \rightarrow "B" Third linear axis "Z" \rightarrow "C"

 $G01X100.0Z50.0 \rightarrow G01A100.0C50.0$

[Restrictions]

• Only a single character can be specified for each address word.

(2) Minimum input increment

Specify a minimum input increment for real numbers in the NC data. Real numbers in the NC data, when output, are rounded according to the minimum input increment.

- Specify a real number.
- Usually, the same value as that specified for the number of decimal places is used.

(3) Number of decimal places

Specify the number of decimal places for real numbers in the NC data.

- Specify an integer.
- Usually, the same value as that specified for the minimum input increment is used.

(4) Whether to output a decimal point

Specify whether to output a decimal point when outputting real numbers in the NC data.

Example:

Minimum input increment 0.001 Number of decimal places 3

Movement on the X-axis	With a decimal point	Without a decimal point
100.1234 \rightarrow	X100.123	X100123
100.0 →	X100.0	X100000

(5) Maximum number of digits

Set the maximum number of digits to be used when numeric values (real numbers and integers) in the NC data are output.

- A warning is output when a numeric value that exceeds the maximum number of digits is output.
- Specify a value that includes no decimal point nor sign.

17.5.2 G code

Set a G code in the NC data for each function. The characters set here are output as NC data.

- Set a group number.
- Modal management is performed on a group-by-group basis.
- Set "G-1" for any unused functions.

Example:

Linear interpolation "G01" \rightarrow "Linear"

 $G01X100.0Z50.0 \rightarrow LinearX100.0D50.0$

[Restrictions]

• Use no more than seven characters for the setting.

17.5.3 M code

Specify an M code in the NC data for each function. The characters set here are output as NC data.

- Specify whether to perform modal management.
- Specify "M-1" for any unused functions.
 - Example:

```
Subprogram call "M98" \rightarrow "Call"
M98P1000 \rightarrow CallP1000
```

[Restrictions]

• Use no more than seven characters for the setting.

17.5.4 Arbitrary character string

Specify arbitrary character strings that will be output in the NC data regardless of the function. Up to 16 character strings can be set. To output character strings, specify the function code for arbitrary character string output in the NC data format specification.

17.5.5 Specifying required functions

Specify the CNC functions to be used. NC data is output based on the functions specified here.

- Whether to use the constant surface speed control function
- Whether to use a canned hole machining cycle
 - Circular interpolation function
 - Radius output/IJK output
 - Minimum radius/maximum radius
 - Allowable error
- Linear axis sign inversion
- Spindle speed fluctuation ratio

Specify a spindle speed fluctuation ratio to output a fluctuated spindle speed that exceeds the currently set fluctuation ratio.

Example: When a fluctuation of 0.01 is set Spindle speed $4000 \rightarrow S4000$ $4010 \rightarrow$ This fluctuation does not exceed the currently set fluctuation, 4000 ± 40 (= 4000×0.01), so this spindle speed is not output. $4050 \rightarrow S4050$

Feedrate fluctuation ratio

Specify a feedrate fluctuation ratio to output a fluctuated feedrate that exceeds the currently set fluctuation ratio.

Example: When the currently set fluctuation is 0.01 Feedrate $300 \rightarrow F0300$ $302 \rightarrow$ This fluctuation does not exceed the currently set fluctuation, $300 \pm 3 \ (= 300 \times 0.01)$, so this feedrate is not output. $310 \rightarrow F0310$

- Minimum number of digits to be specified in a feedrate command
- Threading
 - Whether to use a threading cycle
 - Taper direction
 - Lead direction
 - Number of finish machining passes
 - Chamfer amount

0 = 0.0L (L = Lead) 1 = 0.1L 10 = 1.0L99 = 9.9L

- Format between a home position and index position
- T code output after the completion of machining

17.6 NC Data Format

By using the function codes for an NC machine file, specify the formats of the NC data to be output. (For details of the function codes, see the appendix.)

17.6.1 Example of referencing NC data formats

NC data is output when NC data formats are referenced according to the following procedure.

• Example of reference

Tool path data	Movement from the current point (10,10) to (100,100)
Referenced NC data	Movement with or without tool-tip radius compensation
format	(G90_QG91, G00_QG01,Xmot,Zmot,Fmr,EOB) 1 2 3 4 5 6
	 Output of G codes programmed in either absolute or incremental mode The G code output format is set in the CNC specifications. Output of a G code for rapid traverse or cutting feed The G code output format is set in the CNC specifications. Output of first linear axis The address word output format is set in the CNC specifications. Output of third linear axis The address word output format is set in the CNC specifications. Feedrate output The address word output format is set in the CNC specifications. Feedrate output The address word output format is set in the CNC specifications. Feedrate output The address word output format is set in the CNC specifications. Feedrate output The address word output format is set in the CNC Specifications.
Output NC data	 (a) For rapid traverse in absolute mode G90G00X100.0Z100.0;
	 G90 is output because absolute output is specified. G00 is output because of rapid traverse is specified. Absolute output of the first and third linear axes Because rapid traverse is specified, no F code is output.
	 (b) For a feedrate of 100 in incremental mode G91G01X90.0Z90.0F0100; G91 is output because incremental output is specified. G01 is output because cutting feed is specified. Incremental output of the first and third linear axes Because cutting feed is specified, an F code is output.

- Example of modification
 - To suppress G90/G91 output: Delete function codes G90_G91.

G00_G01,Xmot,Zmot,Fmr,EOB

• Specify a null string for the G code setting in the CNC specifications.

"G90" -> " "

"G91" -> " "

17.6.2 Timing of NC data format reference

Symbolic CAP T supports the standard tool path data output format shown below. This section explains the NC data format(s) referenced at each timing. Note, however, that these data formats and their reference timings are not applicable when machining pattern macros are modified.



The NC data formats referenced at the individual timings are explained below.

When multiple NC data formats are referenced, the order in which reference is performed is determined from the order of the machining pattern macro description or tool path preparation.

- Data that can be output in a referenced NC data format can also be output in a subsequently accessed NC data format, depending on the state of the modal information (with any codes having the same value or the same group number not being output).
- The function codes for modal clear (Gclr, Mclr, Tclr, Sclr) and those for arbitrary character strings (Str1, ..., Str16) can be output at any time. Note, however, that modal management is not applied to arbitrary character strings.

(1) Initial processing

• Start of an NC data file

Referenced NC data format	Data (function code) that can be output
Leader section	- % (EOR)
	- Feed (FEEDnull, FEEDsp)

(2) Start of NC data

- Start of main program
- Machine type selection
- ABS/INCR
- Sequence number setting

Referenced NC data format	Data (function code) that can be output
Start of main program	- Program number (Omain)
	- Program name (Name)

(3) Coordinate system setting/tool setting

- Setting of positions of workpiece coordinate systems for all processes
- Storing of tool numbers and offset numbers for all processes

Referenced NC data format	Data (function code) that can be output
Setting of workpiece coordinate	- Workpiece coordinate system number
system positions	(Pwcoord)
	- Position of workpiece coordinate
	system (Xorg,Zorg)

NOTE

Workpiece coordinate systems are not used. So, the NC data format "setting of workpiece coordinate system positions" is referenced, but no NC data is output.

(4) Preprocessing before start of machining

- Coordinate system specification at the home position (only when this item is set in the CNC specifications)
- Retract mode ON

Referenced NC data format	Data (function code) that can be output
Coordinate system setting	- Coordinate system setting
	(G50 or G92)
	 Coordinates of current point
	(Xmot,Zmot)

(5) Start of main machining

- Start of machining
- Workpiece coordinate system selection
- Machining start point specification
- Coolant ON command
- Maximum speed command
- Constant surface speed control ON/OFF
- Spindle rotation command
- Tool change command

Referenced NC data format	Data (function code) that can be output
Machining start section for each	- Machining type (ProcName)
type of machining	
Workpiece coordinate system	- Workpiece coordinate system selection
selection	(G54,, G59)
Coordinate system setting	 Coordinate system setting
	(G50 or G92)
	 Coordinates of the current point
	(Xmot,Zmot)
Coolant ON command or	- Coolant ON M code (M07_M08)
coolant OFF command	- Coolant OFF M code (M09)
Maximum spindle speed clamp	- Maximum speed clamp (G92 or G50)
value specification	- Maximum speed S code (Smaxrpm)
Constant surface speed control	- Constant surface speed control (G96)
command, or constant surface	- Constant surface speed control cancel
speed control cancel command	(G97)
Spindle rotation command	Constant surface speed control
	(G96_G97)
	Spindle rotation M code (M03)
	S code (Srotsur)
Tool change command block	Tool number (Tsel)
output	Offset number (HcomIng)

NOTE

Workpiece coordinate systems are not used. So, NC data format "workpiece coordinate system selection" is referenced, but no NC data is output.

(6) Start of sub-machining

Almost the same data as that used for starting main machining can be used. However, no specifications are added to the standard machining pattern macros. Add specifications as required.

(7) Cutting

- Approach operation
- Positioning
- Linear interpolation
- Circular interpolation
- Threading
- Hole machining

• Approach operation

Referenced NC data format	Data (function code) that can be output
Movement with or without tool- tip radius compensation	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot)
Command following the first approach motion block (when the number of simultaneously controlled axes is 1)	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot)
Command following the second approach motion block (when the number of simultaneously controlled axes is 1)	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current poin (Xmot,Zmot)
Command following a simultaneous specified-axes approach motion block (when the number of simultaneously controlled axes is 2)	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot)

· • • • • • • • • • • • • • • • • • • •	
Referenced NC data format	Data (function code) that can be output
Movement with or without tool- tip radius compensation	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot)
Command following an ordinary motion block	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot)
Movement after start of tool- tip radius compensation(movement immediately after the specification of the start of tool- tip radius compensation)	 Positioning or linear interpolation (G00_G01) Tool-tip radius compensation (G41_42) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot)

• Positioning/linear interpolation

• Circular interpolation

Referenced NC data format	Data (function code) that can be output
Circular interpolation move command (IJK command, ZX plane)	 Circular interpolation (G02_G03) Plane selection (G18) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot) Conter position (leant Keent)
Circular interpolation move command (Radius command, ZX plane)	 Center position (icent, kcent) Circular interpolation (G02_G03) Plane selection (G18) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot) Radius (Rzx)

• Threading

The referenced NC data formats depend on the threading cycle setting in the CNC specifications.

Referenced NC data format	Data (function code) that can be output
Threading command	- Feedrate command (G94_G95)
(Normal threading)	- ABS/INCR (G90_G91)
(Tapered threading)	- Threading (Gthrd)
	 Coordinates (Xmot,Zmot)
	- Lead (Fthrd_lead)
Threading cycle command	 Threading cycle (Gthrd_cycl)
(normal threading)	 Threading end point coordinates
	(Xthrd_cycl,Zthrd_cycl)
	- Lead (Fthrd_lead)
	- Cutting angle (Qthrd_cang)
Threading cycle command	- Threading cycle (Gthrd_cycl)
(tapered threading)	 Threading end point coordinates
	(Xthrd_cycl,Zthrd_cycl)
	- Lead (Fthrd_lead)
	- Taper amount (Ithrd_tapr)
Multiple repetitive threading	 Multiple repetitive threading cycle
cycle (normal threading)	(Gthrd_mul)
	- Threading end point coordinates
	(Xthrd_cycl_mul, Zthrd_cycl_mul)
	- Crest neight (Kthrd_neight)
	- Initial depth of cut (Dthrd_cut)
	- Thread angle (Athrd tang)
	- Threading type (Pthrd_type)
	- Cutting angle (Othrd cang)
	- Minimum depth of cut (Qthrd_mincut)
	- Finishing allowance (Rthrd_fallow)
	- Number of finishing passes
	(Pthrd_floop)
	- Chamfer amount (Pthrd_chamf)
Multiple repetitive threading	- Multiple repetitive threading cycle
cycle (tapered threading)	(Gthrd_mul)
, (Threading end point coordinates
	(Xthrd_cycl_mul, Zthrd_cycl_mul)
	- Taper amount (Ithrd_tapr)
	- Crest neight (Kthrd_neight)
	- Initial depth of cut (Dthrd_cut)
	- Thread angle (Athrd tang)
	- Threading type (Pthrd_type)
	- Cutting angle (Qthrd_cang)
	- Minimum depth of cut (Qthrd mincut)
	- Finishing allowance (Rthrd_fallow)
	- Number of finishing passes
	(Pthrd_floop)
	 Chamfer amount (Pthrd_chamf)
• Hole machining

The referenced NC data formats depend on the type of canned cycle. When no canned hole machining cycle function is used, ordinary move commands or tapping commands are referenced. The following data can be output in canned cycles other than the cancel command:

- Return level (G98_G99)
- ABS/INCR (G90_G91)
- Hole position (Xmot)
- Hole bottom position (Zcycl)
- R point coordinate (Rcycl)
- Feedrate (Fmin)
- Number of repeats (Lcode)

Referenced NC data format	Data (function code) that can be output
Canned cycle cancel command	- Return level (G98_G99)
	- Canned cycle cancel (G80)
Tapping cycle	- Tapping (G84)
	- Dwell (Pcycl)
Reverse tapping cycle	- Reverse tapping (G74)
	- Dwell (Pcycl)
Rigid tapping	- Rigid tapping (G84_2)
c c	- Shift amount (Qsft)
	- Dwell (Pcycl)
	- S code (Srot)
Reverse rigid tapping	- Reverse rigid tapping (G84_3)
5 11 5	- Shift amount (Qsft)
	- Dwell (Pcycl)
	- S code (Srot)
Boring cycle	- Boring (G85/G86/G88/G89)
0.7	- Dwell (Pcycl)
Drilling cycle	- Drilling (G81)
Peck drilling cycle	- Peck drilling (G83/G73)
	- Pecking amount (Qpeck)
Specification of a hole position	

in a canned cycle

In hole machining based on Symbolic CAP T, the NC statement output format depends on the setting of the G code used for the canned cycle of each machining type and the setting of whether to use a canned cycle function for each machining type. The table below indicates the canned cycle settings for Symbolic CAP T hole machining types in a standard NC machine file. By modifying these settings as required, the NC statement output format can be modified.

NOTE

A machining type marked with (*) can be specified only when the C/Y-axis module is installed.

// 1			
NC data format referenced (standard G code setting)	CNC specification referenced (standard setting of whether to use a canned cycle function)		
Drilling cycle (G81)	Drilling cycle function (used)		
Drilling cycle (G81)	Drilling cycle function (used)		
Peck drilling cycle 1 (G83)	Peck drilling cycle 1 function (used)		
Peck drilling cycle 2 (G83.1)	Peck drilling cycle 2 function (used)		
None			
Tapping cycle (G84)			
Reverse tapping cycle (G84.1)			
Rigid tapping cycle (G84)			
Reverse rigid tapping cycle (G84.1)			
Boring cycle 1 (G85)	Boring cycle 1 function (used)		
Boring cycle 4 (G89)	Boring cycle 4 function (used)		
Fine boring cycle (G86.1)	Fine boring cycle function (used)		
Counter boring (G82)	Counter boring function (used)		
None			
Counter boring (G82)	Counter boring function (used)		
None			
	NC data format referenced (standard G code setting) Drilling cycle (G81) Drilling cycle (G81) Peck drilling cycle 1 (G83) Peck drilling cycle 2 (G83.1) None Tapping cycle (G84) Reverse tapping cycle (G84.1) Rigid tapping cycle (G84) Reverse rigid tapping cycle (G84.1) Boring cycle 1 (G85) Boring cycle 4 (G89) Fine boring cycle (G86.1) Counter boring (G82) None Counter boring (G82)		

• FS15 or FS15-C/Y

Machining type	NC data format referenced (standard G code setting)	CNC specification referenced (standard setting of whether to use a canned cycle function)
Center drilling	Drilling cycle (ZX plane/XY plane) (G83) Drilling cycle (YZ plane) (G87)	Drilling cycle function (used)
Drilling (normal)	Drilling cycle (ZX plane/XY plane) (G83) Drilling cycle (YZ plane) (G87)	Drilling cycle function (used)
Drilling cycle (peck drilling)	Peck drilling cycle 1 (ZX plane/XY plane) (G83) Peck drilling cycle 1 (YZ plane) (G87)	Peck drilling cycle 1 function (used)
Drilling (high-speed)	Peck drilling cycle 2 (ZX plane/XY plane) (G83) Peck drilling cycle 2 (YZ plane) (G87)	Peck drilling cycle 2 function (used)
Drilling (medium- speed)	None	
Tapping	Tapping cycle (ZX plane/XY plane) (G84) Tapping cycle (YZ plane) (G88)	
Reverse tapping	None	
Rigid tapping	Rigid tapping cycle (ZX plane/XY plane) (G84) Rigid tapping cycle (YZ plane) (G88)	
Reverse rigid tapping	None	
Reaming	Boring cycle 1 (ZX plane/XY plane) (G85) Boring cycle 1 (YZ plane) (G89)	Boring cycle 1 function (used)

• FS1618 or FS1618-C/Y

Machining type	NC data format referenced (standard G code setting)	CNC specification referenced (standard setting of whether to use a canned cycle function)
Boring(*)	Boring cycle 1 (XY plane) (G85) Boring cycle 1 (YZ plane) (G89)	Boring cycle 1 function (used)
Fine boring(*)	None	
Counterboring(*)	Drilling cycle (XY plane) (G83) Drilling cycle (YZ plane) (G87)	Drilling cycle function (used)
Circle cutting(*)	None	
Chamfering(*)	Drilling cycle (XY plane) (G83) Drilling cycle (YZ plane) (G87)	Drilling cycle function (used)
Contour chamfering(*)	None	

(8) Preprocessing before retraction

- Coolant OFF
- Spindle rotation stop
- Retract mode ON

Referenced NC data format	Data (function code) that can be output
Coolant OFF command	- Coolant OFF (M09)
Spindle rotation stop command	- Spindle rotation stop (M05)

(9) Retraction

Referenced NC data format	Data (function code) that can be output				
Retraction without compensation	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot) Reference position return (G28/G30) Pass position (Xref,Zref) 				
Command following the first retract motion block (when the number of simultaneously controlled axes is 1)	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot) Reference position return (G28/G30) Pass position (Xref,Zref) 				
Command following the second retract motion block (when the number of simultaneously controlled axes is 1)	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot) Reference position return (G28/G30) Pass position (Xref,Zref) 				
Command following a simultaneous specified-axes retract motion block (when the number of simultaneously controlled axes is 2)	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Feedrate (Fmr) -Coordinates of the current point (Xmot,Zmot) Reference position return (G28/G30) Pass position (Xref,Zref) 				
Retraction after tool-tip radius compensation cancel (movement immediately after the cancellation of tool-tip radius compensation)	 Positioning or linear interpolation (G00_G01) ABS/INCR (G90_G91) Tool-tip radius compensation cancel (G40) Feedrate (Fmr) Coordinates of the current point (Xmot,Zmot) Reference position return (G28/G30) Pass position (Xref,Zref) 				

(10) Termination of main machining

Offset cancel (only when this item is set) ٠

Referenced NC data format	Data (function code) that can be output	
Tool selection (offset cancel)	- Tool number (Tsel)	
· · · ·	- Offset number (HcomIng)	
Termination of each machining	- Line feed (EOB only)	
process		

(11) End of NC data

- Return to the home position (only when this item is set in the ٠ CNC specifications)
- Program end ٠

Referenced NC data format	Data (function code) that can be output	
Output of a block specifying	- INCR (G91) Reference position return (C28/C20)	
return to the home position	- Pass position (Xref,Zref)	
End of the main program	- Program end (M02 or M30)	

(12) Termination processing

End of an NC data file ٠

Referenced NC data format	Data (function code) that can be output
Trailer section	- % (EOR)
	- Feed (FEEDnull, FEEDsp)

17.7 Example of NC Data Output

An example of NC data output is given below.

- (1) NC data start section
 - Example of NC data output O0001 (PROGRAM-1): Program number and name
 - **CNC specification setting** Program number: "0" Total number of program number digits: 4
 - NC data format

Start of the main program: Omain, Name, EOB

• Explanation of codes

Omain : Outputs a main program number

- Name : Outputs a program name as a comment.
- EOB : Line feed code. When a command other than a line feed code is output by one function, a line is fed. When the output of a command is not desired, specify this code only.

(2) Start section of main machining

• Example of NC data output

(Machining type-1))	: Comment
M08	:	Coolant ON command
G50S4000	:	Maximum spindle speed clamp value
G96S0150M03	:	Constant surface speed control, surface
		speed, spindle rotation
T101	:	Tool change

• CNC specification settings

_	-
Coolant (liquid)	: "M08"
Coolant (mist)	: "M07"
Coordinate system	
setting	: "G50" (Maximum spindle speed clamp)
S code	: "S"
Total number of	
S code digits	: 5
Constant surface	
speed control	: "G96"

Spindle rotation		
(clockwise)	:	"M03"
T code	:	"T"
Total number of		
tool number digits	:	2
Tool number		
prefixed by 0	:	0
Total number of offset		
number digits	:	2

• NC data format

Start section of each		
machining type	:	Gclr, FFFF, Sclr, ProcName, EOB
Workpiece coordinate		
system selection	:	EOB
Coordinate system setting	:	EOB
Coolant ON command	:	M07_M08, EOB
Specification of a maximu	ım	
spindle speed clamp value	e:	G92, Smaxrpm, EOB
Constant surface speed		
control command	:	G96
Spindle rotation command	t	: Srotsur, M03, EOB
Output of a tool change		
command block	:	Tclr, Tsel, Hcomlng, EOB

• Explanation of codes

Gclr, FFFF	: Clears G code modal information. (Gclr clears
	group-by-group G code information. FFFF clears
	the G code information for all groups.)
Sclr	: Clears S code modal information. (S codes that
	have been held are cleared.)
ProcName	: Outputs a machining type as a comment.
M07_M08	: Outputs a coolant ON command.
G92	: Outputs a coordinate system setting or maximum
	spindle speed clamp value.
Smaxrpm	: Outputs a maximum spindle speed clamp value.
G96	: Outputs a constant surface speed control
	command.
Srotsur	: Outputs a spindle speed.
M03	: Outputs a spindle rotation command.
Tclr	: Clears T code modal information. (Tool numbers
	that have been held are cleared.)
Tsel	: Outputs a tool number.
Hcomlng	: Outputs an offset number.

-240-

17.8 NC Machine Setting Tool Operation

To enable the editing of an NC machine file, an NC machine setting tool is provided. The flow of NC machine file editing using the NC machine setting tool is described below.

Starting the NC machine setting tool		
Choose the NC Machine Setting Tool icon.	From the menu, select Setting => Library Data Setting => NC Machine File.	

Selecting an NC machine

From the menu, select NC Machine File => Open.

- When creating a new NC machine file, select the most-similar NC machine.

\checkmark		
Setting data		
From the menu, select Machine Setting => Machine Spec.		
From the menu, select CNC Setting => CNC Spec.		
From the menu, select CNC Setting => NC Data Format.		
 When creating a new NC machine file, the NC machine name in the machine specifications must be modified. 		

Saving the NC machine file	
From the menu, select NC Machine File => Save.	From the menu, select NC Machine File => Save As.
When creating a new NC machine file calest Save As	

. [.

When creating a new NC machine file, select Save As.

Modifying a comment file including NC data

From the menu, select NC Machine File => Open.

Click the [Edit the Comment] button.

- To create a new NC machine file, modification must be made.

``

Ending NC machining setting

From the menu, select NC Machine File => Exit.

NOTES

- Enter a character string for a setting item enclosed in double quotation marks ("). The maximum number of characters is defined for character string input.
- Enter a numeric value for a setting item that is not enclosed in double quotation marks ("). Specifiable numeric values are integers and real numbers. Restrictions are imposed on integer input for some setting items.

17.9 Customization

The NC machine setting tool is designed to enable the customization of setting items and their initial values. Customization is described below.

17.9.1 File configuration

An NC machine file consists of nine files:

- Machine setting files
 - xxx.MMF
 - xxx.MDF
- CNC setting files
 - xxx.CMF
 - xxx.CDF
 - xxx.FDF
- Machining pattern macro file
 - xxx.PMP
- Machine setting + CNC setting
 - xxx.MTF
- Prepost processor macro file
 - xxx.PMF
- Comment file
 - xxx.DOC

• CNC specification parameter file (xxx.CMF)

This file contains CNC-related data, extracted from the NC machine file referenced at NC data preparation. This file is newly created at CNC specification setting.

• CNC specification definition file (xxx.CDF)

This file contains data related to the CNC specifications. For CNC specification setting, this file is selected and is displayed in a window. This file is newly created at CNC specification setting.

• NC data format definition file (xxx.FDF)

This file contains data related to the NC data output formats. For NC data format setting, this file is selected and is displayed in a window. This file is newly created at NC data format setting.

• Machine specification parameter file (xxx.MMF)

This file contains machine-related data extracted from the NC machine file referenced at NC data preparation. This file is newly created at machine specification setting.

-242-

• Machine specification definition file (xxx.MDF)

This file defines data related to machine specifications. For machine specification setting, this file is selected and is displayed in a window. This file is newly created at machine specification setting.

• Machining pattern macro file (xxx.PMP)

This file contains machining patterns described in the prepost processor macro language. This file is newly created at machining pattern macro editing.

• NC machine parameter file (xxx.MTF)

This file is a parameter file referenced at NC data preparation. This file is created from the CNC specification parameter file, CNC specification definition file, NC data format definition file, function code definition file, machine specification parameter file, and machine specification definition file by saving the NC machine file.

• Prepost processor macro file (xxx.PMF)

This file is executed at NC data preparation. This file is created from the machining pattern macro file, prepost macro main file (PMACMAIN.PMF), prepost macro subfile (PMACSUB.PMF), and prepost macro header file (MDFILE.H) by saving the NC machine file.

17.9.2 Modifying initial values

Initial values set in parameters (machine specifications, CNC specifications, NC data formats) can be modified as necessary.

- Modify the machine specification definition file, CNC specification definition file, or NC data format definition file.
- 1) Open a file by using a text editor.

Example:

Machine specification definition file

#define	Home position (X-axis)	100.0
#define	Home position (Y-axis)	0.0
#define	Home position (Z-axis)	<u>100.0</u>
	Symbol section	Data section

2) Modify the data section of the definition file.

Example: 100.0 -> 0.0 Machine specification definition file

#define Home position (X-axis) 0.0 #define Home position (Y-axis) 0.0 #define Home position (Z-axis) 0.0

NOTES

- A character string used as data must be enclosed in double quotation marks (").
- To describe a single double quotation mark (") in a character string, specify two double quotation marks ("").
- When modifying the data section of the NC data format definition file, enter a function code symbol name or function code numeric value (4 hexadecimal characters), delimited with a comma.

17.9.3 Modifying setting item names

The names of the items to be set for parameter setting (machine specifications, CNC specifications, NC data formats) can be modified.

- Modify the machine specification definition file, CNC specification definition file, or NC data format definition file.
 - 1) Open a definition file by using a text editor.
 - 2) Modify the symbol section of the definition file.

Example: Home position (X-axis) -> Machine zero point (X-axis)

Machine specification definition file

#define Machine zero point (X-axis) 0.0

- Modify the machine specification parameter file or CNC specification parameter file.
 - 1) Modify the same symbol as that modified in the definition file.

Example: Home position (X-axis) -> Machine zero point (X-axis) Machine specification parameter file

HOME_POSI_XYZ = Machine zero point (X-axis) Home position (Y-axis), Home position (Z-axis)

NOTE

When specifying a symbol name, use only alphanumeric characters, '_', and double-byte characters.

17.9.4 Deleting setting items

Setting items for parameter setting (machine specifications, CNC specifications, NC data formats) can be deleted.

- Modify the machine specification definition file, CNC specification definition file, or NC data format definition file.
 - 1) Open a definition file by using a text editor.
 - 2) Delete a line from the definition file.

Machine specification definition file

```
#define Home position (X-axis) 0.0 \leftarrow Delete
#define Home position (Y-axis) 0.0
#define Home position (Z-axis) 0.0
```

- Modify the machine specification parameter file or CNC specification parameter file.
 - 1) Describe data directly in place of the symbol deleted from the definition file.

Example: Home position (X-axis) -> 0.0 Machine specification parameter file

HOME_POSI_XYZ = 0, 0, Home position (Y-axis), Home position (Z-axis)

17.9.5 Adding setting items

Setting items for parameter setting (machine specifications, CNC specifications, NC data formats) can be added.

- Modify the machine specification definition file, CNC specification definition file, or NC data format definition file.
 - 1) Open a definition file by using a text editor.
 - 2) Add a line to the definition file.

Machine specification definition file

#define Home position (X-axis) 100.0
#define Home position (Y-axis) 0.0
#define Home position (Z-axis) 100.0
#define Home position (α -axis) 0.0

- Modify the machine specification parameter file or CNC specification parameter file.
 - 1) Describe the symbol added to the definition file.

Machine specification parameter file

 $\label{eq:HOME_POSI_XYZ = 0.0,} \\ \mbox{Home position (Y-axis),} \\ \mbox{Home position (Z-axis),} \\ \mbox{HOME_POSI_ROT = Home position } (\alpha \mbox{-axis}) \\ \mbox{0.0,} \\ \mbox{0.0} \\ \$

NOTE

When specifying a symbol name, use only alphanumeric characters, '_', and double-byte characters.

18 FILE TRANSFER

18.1 Function Outline

The file transfer function outputs drawing data to a plotter and transmits NC data to an external device. It is also used to delete or rename files.

18.1.1 Application software

Two types of application software are provided for file transmission. They are the File Transmission Tool and the File Transmission Manager.

The File Transmission Tool is used to determine the file to be transmitted, the device to which that file is to be transmitted, and how the transmission is to be executed. These settings are made via a conversational window and registered in the File Transmission Manager schedule as a transmission job.

The File Transmission Manager executes registered jobs sequentially according to the schedule. Other applications can be executed while a file is being transmitted.

18.1.2 Devices to which data can be transmitted

Data can be transmitted to the following devices via the RS-232C serial port:

- Plotter Plotter manufactured by FANUC or any HP-GL compatible product
- Paper tape reader/punch FANUC PPR etc.
- NC unit NC unit manufactured by FANUC or any unit using an equivalent protocol
- External memory unit FANUC Handy File or PROGRAM FILE Mate
- Personal computer and peripherals
 Those conforming to the protocols described later

18.1.3 Data format that can be transmitted

Data to be transmitted to the plotter must conform to HP-GL format. Data that can be transmitted to an external device is text-format data, such as NC data, in ASCII, ISO, or EIA code.

18.1.4 Transmission protocols

Protocols for serial communication, free run, and start-stop synchronization are supported. Flow control can be achieved by using Xon/Xoff or hardware.

18.1.5 File operation functions

The following operations can be performed on those files stored on the personal computer (both on the hard disk and floppy disks) and on those files stored in an external memory unit manufactured by FANUC (Handy File or PROGRAM FILE Mate):

- Displaying a file list
- Deleting or renaming files, designating read-only files, and designating read/write permitted files
- Specifying the names and numbers of files stored on an external memory unit

18.2 Activating the File Transmission Tool

18.2.1 Activation using the Symbolic CAP T output function

Executing the plot out function or NC data output function of Symbolic CAP T automatically activates the File Transmission Tool. In this case, however, the data files that can be output are limited to those related to the figures and NC data created by Symbolic CAP T. Also, the file operations described in Section 18.1.5 cannot be applied.

The following shows an example of the plot out operation:

- ① From the menu, select File => Plot Out.
- ② Specify the range of data to be plotted out and other required settings. Then, click OK.

	Plot Out			
<u>F</u> orm Size JIS A0 1189*84 <u>A</u> dd <u>R</u> e	1 -20,12.5 mm move [Rotate A <u>x</u> i	s	OK Cancel	
Scale Line Pattern Length O Arbitrary Dashed Line: 1.0 Dash-Dotted Line:				
Position Plotting Range Divide Form	Position <u>O</u> ffset	Las <u>t</u> Position		
No Division 2 Divisions 4 Divisions)ivisions 4 Divi	sions 2 4		

③ The File Transmission Tool is automatically activated. The defaults for the file to be transmitted, the device to which the file is to be transmitted, and how the transmission is to be executed have already been set.

18.2.2 Activating only the File Transmission Tool

Double-click the File Transmission Tool icon in the Symbolic CAP T group window of the Program Manager. Any data file and device can be specified. Also, the file operations described in Section 18.1.5 can be performed.

File Transmission Tool icon:

18.2.3 Activating the File Transmission Manager

The File Transmission Manager is usually activated automatically by activating the File Transmission Tool. Once activated, keep it minimized to an icon until it is to be used.

18.3 File Transmission Tool Window

Menu Bar File Specification Area 1	File Specification Area 2
🗖 🔰 File Tran	ismission Tool 🗾 🔽 🔺
File Device Select Options Help	
Internal File	FANUC I/O Device
File <u>N</u> ame: <u>D</u> irectories:	💿 File Name 🔿 File No.
f:\capt	
00001.dat □ 00002.dat □ 01001.dat □ 01002.dat □ 02001.dat □ 05001.dat □ 05002.dat □ 07001.dat □ 07002.dat □ List Files of <u>Type</u> : Drives:	9881 #8981.DAT 888964488 9881 #8982.DAT 888964.DAT 9883 #8983.DAT 8887936 9884 #8984.DAT 8887936 9884 #8984.DAT 8887936 9884 #8984.DAT 8887936 9886 01981.DAT 88849928 9886 01981.DAT 88849928 9886 01981.DAT 88814996 9887 01982.DAT 8881496 9887 01982.DAT 8881496 9889 0682.DAT 888986488 9889 0682.DAT 888986488 9899 0682.DAT 98986488 9818 07981.DAT 88986488 9811 07982.DAT 989818496
NC Data(*.DAT) ±f: [COMPRE] (± E E E E E E E E E E E E E E E E E E E	Free Space =00776704Byte
Operation Guide Transmit Reciev	re Delete Rename Read & Write Delete All Read Only Refresh

18.3.1 Menus

The following menus are available:

• File menu

This menu is used to send and receive data files and to terminate the File Transmission Tool.

Device Select menu

This menu is used to select the device to which the file is to be transmitted, to register external devices, to specify transmission conditions, and to cancel device registration.

• Options menu

This menu is used to specify the following:

- Whether transmission results are to be checked and whether an alarm is to be issued to warn of file overwrite (when a FANUC external memory unit is being used)
- Whether a TV check is to be performed when outputting NC data to a file
- Whether the contents of the specified transmission operation are to be checked and modified before transmission
- Help menu

This menu is used to obtain information about how to use the File Transmission Tool, and also to determine its version.

18.3.2 File specification areas

The file specification areas display information relating to the device and files. These areas are used to specify the files to be transmitted or manipulated.

A list of files can be specified. File names may be specified by using wild cards.

For an explanation of the basic file specification procedure, refer to the Windows documentation.

- File specification area 1
 This area is used to specify files stored on the personal computer.
- File specification area 2

This area displays information related to the selected device. If the selected device supports file operations, the files to be manipulated can be specified in this area.

18.3.3 Buttons

The following buttons are displayed:

- Send button Starts the sending of data to the selected device.
- Receive button Starts the reception of data from the selected device.

NOTE

The display of this button may be suppressed, depending on the device being used.

- Delete button Used to delete a file.
- Clear button

Deletes all the files currently displayed in the list. In the case of the Handy File, the floppy disk is initialized.

- Rename button Used to rename a file.
- Read-only button Used to change the file attribute to read-only.
- Read/write button Used to change the file attribute to read/write enabled.
 - Refresh button
 Displays the latest list data in the file specification areas.

18.3.4 Operation guide

Provides operation guidance.

18.4 Operating Procedure

18.4.1 Selecting the device and setting the transmission conditions

First, select the device to which data is to be transmitted and set the transmission conditions.

When using the currently selected device and conditions, this step can be skipped.

For details of how to specify the transmission conditions, see section 18.6.

Select a Device		
COM1 Com1 Com1	<u>Close</u> <u>S</u> etting	
	<u>A</u> dd Remove	
	ct a Device COM1 COM1 COM1 COM1	

18.4.2 Specifying files

Specify the files to be transmitted.

If the sending device supports the specification of either the source or destination file, specify the source file.

Multiple files can be specified. Select multiple files from the list or specify a wild card in the file name.

(This step can be skipped when Symbolic CAP T has already specified the files to be transmitted.)

Selected Files				
Internal File				
File <u>N</u> ame:	Directories:			
	f:\capt			
00001.dat	[⊖ f:\			
o0002.dat	🕮 capt			
o1001.dat				
o1002.dat				
o2001.dat				
05001.dat				
03002.uat				
o7002 dat				
01002.000				
List Files of <u>Type</u> :	Dri <u>v</u> es:			
NC Data(*.DAT) 🛓	f: [COMP] 🛨			

18.4.3 Selecting a command

Click desired command button, such as Send, Receive, or Delete.

(This step can be skipped when Symbolic CAP T has already output the files.)

18.4.4 Changing the contents of the transmission operation

The contents of the transmission operation are displayed as shown below. The contents of the display can be changed as required.

Transmission			
Internal File => External File			
o1001.dat	=> 0	1001.dat	
01002.dat	=> 0	1002.dat	
02001.dat	=> 0	2001.dat	
	Discon	tinuance	
Move Bef	Discon ore	tinuance Move After	
Move Befa Specify File N	Discon ore lame	tinuance Move After Destination Directory	

(1) Discontinuing processing

To discontinue the transmission or manipulation of a file or files:

- ① Select the file or files from the list.
- 2 Click the Discontinuance button.
- (2) Move

To change the order in which the files will be transmitted:

- ① From the list, select the file to be moved.
- 2 Click the Move Before or Move After button.
- ③ Select the file before or after which the file selected in step 1 will be placed.
- (3) Specify File Name

This is used to specify either a destination file name, or the new file name when renaming a file. The destination file name is usually the same as that of the source file. (Note that the destination file name may be modified because of limitations imposed on the file name length or on the characters that can be used.) To specify a file name:

- ① Select a file from the list.
- ⁽²⁾ Click the Specify File Name button.
- ③ Enter a new file name.
- ④ Click OK.

(4) Destination Directory

This is used to specify a destination directory. All listed files are transmitted to the specified directory. To specify a destination directory:

- ① Click the Destination Directory button.
- ⁽²⁾ Enter the name of the destination directory.
- ③ Click OK.

18.4.5 Starting transmission

Click OK.

18.4.6 Checking the transmission status

When transmitting files to an internal device, the transmission status can be checked by referring to the displayed list.

When the error check option has been specified, the results of transmission are automatically displayed and error messages are output in the event of any errors occurring.

When transmitting files to an external device, the File Transmission Manager window displays the registered transmission jobs and their status. Check whether the jobs are registered in the list. Also, the job execution results can be checked.

The following shows an example of the File Transmission Manager window displaying the transmission status:



Displayed items:

- (a) Device names
- (b) File names
- (c) Amount of data transmitted
- (d) Send or receive

(e) Transmission status
 Before transmission: Blank
 After transmission has started: Start
 While data is being sent: Expected end time
 While data is being received: Expected end time
 Once transmission has ended: Complete

After checking the transmission status, click the Stop button.

_	Preparing External File		
	Stands by while Transmitting or Receiving.		
x	* When cancel button is clicked, the file list is not displayed.		
Ξ.	* And also, prior error check is skipped.		
	Cancel		

18.4.7 Ending the transmission operation

① A message appears asking whether the same files are to be transmitted to another device. Click the Yes or No button.

😑 🛛 File Transmission Tool			
Transmit to another device?			
Yes No			

- ② Select File => Exit from the File Transmission Tool menu. (This step can be skipped when the File Transmission Tool has been automatically activated by Symbolic CAP T. In this case, the File Transmission Tool automatically terminates.)
- ③ While the File Transmission Manager is performing file transmission, select Manager => Minimize. The File Transmission Manager window is minimized to an icon, but file transmission continues.

Once all files have been transmitted, select Manager => Exit.

18.5 Outline of Processing for Each Function

18.5.1	Send	
		① Select the device to which data is to be sent.
		⁽²⁾ Specify the source file in file specification area 1.
		 Click the Send button.
		④ If files can be specified for the device to which data is to be sent, specify the destination file name. When transmitting data to an internal device, a specific drive and directory can be specified so that all files will be transferred to that directory.
		⑤ Change the file transmission order or discontinue the transmission if necessary.
		6 Click OK to start transmission.
		⑦ Check the transmission status by referring to the list displayed by the
		File Transmission Manager.
		(a) Terminate the operation from the Manager menu.
18.5.2	Receive	
		① Select the device from which data is to be received.
		⁽²⁾ If files can be specified for the device from which data is to be
		received, specify the name of the file from which data is to be received. Otherwise, specify the name of the file in which the received data will be stored
		 Click the Receive button
		 When a file from which data will be received has been specified, specify the name of the file in which the received data will be stored. A specific drive and directory can be specified so that all files will be received to that directory.
		⑤ Change the file transmission order or discontinue transmission if necessary.
		6 Click OK to start transmission.
		 Check the transmission status by referring to the list displayed by the File Transmission Manager.
		(a) Terminate the operation from the Manager menu.
18.5.3	Delete	

 When the file to be deleted is stored on an external device, specify that device. When it is stored on an internal device, this step can be skipped.

② Select the file to be deleted in the file specification area.

3	Click	the	Delete	button.
---	-------	-----	--------	---------

- ④ Cancel the deletion if necessary.
- 5 Click OK to start deletion.
- 6 Check the deletion results by referring to the list displayed by the File Transmission Manager.
- ⑦ Terminate the operation from the Manager menu.

18.5.4 Clear

١
2
3
4
5
 (1) (2) (3) (4) (5)

18.5.5 Rename

- ① When the file to be renamed is stored on an external device, specify that device. When it is stored on an internal device, this step can be skipped.
- ⁽²⁾ Select the file to be renamed in the file specification area.
- ③ Click the Rename button.
- ④ Enter a new file name.
- **(5)** Cancel renaming of the file if necessary.
- ⁶ Click OK to start renaming.
- ⑦ Check the results by referring to the list displayed by the File Transmission Manager.
- (8) Terminate the operation from the Manager menu.

18.5.6 Read-only and read/write

- ① When a file whose attribute is to be changed is stored on an external device, specify that device. When it is stored on an internal device, this step can be skipped.
- ② Select the file whose attribute is to be changed in the file specification area.
- ③ Click the Read-Only or Read/Write button.
- ④ Cancel file attribute change if necessary.
- 5 Click OK to change the file attribute.
- ⑥ Check the results by referring to the list displayed by the File Transmission Manager (this is possible only for files stored on an external device).

O Terminate the operation from the Manager menu.

18.5.7 Refresh

Clicking the Refresh button displays the latest data in the file specification areas.

18.6 Setting Transmission Conditions

18.6.1 Setting procedure

This procedure is necessary whenever a device is used for the first time. Create and install a setting file for the device. Then, register the device in the list of regularly used devices.

- (1) Creating a setting file for a new device
 - ① Select the Device Select menu.
 - ② Select File, Plotter, or Other Device from the menu. When File is selected, the following steps can be skipped.

		File Transı
<u>F</u> ile	Device Select	<u>O</u> ptions <u>H</u> elp
	<u>F</u> ile	
Inte	<u>P</u> lotter	
File	<u>O</u> ther Device	ectories:
	f	:\capt

③ Click the Add button.

Select a Device
<u>S</u> etting
Add
<u>R</u> emove

④ Install one of the standard device files.

Select a device from the List of Devices, then click press the Install button.

	Setup	for Device	
Installed <u>D</u> evices: FANUC I/O Device FANUC CNC Series Paper Tape Unit	COM1 COM1 COM1	fanuc_io.cmm fanuc_nc.cmm ppr.cmm	OK Cancel Setting Remove
List of Devices:			
fanuc_io.cmm fanuc_nc.cmm pcaux.cmm plotter.cmm ppr.cmm			<u>I</u> nstall <u>H</u> elp

- Select an equivalent device from the installed device list, then click the Setting button.
- Set the communication condition.For details, see Section 18.6.4.
- ⑦ Set the transmission control.For details, see Section 18.6.5.
- (8) Set the reception control.

For details, see Section 18.6.6. (When devices is Plotter, this step can be skipped).

Save the settings.

Return to the Setting window, then click the Save As button. Enter a new name, then click OK.

Communication Condition Setting(m:\turn\pcfapttn.e\ppr.cmm)				
<u>U</u> nit Name	P <u>o</u> rt	Page		
Paper Tape Unit	COM1 👤	Transmission Control		
File can be specified		Reception Control		
Data <u>L</u> ength				
07 08	Flow <u>C</u> ontrol			
Stop bit 1 01.5 © 2 Parity © None O Even O Odd	for <u>R</u> eceiving Xon/Xoff T RS for <u>I</u> ransmitting Xon/Xoff C CS Xon : <u>0x11</u>	S(RTS) Watch T DR(DSR) CD S(CTS) Xoff : 0x93		
Baud Rate ○ 600 ○ 1200 ○ 2400 ● 4800 ○ 9600 ○ 19200	Load Save	as Cancel <u>H</u> elp		

⁽¹⁾ Install the setting file.

Select the file from the List of Devices, then click the Load button (this operation is the same as that performed in step 4).

ID From the list, select the device to be added, then click OK. Selected Device

	1		
	Setup	for Device	
Installed <u>D</u> evices:			ΟΚ
FANUC I/O Device /	COM1	fanuc io.cmm	
FANUC CNC Series Paper Tape Unit	COM1 COM1	fanuc_nc.cmm ppr.cmm	Cancel
			Setting Remove
List of Devices:			I
fanuc_io.cmm			Inotall
fanuc_nc.cmm			Instan
pcaux.cmm			
plotter.cmm			Heln
[ppr.cmm			<u></u>

(2) From the device list, select the device to be used, then click the

Close button.



- (2) Changing the device to be used
 - ① Select the Device Select menu.
 - ② Select Plotter or Other Device from the menu, according to the device type.
 - ③ From the device list, select the device to be used. Then, click the Close button.
- (3) Changing the communication condition
 - ① Select the Device Select menu.
 - ② Select Plotter or Other Device from the menu, according to the device type.
 - ③ Select the device from the list, then click the setting button.
 - ④ Change the communication condition.
 - 5 Click OK.
 - ⑥ From the device list, select the device to be used. Then, click the Close button.
- (4) Deleting a device from the list
 - ① Select the Device Select menu.
 - ② Select Plotter or Other Device from the menu, according to the device type.
 - ③ Select the device from the list, then click the Remove button.
 - ④ From the device list, select the device to be used. Then, click the Close button.

18.6.2 Specifying the device and plotter

_		Setup	for Device	
Installed	l <u>D</u> evices:			ΟΚ
FANUC FANUC Paper	I/O Device CNC Series Tape Unit	COM1 Com1 Com1	fanuc_io.cmm fanuc_nc.cmm ppr.cmm	Cancel
List of D	evices:			<u>R</u> emove
fanuc fanuc	_io.cmm _nc.cmm _cmm			<u>I</u> nstall
plott ppr.c	er.cmm mm			<u>H</u> elp

Install the setting file for the device, then select the device from the list.

• Setting button

Used to set the communication conditions, transmission control, and reception control.

Install button

Used to install the device file selected from the List of Devices. The installed device name, connected port name, and setting file name are displayed in the Installed Device list.

• Remove button

Used to delete devices from the Installed Device list.

List of Devices

This list contains the setting files that can be installed. Select the device to be installed from this list.

• Installed Device list

This list contains the installed device names. Select the device to be added to the list of devices to be used.

18.6.3 Standard setting files

The following setting files are provided as standard:

- (1) plotter.cmm
 - For the plotter
- (2) fanuc_nc.cmmFor the FANUC CNC series
- (3) fanuc_io.cmm

For FANUC external memory units (Handy File and PROGRAM FILE Mate)

(4) ppr.cmm

For the paper tape reader/punch (FANUC PPR etc.)

(5) pcaux.cmm

For the personal computer

18.6.4 Setting the communication conditions

Unit Name

Specify the name of the device to be connected. A device name can consist of up to 16 alphanumeric characters.

- File can be specified Check this item if a FANUC external memory unit (Handy File or PROGRAM FILE Mate) is to be used.
- Port Specify the name of the port through which connection is to be made.
 - Data Length

Specify the number of bits contained in a one-word data packet.

- Stop bit Specify a space between characters.
 - Parity

Specify a bit that is attached to each data packet to enable the detection of transmission errors.

Baud Rate

Specify the transmission speed at which data will be sent to the connected device.

• Flow Control for Receiving

Specify the method of flow control to be used for data reception. This control temporarily suspends data transmission from the connected device if the receive buffer becomes full.

• Flow Control for Transmitting

Specify the method of flow control to be used for data transmission. This control monitors the connected device to determine whether it is capable of receiving data and, if necessary, temporarily suspends data transmission to the device.

Xon

Specify the code used to request the transmission of data. This code is either a hexadecimal code ("0x" plus two digits) or a decimal code (consisting only of digits).

• Xoff

Specify the code used to suspend the transmission of data. This code is either a hexadecimal code ("0x" plus two digits) or a decimal code (consisting only of digits).

Load

Specify this item to load communication conditions from a previously saved file.

• Save As

Specify this item to save the currently specified communication conditions to a file.

- Transmission Control Specify this item to switch to the page on which the conditions for data transmission can be set.
- Reception Control Specify this item to switch to the page on which the conditions for data reception can be set.

18.6.5 Setting the transmission control conditions

• Data Code

Select the character code.

EOB Code

Specify the code indicating the end of a transmitted data block (line). When specified with a word, this code is either a hexadecimal code ("0x" plus two digits), a decimal code (consisting only of digits), or a character string (consisting only of characters).

- TV Check Specify this item to insert a blank code for TV check into a data block.
- Feed

Specifying this item causes a feed mark to be output to the paper tape when the mark is encountered in the transmitted file. The feed mark can be added to the file when NC data is created.

• EOF

Specifying this item causes an EOF code to be output when encountered in the transmitted file.

• Transfer interval

Transfer data is temporarily stored in an area called a buffer. Specify the intervals at which the file transfer manager sends the transfer data into a Windows transmission buffer. When no value is specified, 300 ms is assumed. Usually, this value need not be changed. If the remote unit cannot receive data at a lower baud rate, increase the transfer interval. Alternatively, decrease the buffer size, described below. Increasing the transfer interval reduces the transfer rate.

• Buffer size

Specify the size of the transmission buffer described above. When no value is specified, the system assumes 1024 bytes. Usually, this value need not be changed. If the remote unit cannot receive data at a lower baud rate, reduce the buffer size. Alternatively, increase the transfer interval, which is described above. Reducing the buffer size reduces the transfer rate.

Timeout

Specify the maximum period that is allowed to elapse before the receiving device, for which data reception is currently suspended, becomes ready for receiving data again.

Preprocessing Data

Set the commands to be used before data transmission. A command consists of a combination of a hexadecimal code ("0x" plus twodigits), a decimal code (consisting only of digits), and a character string (consisting only of characters), each delimited by a comma (,).

Postprocessing Data

Set the commands to be used after data transmission. A command consists of a combination of a hexadecimal code ("0x" plus twodigits), a decimal code (consisting only of digits), and a character string (consisting only of characters), each delimited by a comma (,).

- Reception Control Specify this item to switch to the page on which the conditions for data reception can be set.
- Communication Condition Specify this item to switch to the page for saving communication conditions to a file.

18.6.6 Setting reception control conditions

• Data Code

Select the character code.

Starting Code

Specify Effective as the Starting Code when data is not to be written to the file. Specify Write to enable writing of the starting code to the file. This code consists of a combination of a hexadecimal code ("0x" plus two digits), a decimal code (consisting only of digits), and a character string (consisting only of characters), each delimited by a comma (,).

Ending Code

Specify Effective as the Ending Code when data reception is to be stopped upon the reception of the ending code. Otherwise, data reception is not stopped until a timeout occurs. Specify Write to enable writing of the ending code to the fill. This code consists of a combination of a hexadecimal code ("0x" plus two digits), a decimal code (consisting only of digits), and a character string (consisting only of characters), each delimited by a comma (,).

EOB Code

Specify the code indicating the end of a transmitted data block (line). When specified with a word, this code is either a hexadecimal code ("0x" plus two digits), a decimal code (consisting only of digits), or a character string (consisting only of characters).

• TV Check

Specify this item to detect TV check errors.

• Timeout

Specify the maximum period that is allowed to elapse while waiting to receive data.

• Preprocessing Data

Set the commands to be used before data reception. A command consists of a combination of a hexadecimal code ("0x" plus twodigits), a decimal code (consisting only of digits), and a character string (consisting only of characters), each delimited by a comma (,).

• Postprocessing Data

Set the commands to be used after data reception. A command consists of a combination of a hexadecimal code ("0x" plus two digits), a deci-mal code (consisting only of digits), and a character string (consisting only of characters), each delimited by a comma (,).

- Transmission Control Specify this item to switch to the page on which the conditions for data transmission can be set.
- Communication Condition Specify this item to switch to the page for saving communication conditions to a file.

18.7 Transmission Schedule Management

The File Transmission Manager supports functions for changing the transmission sequence specified in the schedule for discontinuing data transmission.

18.7.1 Contents of the window

🗕 🗾 File Transmission Manager 💽 🗖					-
<u>M</u> anager	<u>Setup H</u> el	Р			
<u>P</u> ause	<u>R</u> esume	<u>D</u> elete	Up	Down	
P <u>o</u> rt:	COM1	<u></u>	ror: 🗌 Pau	nse	
Status: Worl	king D Device o	0001.dat	OB	Send Starting	
FANUC I/	D Device o	0002.dat	ØB	Send	
FANUC I/	D Device o	2001.dat	0B	Send	
FANUC I/	D Device o	5001.dat	ØB	Send	
FANUC I/) Device o	5002.dat	ØB	Send	

- Manager menu
 Used to terminate the File Transmission Manager.
- Setup menu
 Used to create a transmission condition setting file.
- Port box

Used to select the port for which a transmission schedule is to be displayed.

On Error check box

Used to specify whether the next job is to be executed or whether transmission is to be temporarily suspended upon the occurrence of a transmission error.

Status list

Displays the transmission schedule in which registered jobs are listed in the specified sequence, along with their current status. To change the schedule, change the positions of the pertinent jobs in this list.

• Pause button

Used to temporarily suspend the execution of a job. Data transmission can also be suspended.

Resume button

Used to resume the execution of a job that was previously suspended using the Pause button.
- Delete button Discontinuance a specified job from the schedule.
 - Up button Raises the position of the specified job in the scheduled sequence.
- Down button Lowers the position of the specified job in the scheduled sequence.

18.7.2 Changing the schedule

- Suspending transmission
 Clicking the Pause button temporarily suspends transmission.
- Resuming transmission Clicking the Resume button resumes the transmission that was previously suspended.
- Specifying the job to be changed Select a job from the status list.

💳 File Transmission Manager					•	
<u>M</u> anager <u>S</u> etup <u>H</u> elp						
<u>P</u> ause	<u>R</u> esume	Delete	Up	Down		
P <u>o</u> rt:	COM1	<u></u> ± 0 <u>n</u> Er	ror: 🗌 Pa	use		
Status: Wor	king					
FANUC I/	O Device	00001.dat	ØB	Send Sta	arting	
FANUC I/	O Device	00002.dat	ØB	Send		
FANUC I/	O Device	o2001.dat 🧃	GB	Send		
FANUC I/	O Device	o5001.dat 🖊	ØB	Send		
FANUC I/	O Device	o5002.dat/	ØB	Send		

Selected Job

• Discontinuing transmission

To discontinue transmission, select the associated job then click the Delete button.

• Changing the scheduled sequence

To change the job execution sequence specified in the schedule:

- ① Select the job for which the schedule is to be changed.
- To raise the job's position in the sequence, click Up.
 To lower the job's position in the sequence, click Down.

18.7.3 Exiting the File Transmission Manager

To exit the File Transmission Manager, first select the Manager menu, then select either of the following:

• Minimize

Selecting Minimize allows another application to be executed while continuing transmission according to the current schedule. The File Transmission Manager is reduced to an icon.

• Exit (Discontinuance of the job) Selecting Exit cancels the current schedule and forcibly terminates transmission.

18.8 Example Transmission Procedures

18.8.1 Transmission to and from the CNC

- (1) Reception using the FANUC online cable
 - Set the CNC transmission conditions. The specified conditions must be the same as those specified for the paper tape reader (PPR).
 - ② Select the transmission device and transmission conditions by using the File Transmission Tool.
 - ③ Start the transmission (reception) of data by using the File Transmission Tool.
 - ④ Refer to the Status List to verify that transmission has started.
 - Start transmission (punching) within the specified time (one minute) by using the CNC.
 - 6 Check the transmission results in the Status List.
- (2) Transmission using the FANUC online cable
 - ① Set the CNC transmission conditions. The specified conditions must be the same as those specified for the paper tape reader (PPR).
 - ② Select the transmission device and transmission conditions by using the File Transmission Tool.
 - ③ Start the transmission (reading) of data by using the CNC.
 - ④ Start transmission by using the File Transmission Tool.
 - ^⑤ Check the transmission results in the Status List.
- (3) Transmission or reception using the cross cable
 - ① Set the CNC transmission conditions. The specified conditions must be the same as those specified for the paper tape reader (PPR).
 - ② Select the transmission device and transmission conditions by using the File Transmission Tool.
 - ③ Start the transmission or reception of data by using the File Transmission Tool.
 - ④ Check the Status List to verify that transmission or reception has started.
 - Start transmission within the specified time (one minute) by using the CNC.
 - 6 Check the transmission results in the Status List.

19 PLOTTING

This section explains how to output a machining figure and tool path which have been created.

19.1 Setting up a Plotter

Setup the plotter before starting plotting.

Selecting the plotter setting menu item
 Select File => Setup Plotter... from the menu.

(2) Setting communication conditions

Specify where the origin of the plotter will be when a form is loaded. Select either the lower left corner or the center of the form, depending on the plotter being used. For details, refer to the manual supplied with the plotter.

(3) Shift

The origin of the plotter may not correspond exactly to the lower left corner or the center of the form, as selected above. Set any difference as a shift.



(4) Pens

Specify pen speeds for all the pens mounted on the plotter. The units in which the pen speed is specified varies with the plotter being used. Refer to the manual supplied with the plotter.

(5) Minimum unit

Set the amount the plotter will actually move for a specified move amount of 1. If the minimum unit supported by the plotter is 0.025 mm and the units of the figure dimensions are mm, specify 0.025.

(6) Clipping function ON/OFF

If the plotter supports a command (IW command) that enables the specification of the drawing area, place a check on the clipping function to limit plotting to within the range selected at plot out.

(7) Ending setup

Click OK to complete the setup.

19.2 Procedures

Select File => Plot Out from the menu bar.

(1) Selecting a form size

Select a form size from the displayed list, or type in the form size data. When the form size data is typed in, the entered data can be registered in the list by clicking the Addition button. To delete any unnecessary data from the list, select the data from the list and click the Erase button. Form size data must be entered according to the following format.

Form standard name Form size name Width*Height -dx, dy Unit_

The form standard name, form size name, and units can be specified using any characters other than an asterisk "*." The form size must be specified in the same units as the figure(s) to be output. dx represents the horizontal margin. dy represents the vertical margin.



(2) Specifying Rotate Axis

Place a check on Rotate Axis to rotate the drawing direction through 90 degrees.



(3) Setting the scale factor

When auto scale is selected, the figure(s) within the plotting range are plotted in the drawable area of the form, the maximum scale factor being applied. When free scale is selected, type in a scale factor.

(4) Setting a line pattern length

Enter the lengths of the line patterns, using the same units as those used to specify the form size.



(5) Setting form division

When No division is selected, the figure in the plotting range is plotted using the entire form.

When 2 Divisions is selected, the figure in the plotting range is plotted within the specified division of the two form divisions.

When 4 Divisions is selected, the figure in the plotting range is plotted within the specified division of the four form divisions.



(6) Specifying the plotting range

Click the Last Position button to output a plot in the same range as the previous plotting range (for example, when repeating the output of the most recently plotted figure). Click the Plotting Range button to specify a different plotting range. Specify a plotting range within the drawing window by means of dragging.



(7) Position offset

When a new plotting range is specified, the plotting position is initially specified as follows:

When the origin of the plotter is at the lower left corner of the form, the lower left position of the drawing area corresponds to the lower left position of the plotting area.

When the origin of the plotter is at the center of the form, the center of the drawing area corresponds to the center of the plotting area.

Press the Position Offset button to adjust the plotting position. In the drawing window, the plotting range is indicated by a dotted line, while the form position is indicated by a broken line. Drag the form to adjust its position.



(8) Plot out

Clicking OK causes the File Transmission Tool window to appear. For details of the related procedures, see the chapter on file transmission. The file transmission tools can transform data into files or output data to the plotter.

20 CAD DATA INPUT/OUTPUT

20.1 CAD Data Input/Output

The following subsections provide a general description of CAD data input/output.

20.1.1 CAD data

Usually, a CAD/CAM system cannot read the internal data of a dissimilar system directly, nor write data to such a system.



To overcome this problem, several CAD data formats have been placed in the public domain. (Data in such a CAD data format is referred to as CAD data in this manual.)

Data can be transferred between dissimilar systems, therefore, by converting that data to CAD data.

Typical CAD data formats include IGES and DXF. This system allows DXF-format CAD data to be input and output.



20.1.2 Handling of CAD data by this system

This system can input and output general CAD data.

(1) Input of CAD data

Drawing data created using another CAD system can thus be loaded into this system.

This data can be handled in the same way as figure entities created using symbolic CAP T, and be used for the creation of part figures and others.





Drawing data created using another CAD system can be used for machining as is, eliminating the need to create figure entities and thus reducing the lead time.

(2) Output of CAD data

Figure entities created using this system can be output as data for use in another CAD system.

Parts figures, tool paths, and so on, can also be output as figure entities.



NOTE

In CAD data input/output, data may not be converted completely. Any entities that cannot be converted perfectly are converted to the closest approximation.

20.2 DXF File Input/Output

The following subsections explain DXF file input/output.

20.2.1 Overview of DXF

DXF(Drawing Interchange Format) is a data format used for drawing data conversion.

Usually, the extension ".dxf" is appended to a DXF file name.

Data in a DXF file consists of the following sections:

a) HEADER section

Contains general information relating to the drawing.

b) <u>TABLES section</u>

Contains the contents of each of the tables listed below:

- Line type table (LTYPE)
- Drawing layer table (LAYER)
- Character style table (STYLE)
- View table (VIEW)
- User coordinate system table (UCS)
- Viewport setting table (VPORT)
- Dimension style table (DIMSTYLE)
- Application identification table (APPID)

c) <u>BLOCKS section</u>

Contains the data constituting each block of the drawing.

d) **ENTITIES section**

Contains drawing data, including instructions to reference block definitions.

e) <u>END OF FILE</u>

4	0[Beginning of the HEADER section]
	SECTION
	2
	HEADER
	9
	<the are="" entries="" header="" here.="" in="" listed="" the="" variable=""></the>
a	•
	•
	•
	\$VISRETAIN
	70
	0
•	ENDSEC[End of the HEADER section]
4	0 [Beginning of the TABLES section]
	SECTION
	2
	TABLE
	2
	LTYPE
	70
	(Maximum number of entries in the LTYPE(line type) table) <the are="" entries="" here.="" in="" listed="" ltype(line="" table="" the="" type)=""></the>
b	0
	ENDTAB
	0
	LAYER
	(Maximum number of entries in the LAYER(drawing layer) table) <the are="" entries="" here.="" in="" layer(drawing="" layer)="" listed="" table="" the=""></the>
	STYLE,VIEW,UCS,VPORT,DIMSTYLE,APPID
	• 0
	ENDTAB
	0
,	'ENDSEC [End of the TABLES section]

	† 0	····· [Beginning of the BLOCKS section]
	SECTION	
	2	
	BLOCKS	
	<block definit<="" th=""><th>ion data is listed here.></th></block>	ion data is listed here.>
	0	
с	BLOCK	
	8	
	0	
	◆ENDSEC	····· [End of the BLOCKS section]
	◆ENDSEC ↑ ⁰	····· [End of the BLOCKS section] ····· [Beginning of the ENTITIES section]
	ENDSEC 0 SECTION	····· [End of the BLOCKS section] ····· [Beginning of the ENTITIES section]
	ENDSEC 0 SECTION 2	<pre>[End of the BLOCKS section][Beginning of the ENTITIES section]</pre>
	ENDSEC 0 SECTION 2 ENTITIES	<pre> [End of the BLOCKS section] [Beginning of the ENTITIES section]</pre>
ď	ENDSEC 0 SECTION 2 ENTITIES <drawing data<="" th=""><th>••••• [End of the BLOCKS section] ••••• [Beginning of the ENTITIES section]</th></drawing>	••••• [End of the BLOCKS section] ••••• [Beginning of the ENTITIES section]
d	ENDSEC 0 SECTION 2 ENTITIES <drawing data<="" th=""><th>••••• [End of the BLOCKS section] ••••• [Beginning of the ENTITIES section]</th></drawing>	••••• [End of the BLOCKS section] ••••• [Beginning of the ENTITIES section]
d.	ENDSEC 0 SECTION 2 ENTITIES <drawing data<="" th=""><th>•••••• [End of the BLOCKS section] ••••• [Beginning of the ENTITIES section]</th></drawing>	•••••• [End of the BLOCKS section] ••••• [Beginning of the ENTITIES section]
d	ENDSEC 0 SECTION 2 ENTITIES <drawing data<br="">0</drawing>	•••••• [End of the BLOCKS section] ••••• [Beginning of the ENTITIES section]
d,	ENDSEC 0 SECTION 2 ENTITIES <drawing data<br="">0 ENDSEC</drawing>	<pre> [End of the BLOCKS section] [Beginning of the ENTITIES section] [End of the ENTITIES section]</pre>
d e	ENDSEC 0 SECTION 2 ENTITIES <drawing data<br="">0 ENDSEC 0</drawing>	<pre> [End of the BLOCKS section] [Beginning of the ENTITIES section] [End of the ENTITIES section]</pre>

20.2.2 DXF file input

The following illustrates the flow of the DXF file input procedure.



(1) Start of DXF file input

Select File => Exchange CAD data => DXF-Input from the menu.

(2) DXF file selection

Select the DXF file to be input. Clicking Cancel cancels the input procedure and returns the system to normal operation.

(3) Selection of either input or setting

Specify whether to start input or set the conversion method.

Input button Starts input.

Cancel button Cancels input and returns the system to normal operation.

Setting... button Displays the window used to set the conversion method. Using this window, set the conversion method to be used for input.

Help button Displays help messages relating to input.

(4) Setting the conversion method

When the Setting... button is clicked in 3, the window used to set the conversion method appears. For details of the items to be set, see the next subsection.

(5) Input processing

During input processing, a gauge indicating the progress of the processing is displayed.

To cancel the processing, click the Cancel button. This returns the system to normal operation.

(6) Display of input results

When input processing terminates normally, the number of entities converted, number of layers, the time required for the conversion, and other information are displayed.

20.2.3 Setting the input conversion method

The conversion method applied to input can be set. The following explains the items that can be set.

(1) New input/Additional input

Specify whether the figure database should be initialized before starting input of the DXF file.

New input: Initializes the figure database before reading the file. This is equivalent to inputting the DXF file after selecting File => New from the menu.

Additional input: Does not initialize the figure database before reading the file.

The figures defined in the DXF file are read and added to the current figure database.



[Additional input]



(2) Input/output coordinate system

When inputting a DXF file, the coordinate system to be used for the entities can be selected.

[World coordinates]

The x, y, and z coordinate values of the DXF file are input, these being assumed to be the coordinate values in the figure database.

[Coordinates of current window]

The x, y, and z coordinate values of the DXF file are input, these being assumed to be the coordinate values in the coordinate system displayed on the window when input starts.

This option is used to read the image of the DXF file directly onto the window.

(3) Setting a group name

A name can be assigned to a group of settings for conversion methods 4 through 7, as explained below.



Heavy-line entity conversion method (Layer conversion method 0 Line type conversion method 0 Color conversion method 0

A group of settings for a previously used conversion method can be easily reused by assigning a name to the group.

It is convenient if the names assigned to the conversion method setting groups reflect the system that outputs the DXF files to be converted, or the intended use of the data to be converted.

(4) Heavy-line entity (TRACE) conversion method

The conversion method applied to heavy-line entities defined in the DXF file can be specified.

Because heavy-line entities are not supported by this system, they are converted by means of either of the following methods:

-285-

[Center line]

Converts heavy lines to center lines.



(The figure indicated by the broken line is for explanation purposes only. It is not included in the actual data.)

[Outline]

Converts heavy lines to polygons representing outlines.



(5) Layer

The conversion method applied to the layers in which entities exist can be specified.

NOTE

Symbolic CAP T does not support multiple layers. It is therefore recommended that Multiple layers to one layer be selected and layer number 1 be used.

[Multiple layers to one layer]
 [Layer no.]

[Layers to layer] [Start layer no.] [Return to start layer]

[Remain at last layer]

[Multiple layers to one layer]

Makes entities defined in different layers of the DXF file correspond to one layer in this system.



[Layer no.]

Specifies the number of the layer where the converted entities will be placed.

[Layers to layer]

Makes the entities defined in different layers of the DXF file correspond to different layers in this system.



[Start layer no.]

Specifies the number of the first of the series of layers in which the converted entities will be placed.

When Layers to layer is selected, either of the following can be specified as the method to be applied once the last layer has been used.

[Return to start layer]

When the last layer number of this system is reached, subsequent layers are made to correspond to the layers of this system starting from the start layer again.



[Remain at last layer]

When the last layer number of this system is reached, all remaining layers are made to correspond to the last layer.



(6) Line type

Specifies the line type of this system to which each line type specified in the DXF file should be converted.

A line type can be specified for each of the main 25 line types that can be specified in a DXF file, as well as for other line types.

(7) Color

Specifies the color number of this system to which each color number specified in the DXF file should be converted.

A color number can be specified for each of color numbers 1 through 255 that can be specified in a DXF file.

20.2.4 DXF file output

The following illustrates the flow of the DXF file output procedure.



(1) Start of DXF file output

From the menu, select File => Exchange CAD data => DXF-Output.

(2) Output file name specification

Specify the name of the DXF file to be output. Clicking the Cancel cancels the output procedure and returns the system to normal operation.

(3) Selection of either output or setting

Select either the start of output or setting of the conversion method.

Output button Starts output.

Cancel button Cancels output and returns this system to normal operation.

Setting... button Displays the window used to set the conversion method. In this window, set the conversion method to be applied to output.

Help button Displays help messages related to output.

(4) Setting of the conversion method

When the Setting... button is clicked in 3, the window used to set the conversion method appears. For details of the items to be set, see the next subsection.

(5) Output processing

During output processing, a gauge indicating the progress of the output processing is displayed.

To cancel the processing, click the Cancel button. This returns this system to normal operation.

(6) Display of conversion results

When the output processing terminates normally, the number of entities converted, the number of layers, the time required for the conversion, and other information are displayed.

20.2.5 Setting the output conversion method

The conversion method applied to output can be set. The following explains the items that can be set.

(1) Comment

A comment can be added to the DXF file.

Enter the creator's name, company and position, and a comment if required.

A comment, if specified, has no effect on the DXF figure data.

(2) Output coordinate system

The coordinate system used for the entities to be output to the DXF file can be selected.

[World coordinates]

The coordinate values registered in the figure database are output directly as three-dimensional data to the figure entity DXF file.

[Coordinates of current window]

The figure entities are output to the DXF file as coordinate values in the coordinate system displayed on the window when output starts. As a result, the figure entities are output to the DXF file as planar entities.



(3) Setting a group name

A name can be assigned to a group of settings for conversion methods 4 through 7, as explained below.



A group of settings for a previously used conversion method can be easily reused by assigning a name to the group.

It is convenient if the names assigned to the conversion method setting groups reflect the system to which the converted DXF files are transferred, or the intended use of the data to be converted.

(4) Line approximation for ellipse

When Coordinate of current window is selected as the coordinate system used to output figure entities, circle and arc entities, viewed slantwise, are output to the DXF file as ellipses and parts of ellipses. Because elliptic entities are not included among the entity types that can be defined in a DXF file, this system approximates ellipses with fine line segments prior to outputting them.

[Tolerance of line approximation for ellipse]

The allowable tolerance when an ellipse is approximated with fine line segments is referred to as the line segment approximation tolerance. The ellipse is divided into fine line segments, such that the distances between the ellipse and the line segments do not exceed this tolerance.



(5) Output layer designation

Specifies whether entities existing in each layer should be output to the DXF file.

The following three layers can be designated.

[Drawing layer] Layer in which the figure entities created with the figure definition exist

[Tool path layer] This layer is not used with Symbolic CAP T.

[NC path layer] Layer in which the tool paths drawn with NC Data Preparation exist

For each layer, one of the following output methods can be specified:

[Output all]

The entities existing in the specified layer are output irrespective of whether the layer is visible or invisible.

[Not output]

The entities existing in the specified layer are not output, irrespective of whether the layer is visible or invisible.

[Visible entities only]

Whether entities are to be output is determined according to the current state of their layer. That is, those entities existing in a layer are output provided the layer is visible. They are not output if the layer is invisible.

(6) Line type

Specifies the line type that can be specified in the DXF file to which each line type specified in this system should be converted.

-293-

Each of the seven line types supported by this system can be converted to any of the main 25 line types that can be defined in a DXF file.

(7) Color

Specifies the color number that can be specified in the DXF file, to which each of the 16 color types used by this system is to be converted.

Each of the colors used by this system can be converted to any of color numbers 1 through 255 that can be defined in a DXF file.

20.3 Handling CAD Data Efficiently

When data created using another CAD system is to be loaded into this system, note the following to ensure the optimum efficiency of the subsequent tasks.

Creating data in another CAD system

(1) Save entities into separate layers according to their purpose.

By saving figure entities in one layer and other entities (such as dimension lines, drawing frames, and comments) into other layers, that data to be used for machining can be extracted into this system more easily.

(2) Delete hatching.

Hatching requires a very large number of entities, adversely affecting the speed of entity search, redrawing, etc. Delete as much hatching as possible.

(3) Output high-precision CAD data.

Low-precision CAD data produces coordinate value errors when converted.

To create data having the best precision possible, output CAD data using as many digits (as high a precision) as possible.

Operating procedure after CAD data has been loaded into the system

(1) Extract only that data that is required for machining by deleting all other entities.

This system supports layers and groups.

There are several methods of deleting unnecessary entities from the converted data, such as the deletion of entities from one layer at a time, from one group at a time, and deleting one entity at a time.

(2) For low-precision CAD data

Problems may occur due to errors that arise when using converted data.

When entity gap errors occur, any problems may be handled by changing the allowable entity gap tolerance.

If this does not solve the current problems, or if it gives rise to other problems, entities must be corrected or redefined.

20.4 Conversion Specifications for DXF File Input/Output

20.4.1 DXF input conversion specifications

The conversion specifications for DXF file input are explained on a sectionby-section basis.

(1) HEADER section

This system does not reference this section.

(2) TABLES section

DXF		Handling by
Table entry	Description	Symbolic CAP T
LTYPE	Line type	Ignored
LAYER	Drawing layer	Referenced during conversion
STYLE	Character style	Ignored
VIEW	View	Ignored
UCS	User coordinate system	Ignored
VPORT	Viewport	Ignored
DIMSTYLE	Dimension style	Ignored
APPID	Application identification	Ignored

If the LAYER table entry cannot be found (or if the TABLES section does not exist), layer information cannot be obtained and, therefore, converted entities are allocated to one layer; Layers to layer is ignored if specified for the input conversion method.

(3) BLOCKS section

When the blocks written in the ENTITIES section are to be converted, they are converted using the data in the BLOCKS section.

(4) ENTITIES section

Figure data appears in the ENTITIES and BLOCKS sections. In both of these sections, figure data is represented in the same way. The following table lists the correspondence between the figure data before conversion and that after conversion.

Entity type		Description	Representation in Symbolic CAP T	
LINE	Line segment		Line segment entity	
POINT	Point		Point entity	
CIRCLE	Circle		Circle entity	
ARC	Arc		Arc entity	
TRACE	Heavy li	ine	Polygon representing the outline; or center line	
SOLID	Painted	figure	Polygon representing the outline	
TEXT	Charact	er	Not supported.	
SHAPE	User-de	fined character	Not supported.	
BLOCK	Block		Group entity	
ENDBLK	End of b	block	Group entity	
INSERT	Block insert		Group entity	
ATTDEF	Character attribute definition		Not supported.	
ATTRIB	Character attribute		Not supported.	
PLINE	Polyline		Polygonal line/polygon	
VERTEX	Polyline vertex			
	Details	Straight line	Straight-line entity in polyline/ polygon	
		Arc	Arc entity in polyline/polygon	
		Fit curve	Not supported.	
		Spline curve	Approximation with polyline/	
		Three-dimensional polygon mesh	Not supported.	
SEQND	End of polvline		Polyline/polygon	
3DFACE	Three-dimensional face		Polygon representing the outline	
VIEWPORT	Viewpor	rt	Not supported.	
DIMENSION	Dimension figure		Not supported.	

20.4.2 DXF output conversion specifications

The conversion specifications for DXF file output are explained on a section-by-section basis.

(1) HEADER section

Group code	Description	Entry in Symbolic CAP T	
\$ACADVER	Version	Fixed to "AC1009," indicating GX-5, as	
		the version of the AutoCAD drawing	
		database	
\$CLAYER	Drawing layer	Number of the currently effective	
	number	drawing layer	
\$ELEVATION	Drawing plane	The current drawing plane height is	
	height	output. Fixed to 0 if the output coor-	
		dinate system is Coordinate of	
		current window.	
\$UCSORG	Drawing	The origin of the current drawing	
	coordinate	coordinate system is output.	
	system origin	Fixed to "0, 0, 0" if the output coordinate	
		system is Coordinate of current window.	
\$UCSXDIR	X axis of the	The direction of the X axis of the current	
	drawing	drawing coordinate system is output.	
	coordinate	Fixed to "1, 0, 0" if the output coordinate	
	system	system is Coordinate of current window.	
\$UCSYDIR	Y axis of the	The direction of the Y axis of the current	
	drawing	drawing coordinate system is output.	
	coordinate	Fixed to "0, 1, 0" if the output coordinate	
	system	system is Coordinate of current window.	
999	Comment	Output system name "PCFAPT" and any	
		comment specified by the user in the	
		setting window are output.	

(2) TABLES section

Information relating to the current layers is output.

For the drawing layers, the numbers of the layers currently being used (a layer number being generated for each layer name) and the display status of the layers (visible or invisible) are output.

No information is output for those layers for which output is not specified.

Other table entries (LTYPE, STYLE, VIEW, VPORT, and DIMSTYLE) are not output.

(3) **BLOCKS section**

Groups and machining figures, as well as any entities which they contain, are output, together with hierarchical structures.

Groups are output in the format that uses the syntax of the BLOCK and INSERT of DXF.

The group IDs used internally by this system are used for BLOCK names. If any group names exist, they are output as comments.

Machining figures are output as normal group entities, machining information being removed before the figures are output.

The entities and attributes that constitute groups and machining figures are the same as those supported by the ENTITIES section.

(4) ENTITIES section

Figure data appears in the ENTITIES and BLOCKS sections. In both of these sections, figure data is represented in the same way. The following table lists the correspondence between the figure data before conversion and that after conversion.

Entity in	Entity type	Description	
Symbolic CAP I			
Point	POINT	Point	
Line segment	LINE	Line segment	
Circle	CIRCLE	Circle (if the normal vector of the circle is	
		not horizontal to the Z axis of the world	
		coordinate system, the projection direction	
		(210, 220, 230) is also output.)	
		If Coordinate of current window is	
		selected and the normal vector is not	
		horizontal to the Z axis of the coordinate	
		system, the circle is approximated with	
		fine line segments.	

Entity in		Entity type	Description
Symb	olic CAP T		
Arc		ARC	Arc (if the normal vector of the arc is
			not horizontal to the Z axis of the
			world coordinate system, the
			projection direction (210, 220, 230) is
			also output.)
			If Coordinate of current window is
			selected and the normal vector is not
			horizontal to the Z axis of the
			coordinate system, the arc is
			approximated with fine line segments.
Group entity,		BLOCK	Block definition and block call
Blank figure,		INSERT	
Parts fi	gure		
Polyline/polygon			3D polyline and the definitions of its
			vertexes
Details	All	POLYLINE	2D polyline and the definitions of its
	constituent	VERTEX	vertexes
	entities are		
	line		
	segments.		
	All	POLYLINE	Block definition and block call
	constituent	VERTEX	
	entities are		
	on the same		
	plane.		
	Other than	BLOCK	
	the above	INSERT	
Fitting curve		POLYLINE	2D polyline and the definitions of its
		VERTEX	vertexes

Each of these entities is output with the following attributes:

- Line type
- Color
- Layer to which the entity belongs

21 FAPT LANGUAGE FIGURE INPUT

21.1 Overview

The FAPT language figure input function analyzes and executes the figure definition statements in part programs written in FAPT, converts them to figure elements to be saved to Symbolic CAP figure databases, and inputs the conversion results.

Figure elements input from a part program can be handled in exactly the same way as those created using the figure creation function.



21.1.1 Figure types that can be input

The following table lists the figure definition statements that can be input. Each figure definition statement is converted to the corresponding figure element on the figure database.

FAPT language figure definition statement	Figure database
Point definition statement (Pi)	Point
Straight line definition statement (Si)	Straight line
Circle definition statement (Ci)	Circle
Curve definition statement (CVi)	Polyline/polygon
Multiple points definition statement (Gi)	Multiple points
Fitting curve definition statement (Ti)	Fitting curve

21.1.2 FAPT statement types that can be input

In addition to the figure definition statements, the following FAPT statements can be written in part programs.

- Variable definition statement (Vi)
- Macro instruction statements (MAC and MEND)
- Branch instruction statements (JUMP, IF, and statement number)
- Definition data-related statements (DEL and CHECK)

21.2 Operation

21.2.1 Start-up

Select the CAD data exchange-FAPT-input command from the File menu.

-			F	ANUC S	Symbo	olic C	APT - (Un	titl
-	<u>F</u> ile	<u>M</u> odify	<u>E</u> dit	⊻iew	<u>W</u> in	dow	<u>A</u> uxiliary	ŝ
R	<u>N</u> ev	/		Ctr	I+N	1		
کا	<u>О</u> ре	:n		Ctr	l+0	<u> </u>		
	<u>S</u> av	e		Ctr	l+S			
	Sav	e <u>A</u> s						
	<u>C</u> lea	ar Machin	ing Da	ita				
	<u>E</u> xc	hange CA	D Data	ì		DXF	⁼ - <u>I</u> nput	
	<u>P</u> lot	Out				DXF	⁼ - <u>O</u> utput	
	Plot	ter Set <u>u</u> p				<u>F</u> AP	T - Input	
	E <u>×</u> it	1						

21.2.2 Part program selection

Select the file name of a part program to be input.

	Open File	
File <u>N</u> ame: *.fpt # # # # #	Directories: f:\capte033 Print f:\ Print capte033	Cancel
List Files of <u>T</u> ype: FAPT Files(*.FPT)	Dri <u>v</u> es: Ef: compressed	±

Clicking the OK button opens the FAPT language figure input dialog box. Clicking the Cancel button terminates the FAPT language figure input function.

21.2.3 FAPT language figure input

Specify how figure data is input from a selected part program file.

😑 FAPT Language Figure Input
Method for Input New Additional
<u>C</u> oordinate System for Input O World Coordinate System © Coordinate System Displayed in Drawing Window
Input Cancel Setting Help

• Method for Input

Selects either of the following two figure database input modes:

- New Inputs figures after the figure database is initialized.
- Additional Inputs figures without initializing the figure database.
- Coordinate System for Input Selects either of the following two coordinate systems for use at input.
 - World Coordinate System

Inputs h and v coordinates in FAPT statements as x and y coordinates in the figure database, respectively.

- Coordinate System Displayed in Drawing Window
 - Inputs h and v coordinates in FAPT statements as x and y coordinates in the display coordinate system (view coordinate system) of the drawing window that has focus when input begins.

• Input button

Starts inputting a part program. If New has been selected, a message appears to prompt you to specify whether you want to have the database initialized.



Selecting Yes starts to execute the part program. Selecting No brings you back to the FAPT language figure input dialog box.

Cancel button

Terminates the FAPT language figure input function completely.

- Setting button Opens the Set FAPT language figure input dialog box.
 - Help button Opens help for the FAPT language figure input function.

21.2.4 Settings related to the FAPT language figure input function

This function lets you make settings for inputting FAPT language figures.

Setting of FAPT Language Figure Input	
■ Programming Coordinate Axes	ОК
Layer Number: 1	Cancel
Input Plane Depth: 0	Help
Drawing Color and Line Type	

Programming Coordinate Axes

Selects a programming coordinate system for a part program to be input.
• Layer number

Selects a layer number to which a figure element is to belong after it is input.

• Input Plane Depth

Specifies the height of an input surface for a FAPT language figure as a third-axis value (default value is 0.0); the FAPT language figure data is two-dimensional.

Drawing Color

Specifies the color of a figure element to be used after it is input. All figure types are drawn in the specified color except for the CV curve, for which the color specified using the PCOLOR statement is valid.

- Line type Specifies the line type of a figure element to be used after it is input. All figure types are drawn with the specified line type.
- OK button

Saves the settings and brings you back to the FAPT language figure input dialog box.

- Cancel button Cancels the settings and brings you back to the FAPT language figure input dialog box.
- Help button

Opens help for Settings related to the FAPT language figure input function.

21.2.5 Displaying the progress of input processing

This function opens the following box during input processing of FAPT language figures to display the progress of processing.

Exec	uting of FAPT Language Figure In
	0 line is executing.
0%	100%
	Stop

• Stop button

Stops the FAPT language figure input processing. The figures input before the Stop button is clicked are not canceled.

21.2.6 Displaying input information

This function opens the following box when the FAPT language figure input processing ends (either normally or abnormally).

😑 Result of FAPT Language Figure Input		
Input process was completed.		
Number of Error: 0 Number of Warning: 2		
Processing Time: 00 min. 03 sec.		
ΟΚ		

• Details button

Opens the FAPT input list box to display input results.

😑 Notepad - "FPTFIG.LOG 🔽	
<u>F</u> ile <u>E</u> dit <u>S</u> earch <u>H</u> elp	
▶ ** Start of Part Program ***	+
File Name: F:\FPT\TEST01.FPT 5/9/97 1:37:46	
Start Date:8/18/97 16:32:24	
00001 P1=0,0 (P1= 0.000000 0.00000) 00002 P2=0,100	
(P 2 = 0.000000 100.000000)	
00003 P(100,100)	
EXEC-W 299: In the FAPT Language Figure Inpu (line 3), "* MAIN *"	
00004 P(-100,-100)EXEC-W 299: In the FAPT	
(line 4), "* MAIN *"	
End Date:8/18/97 16:32:24	+
•	

The input list contains the following information:

- Part program
- Error information
- Alarm information
- Figure information (variable, point, straight line, and circle)
- Part program file name and file information
- Input start and end time
- OK button

Terminates the FAPT language figure input function.

-306-

21.3 Cautions and Restrictions

- A straight line is drawn after it is clipped according to the setting of Figure Operating Area in the Figure Operation setting dialog box. This setting dialog box is displayed by selecting Setting => Parameter Setting => Figure Operation from the menu.
- A part program created in the P-G/P-H system cannot be used in Symbolic CAP T, because its format is different. (Format conversion can be made using Handy files.)
- In the P-G/P-H system, statements not specified in its manual may execute with no error detected. These statements (not conforming to the FAPT language syntax) may be detected as errors by Symbolic CAP T, because the latter makes stricter syntax checks.
- Compatibility between the P-G/P-H system and Symbolic CAP T is maintained as much as possible. Some statements are not compatible, however.
- Figure inputs cannot be undo/redo.

22 DIMENSION MEASUREMENT

22.1 Dimension Measurement Function

This function measures coordinates and distances related to figures. It can

- Blank figures created by symbolic input method
- Figures entered using CAD
- Tool paths
- Tool paths created by NC data preparation

22.2 Measurable Dimensional Values

The dimension measurement function can measure the dimensional values listed below. The measurements can be displayed as required.

Coordinates

- Entity start/end point
- Intersection of two entities (neighborhood to the intersection is selected)
- Intersection of two entities (two entities are selected)
- Midpoint of an entity
- Center of a circle/arc
- Existing point
- Mouse pointer position

Distances

- Distance between two points (that is measured using the coordinates mentioned above)
- Distance between a point and entity
- Radius of a circle/arc

Angles

• Angle formed by two lines

NOTE

- 1. The entities must previously be defined.
- 2. Measurements are saved to a part file.

22.3 Activating Dimension Measurement

To start dimension measurement, select Auxiliary => Measurement from the menu bar. Detailed descriptions of functions appear. Select the desired function from the submenu.

PT	-	<u>F</u> ile	<u>M</u> odify	<u>E</u> dit	⊻iew	<u>A</u> uxiliary	<u>S</u> etting	<u>H</u> elp
		3 🖬	5	3 X	÷	<u>D</u> rawing	Format	
i						<u>E</u> ntity Inf	ormation	hutac
<u>.</u>						Set <u>C</u> urre	ent Attribu	tes
	<u>C</u> oo	ordinat	e Values			Mea <u>s</u> ure	ment	►
	Dis Die	tance j tance	<u>b</u> etween] of Point a	Two Po nd Enti	ints it/	<u>L</u> ayer		Þ
	<u>R</u> ac Ang	lius of le of T	Circle/Ar	C	ity			
	 Dis	play M	leasured	<u>I</u> nform	ation			

Coordinate Values:

Measures the coordinates explained in Section 22.2.

Distance between Two Points:

Measures the distance between two points.

Distance of Point and Entity:

Measures the shortest distance between a point and a line segment or arc.

Radius Circle/Arc:

Measures the radius of a circle/arc.

Angle of Two Line:

Measure the angle formed two lines

Display Measured Information:

Displays measured dimensions.

22.4 Measuring Coordinates

When Auxiliary => Measurement => Coordinate Values is selected from the menu bar, the prompt area changes as follows:



First select which coordinate type to measure.



Upon completion of selection, picking an entity whose coordinates you want to obtain causes measured coordinates to be displayed in the lower right section of the window.

DX: 200.000000 Mach. Plane Coord. ± Clo	Measured Information				
	se	200.000000	DX:		
Z: 100.000000 Diameter ±		100.000000	z:		

In the drawing window, the obtained coordinates are indicated with the mark: \Box

Coordinates can be measured continuously. To obtain coordinates of the same type as before, pick entities continuously. To obtain coordinates of different types, first select a type, then pick an entity.

NOTE

It is possible to select the coordinate system of coordinates to be displayed and the type of an X-coordinate. The selected types become valid when measurements are to be displayed as described in Section 22.9.

To switch between coordinate systems, select the desired coordinate system from the following combo box.

Mach. Plane Coord.	ŧ	
Mach. Plane Coord.		
World Coordinate		
	_	

To switch between X-coordinate types, select the desired Xcoordinate type from the following combo box.

Diameter	±
Diameter	
Radius	

22.5 Distance between Two Points

When Auxiliary => Measurement => Distance between Two Points is selected from the menu bar, the prompt area changes as follows:



Similarly to coordinate measurement, select the desired point type, then pick the point where measurement is to start. You will be prompted to specify the position where measurement is to end. Pick the desired end position, and the distance between the two points will be displayed in the lower right section of the window.



In the drawing window, the distance between the two points is drawn as: \leftrightarrow If the type of the specified start position is different from that of the specified end position, after picking the first position, select the type of the end position.

It is possible to measure distances continuously.



[Example of a distance between the intersection of two entities and a circle center]



22.6

When Auxiliary => Measurement => Distance of Point and Entity is selected from the menu bar, the prompt area changes as follows:

	Input point	
Eec.	2D 🛨 📝 📈	<u>~ * * * ⊗ × × ×</u> >
Esc	<u>Z</u> : <u>DX</u> :	

point where measurement is to start. The prompt area changes as follows:

	Select entity
Esc	

Pick the desired entity. The shortest distance between that entity and the point picked before will be displayed in the lower right section of the window.

Measured Information				
D:	100.000000	Close		

In the drawing window, the distance between the two points is drawn as: It is possible to measure distances continuously.



22.7 Circle/Arc Radius

When Auxiliary => Measurement => Radius of Circle/Arc is selected from the menu bar, the prompt area changes as follows:

	Select circle/arc
Esc	

Pick a circle/arc entity whose radius you want to obtain, and the measured circle/arc radius will be displayed.



In the drawing window, the measured radius is drawn as: \rightarrow It is possible to obtain radiuses continuously.



[Obtaining a circle radius]

[Obtaining an arc radius: The arc center and the point picked are connected.]

22.8 Angles Formed by Two Lines

When Auxiliary => Measurement => Angle of Two Lines is selected from the menu bar, the prompt area changes as follows:

	Select 1'st line	Angle
		Small 🛨
E	sc	

Pick one of entities that form the angle to be measured, and you will be prompted to specify the other entity. Pick the second entity, and the measured angle will be displayed in the lower right section of the window.

Measured Information		
A:	45.000000	Close

In the drawing window, the measured angle is drawn as: \leftrightarrow Drawing of the angle begins at the first entity to be picked. It is possible to measure angles continuously.

NOTE
It is possible to select the type of an angle to be measured.
To switch between angle types, select the desired one from the
following combo box.
Angle
Small 🛨
Small
Large

22.9 Displaying Measurements

When Auxiliary => Measurement => Display Measured Information is selected from the menu bar, the prompt area changes as follows:

	Select measured entity
Esc	

When an entity for which dimension measurement is finished is picked, the information about a figure that contains the picked entity is displayed.

[If a coordinate entity is picked]

Measured Information		
DX:		Mach. Plane Coord. 🛨 <u>C</u> lose
Z: 100.0000	100.000000	Diameter 👱

NOTE

It is possible to switch the coordinate system of coordinates to be displayed and the type of an X-coordinate.

[If a distance entity is picked]

Measured Information		
D:	100.000000	Close

[If a radius entity is picked]

- Measured Information		
R:	20.00000	Close

[If an angle entity is picked]



22.10 Cautions

The following cautions should be observed in measuring dimensions.

- (1) The double-head arrow \leftrightarrow indicating a distance varies as the figure is enlarged/reduced.
- (2) The heads of the double-head arrow ↔ may become invisible depending on the display method used.
- (3) The attributes (color and line type) of entities whose dimensions are to be measured are become same as current attributes.

23 STANDARD SETTING

This chapter explains how to set the initial values displayed on each conversational window, displayed immediately after Symbolic CAP T starts.

23.1 Standard Setting for Blank Figure and Parts Figure

Set the initial values to be displayed on the conversational window used for entering blank input data and symbolic figures.

From the menu, select Setting => Standard Setting => Blank Figure, Parts Figure.

The Standard Setting conversational window appears. Change the values as desired and click OK.

The following initial values can be set:

Blank Figure

.

Select one value from among the listed blank figures.

- Color/Line of Blank Set the color and line type/width of the blank figure to be displayed after the blank data input conversational window is cleared.
- Standard Surf Rough Select a value from among the listed surface roughness values.
 - Part Position for Path Select the part position for the path that will be used when a symbolic figure is input.

23.2 Standard Setting for NC Data Preparation

Set the initial values to be displayed on the conversational window used for NC data preparation.

From the menu, select Setting => Standard Setting => NC Data Preparation. The Standard Setting conversational window appears. Change the values as desired then click OK.

The following initial values can be set:

- Program No.
- Program Name
- Output Mode

Select either Absolute or Incremental.

• Sequence No.

Set sequence number output to ON or OFF and set the initial sequence number and increment for each machining process or for each block.

• Sub program Set items for output in sub program format.

24 PARAMETER SETTING

This chapter explains how to set the line colors and types used when Symbolic CAP T is active.

24.1 Parameter setting for View

Make the required settings for the view of coordinate systems and others. From the menu, select Setting => Parameter Setting => View. The Setting conversational window appears. Change the values as desired and click OK.

The following data items can be set:

- Machining Plane Coordinate System Set the display of the axis mark, origin mark, and drawing of the drawing coordinate system to ON or OFF, and set the display range.
- World Coordinate System Set the display of the axis mark and the origin mark of the world coordinate system to ON or OFF.
- Grid

Set the display of the grid to ON or OFF; and set the base point position, space, and number of mark spaces.

Cursor Tracking

Set the display of cursor tracking to ON or OFF; and set the display position update timing and the display coordinate system.

24.2 Parameter Setting for Figure Operation

Make the required settings for figure operation. From the menu, select Setting => Parameter Setting => Figure Operation. The Setting conversational window appears. Change the values as desired and click OK.

The following data items can be set:

- Figure Operating Area Set the figure operating area with world coordinate values.
- Figure Calculating Range Specify whether, when creating a new figure by referencing another figure, the effective range of the referenced figure should be processed using the actual sizes of its entities or their extended sizes.
- Plural Possibilities of Figure Creation Specify whether, when there are multiple figure creation possibilities, the system should automatically select one according to the selected position of the referenced figure, or display all possibilities so that the user can select one.
- Display Attributes for Plural Possibilities of Figure Creation Set the color and line type/width used to display multiple possibilities.
- Range of the Straight Line Set the range of a line segment used when a straight line is created.
- Range of the Arc Set the range of an arc used when an arc is created.
- Grouping of Plural Points Specify whether multiple points should be created as a group or as individual points.
- Polyline/Polygon Created by Trace Mode Specify whether, when creating a polyline/polygon by tracing, the polyline/polygon should be created at the same position as the referenced figure, or whether it should be projected onto the drawing.

• Group Name

Specify whether, when creating multiple points as a group, the system should automatically assign a group name to the points orprompt the user to assign one.

- Assemble Polyline/Polygon Set the maximum allowable tolerance to be applied when the system attempts to determine whether entities are continuous when assembling polylines/polygons.
- Round/Outer Round/Chamfer (Object Entities) Specify whether, when inserting a corner R/chamber entity between two entities, the two entities should be extended or trimmed to the points where they meet the inserted entity, or be left as is.
- Round/Outer Round/Chamfer (Change Every Corners at a Time) Set the maximum angle for corners to be inserted when inserting corner Rs/chambers into polylines/polygons as a batch.
- Offset (Interfering Entities) Specify whether to remove an interfered element if interference occurs during offset.
- Offset (Length to Corner Cut Position) Set the factor used to determine the distance to the corner cut position when a corner is cut.

24.3 Parameter Setting for Overall Setting of Symbolic CAP T

Specify the units and the drawing format to be referenced when calculating machining conditions.

From the menu, select Setting => Parameter Setting => Overall Setting for Symbolic CAP T.

The Setting conversational window appears. Change the values as desired and click OK.

The following data items can be set:

- Unit Type Select either MM or INCH.
- Drawing format
 Select a drawing format from among the four supported formats.
- Text Editor Type

The path of text editor which is used to edit created NC data etc. is specified.

The part which is replaced with object file is described as % file%. If path has any space character, path is enclosed in double quotes. (Example 1) notepad % file% (Example 2) "C:\Program Files\Hidemaru\hidemaru.exe" % file%

Tool Path Editing Function

Select wheather Tool path editing function is used or not.

24.4 Parameter Setting for Blank Figure and Part Figure

Set the color and line type/width for a part figure.

From the menu, select Setting => Parameter Setting => Blank Figure, Parts Figure.

The Setting conversational window appears. Change the values as desired and click OK.

The following data items can be set:

Color and Line

Specify the colors and line types/widths for a part figure, grooving, threading, necking, and an undefined figure.

• Coefficient of a Thread

Specify the thread coefficient used to determine the thread depth.

24.5 Parameter Setting for Pre-Machining Setting

Set, to ON or OFF, the prompts for Home position, etc., displayed on the Setting of Pre-machining conversational window.

From the menu, select Setting => Parameter Setting => Setting of Premachining.

The Setting conversational window appears. Change the values as desired and click OK.

The following data items can be set:

- Prompt for Home position (ON or OFF)
- Prompt for Index position (ON or OFF)

24.6 Parameter Setting for Machining Definition

The length of arrow which indicate an area division point displayed in machining definition for Lathe is set.

When Setting => Parameter Setting => Machining Definition is selected from the menu bar, dialog box for setting appears. It is possible to set the following data by modifying values and pushing OK button.

 Length of arrow which indicate an area division point The length of white and green arrows which specify cutting area is set. The default value is 10.0. If blank figure or parts figure is too big or small, the value should be modified.

24.7 Parameter Setting for Machining Condition Automatic Decision

When the machining condition is decided in machining definition automatically, the reference value for automatic decision is set.

The dialog box of Parameter Setting of Machining Condition Automatic Decision is displayed by the following operation.

- 1. From the Setting menu, select Parameter Setting.
- 2. From the cascading menu, choose Auto Process Determination.

The reference value for an automatic decision of following machining type can be set.

Machining Types:

- Common in All Machining
- Center Drilling
- Drilling
- Tapping
- Reaming
- Common in Turning
- Turning : Rough (Outer/Face)
- Turning : Semi Finish (Outer/Face)
- Turning : Finish (Outer/Face)
- Turning: Rough (Inner)
- Turning : Semi Finish (Inner)
- Turning : Finish (Inner)
- Grooving : Rough (Outer/Face)
- Grooving : Rough (Inner)
- Grooving : Finish (Outer/Face)
- Grooving : Finish (Inner)
- Threading : Outer
- Threading : Front
- Threading : Inner

WARNING

Before programming using the conversational function, ensure that correct data is set in each item of the conversational window. Failure to set correct data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

(1) Common to all machining types

- Maximum spindle speed Set the maximum rotation speed of the spindle.
- Tool number

Set the initial value and increment for automatically determining the tool numbers.

Compensation number

Set the initial value and increment for automatically determining the compensation numbers.

(2) Center drilling

• Clearance

Set the clearance in the direction of the tool axis.

• Maximum hole diameter

A tool having a diameter which does not exceed the maximum hole diameter is selected during automatic tool selection.

(3) Drilling

Clearance

Set the clearance in the direction of the tool axis, as well as the secondary clearance.

• Maximum hole diameter

A tool having a diameter which does not exceed the maximum hole diameter is selected during automatic tool selection.

• Return escape amount

Set the return escape amount for peck drilling.

Dwell

Set whether the dwell is specified in units of seconds or revolutions. Also set the initial dwell.

• Remaining thickness allowing drilling

The automatic determination of drilling processes is not performed if the remaining cutting allowance is less than the specified remaining thickness.

• Finishing allowance

A tool is automatically selected according to the hole diameter with the specified finishing allowance considered.

(4) Tapping

• Clearance

Set the clearance in the direction of the tool axis.

Dwell

Set whether the dwell is specified in units of seconds or revolutions. Also set the initial dwell.

(5) Reaming

• Clearance

Set the clearance in the direction of the tool axis.

• Dwell

Set whether the dwell is specified in units of seconds or revolutions. Also set the initial dwell.

(6) Common to all turning types

- Cutting edge protection angle Set the cutting edge protection angle for the tool.
 - Wall upward cutting method Select judge according to the angle between the cutting edge and wall, always cut up, or never cut up
- Angle between the cutting edge and wall for wall upward cutting Set the threshold angle used to determine whether the wall is to be traced.
- Pocket

Set whether a pocket portion of the parts figure is to be machined.

Overhanging

Set whether an overhanging portion of the parts figure is to be machined.

(7) Turning: Roughing (outer or end face)

- Clearance
 - Set the clearances in the directions of the Z- and X-axes.
- Return escape amount

Set the return escape amounts in the directions of the Z- and X-axes.

• First override value Set the override value to be used for the first cut.

(8) Turning: Semifinish machining (outer or end face)

- Clearance
 - Set the clearances in the directions of the Z- and X-axes.
- Return escape amount
 Set the return escape amounts in the directions of the Z- and X-axes.

(9) Turning: Finishing (outer or end face)

- Clearance
 Set the clearances in the directions of the Z- and X-axes.
- Return escape amount
 Set the return escape amounts in the directions of the Z- and Xaxes.

(10) Turning: Roughing (inner)

- Clearance Set the clearances in the directions of the Z- and X-axes.
 - Return escape amount Set the return escape amounts in the directions of the Z- and Xaxes.

(11) Turning: Semifinish machining (inner)

- Clearance Set the clearances in the directions of the Z- and X-axes.
- Return escape amount
 Set the return escape amounts in the directions of the Z- and Xaxes.

(12) Turning: Finishing (inner)

• Clearance

Set the clearances in the directions of the Z- and X-axes.

Return escape amount Set the return escape amounts in the directions of the Z- and Xaxes.

(13) Grooving: Roughing (outer or end face)

Clearance

Set the clearance in the direction of the tool axis.

• Return escape amount

Set the return escape amount when the tool moves away from the groove wall.

Pecking return escape amount

Set the return escape amount in the direction of the tool axis.

Dwell

Set whether the dwell is specified in units of seconds or revolutions. Also set the initial dwell.

(14) Grooving: Roughing (inner)

- Clearance Set the clearance in the direction of the tool axis.
- Return escape amount Set the groove return escape amount when the tool moves away

from the wall.

• Pecking return escape amount

Set the return escape amount in the direction of the tool axis.

Dwell

Set whether the dwell is specified in units of seconds or revolutions. Also set the initial dwell.

(15) Grooving: Finishing (outer or end face)

• Clearance

Set the clearance in the direction of the tool axis.

• Return escape amount

Set the return escape amount when the tool moves away from the groove wall.

• Dwell

Set whether the dwell is specified in units of seconds or revolutions. Also set the initial dwell.

(16) Grooving: Finishing (inner)

• Clearance

Set the clearance in the direction of the tool axis.

• Return escape amount

Set the return escape amount when the tool moves away from the groove wall.

• Dwell

Set whether the dwell is specified in units of seconds or revolutions. Also set the initial dwell.

(17) Threading: Outer

- Cutting method
 - Set the cutting method for threading.
- Clearance

Set the clearances in the directions of the surface, entry, and exit.

(18) Threading: Front

• Cutting method

Set the cutting method used for threading.

Clearance

Set the clearances in the directions of the surface, entry, and exit.

(19) Threading: Inner

- Cutting method Set the cutting method for threading.
- Clearance

Set the clearances in the directions of the surface, entry, and exit.

24.8 Parameter Setting for Tool Path Creation

Set the approach/escape method, etc. From the menu, select Setting => Parameter Setting => Tool Path Creation. The setting dialog box appears. Change the values as desired, then click OK.

The following data items can be set:

- Tool compensation cancel (ON or OFF)
- Constant surface speed effective/not effective (except for hole machining)
 - Approach/escape method Specify whether approach/escape should be performed for one axis or two axes at a time.

WARNING

•

Before programming using the conversational function, ensure that correct data is set in each item of the conversational window. Failure to set correct data may result in the tool colliding with the workpiece and/or machine, possibly causing damage to the machine, workpiece, and/or tool itself, or injury to the user.

24.9 Parameter Setting for NC Data Preparation

Set the colors and line types/widths for the tool paths used during NC data preparation.

From the menu, select Setting \Rightarrow Parameter Setting \Rightarrow NC Data Preparation. The setting dialog box appears. Change the values as desired, then click OK.

The following data items can be set:

- Color and line type/width of tool paths (for cutting feed and rapid traverse)
- Execution list output item
- Graphic item
- NC data file extension
- NC data file overwrite check (ON or OFF)
- Comment characters for each machining type for NC data preparation

For each machining type, set the comment characters to be output at the beginning of each process in the NC data.

NOTE

When the C/Y-axis module is installed, comment text for each machining type can also be set for the C- and Y-axes.

24.10 Parameter Setting for Auto Saving

Set auto saving of a part file to ON or OFF, and also set the saving interval, etc.

From the menu, select Setting => Parameter Setting => Auto Saving.

The Setting conversational window appears. Change the values as desired and click OK.

The following data items can be set:

- Auto saving (ON or OFF)
- Auto saving interval
- Prompt for Auto Saving (ON or OFF)
- Save file name

25 TOOL BAR SETTING

25.1 Tool Bar Setting

Select those menu functions that are frequently used and register them as a tool bar icon. This saves time and effort when selecting menu items.

25.1.1 Storing the tool bar definition file

The contents of the tool bar are saved in the tool bar definition file. Make a copy of this file to guard against it being destroyed.

Normally, the tool bar definition file is registered with file name 'CAPT.TBT' in the same directory as that in which Symbolic CAP T is installed. Make copy of this file by using the File Manager, and store it in a safe place.

25.1.2 Creating icons

When registering a new function from the menu bar, if an icon for the function is not provided as standard, create a suitable icon by using the Windows Icon editor.

25.1.3 Displaying the Tool Bar Setting window

From the menu, select Setting => Tool Bar Setting to display the Tool Bar Setting window.

When displayed, the Tool Bar Setting window displays the contents of the tool bar and, at the same time, this system enters a special operating state:

- Selecting a function from the side menu results in an error (functions available from the side menu cannot be registered in the tool bar.)
- Selecting a function on the menu bar then clicking the tool bar is always interpreted as the addition of that function to the tool bar.
- The prompt area cannot be selected.

NOTE

If Drawing Window is selected by mistake, the Tool Bar Setting window can be made visible again by minimizing the drawing window.

25.1.4 Registering menu items

Register a menu item by means of the following procedure:

• On the Tool Bar Setting window, select the number with which the selected menu function is to be registered.

25.1.5 Deleting buttons/spaces

Delete a button by means of the following procedure:

- On the Tool Bar Setting window, select the number of the button/space to be deleted. More than one number can be selected at one time.
- Click the Erase button.

The selected button is deleted. The buttons subsequent to the deleted button are moved up, new numbers being assigned to each.

25.1.6 Inserting spaces

Insert a space by means of the following procedure:

- On the Tool Bar Setting window, select the number of the position where a space is to be inserted.
- Click the Insert Space button.

A space is inserted at the position having the specified number. The menu items subsequent to that position are assigned numbers equal to their previous numbers + 1.

25.1.7 Changing icons

Change an icon by means of the following procedure:

- On the Tool Bar Setting window, select the number of the button for which the icon is to be changed.
- Click the Change Icon button.
- A list of icons is displayed. Specify the file corresponding to the new icon to be registered.

Refer to the help feature for details of the procedure for selecting an icon. Icon files (files having a .ICO extension) and bitmap files (files having a .BMP extension) can be used for this purpose. These files can be created with the Windows PaintBrush application.

25.1.8 Saving settings/returning to the normal state

To save the new settings and return to the normal operating state, click the End button.

Clicking the End button causes the following processing to be performed:

- The new settings are saved to the tool bar definition file.
- The tool bar is redisplayed with the new settings.
- The Tool Bar Setting window is cleared and the system returns to its normal operating state.

25.1.9 Discarding the settings/returning to the normal state

To discard the new settings and return to the normal operating state, click the Cancel button.

Clicking the Cancel button causes the new settings to be discarded, the Tool Bar Setting window to be cleared, and the system to return to its normal operating state.

26 TROUBLESHOOTING

26.1 Before Trouble Occurs

Trouble may occur when you are preparing data. So, you should take the following measures to protect the data from being lost because of any possible trouble.

• Automatic saving of data being prepared Automatic saving of data being prepared is enabled by following the procedure below.

This automatic saving is enabled by default. The following procedure can be used when it is necessary to change the file name and update interval.

- From the menu, select Setting => Parameter Setting => Auto Saving.
- (2) Set the amount of standard setting to Saving.
- (3) Enter the update interval and extended time.
- (4) Enter the name of the file to be saved.
- (5) Click OK.
- Executing the save command Save the data using the save command at the end of each operation.
 - (1) From the menu, select File \Rightarrow Save.
 - (2) If the file is going to be saved for the first time, you will be prompted to specify the file name. Enter the file name when requested. If the file has been saved at least once, the data will be saved immediately after step (1).

26.2 If Trouble Occurs

This section describes what should be done to recover a normal operation quickly.

- Saving the current data Try to save data by following the procedure below.
 If it is impossible to execute the procedure, go to the next step.
 - (1) Press the ALT--TAB keys several times to activate Symbolic CAP T again.
 - (2) Press the ESC key several times to make menus selectable.
 - (3) From the menu, select File \Rightarrow Save.
- Restarting Symbolic CAP T
 - (1) Exit Symbolic CAP T.
 - (2) Terminate all applications (if running).
 - (3) Terminate Windows.
 - (4) Start Windows.
 - (5) Start Symbolic CAP T.

If it is impossible to exit Symbolic CAP T or Windows, press the CTRL-ALT-DEL keys to reset the system.

- Restoring data from a file saved previously
 - (1) From the menu, select File \Rightarrow Open.
 - (2) Enter the name of the file previously saved using the save command or automatic saving.

26.3 Error Messages

If an error or warning occurs, the window as shown below appears and describe what the error or warning is like.

The title of the error/warning window is the abbreviation of each function. CAPSYMT at the top of the window is an abbreviation. W 0102 is the type and code of the error/warning. W stands for warning, and E for error.

Abbreviation of a function in which the error or warning occurred



[C/Y]

26.3.1 Abbreviations of functions

The abbreviation of each function is indicated below. For details, see the description of each group of messages. A message including [C/Y] is output only with the C/Y-axis module.

-C-	
CAPBLANK	Messages concerning blank figure creation (Section 26.3.2)
CAPSYMT	Messages concerning part figure creation
	(Section 26.3.2)
CHCKANMD	Messages concerning chuck data setting for
	animeted simulation (Section 26.2.2)
	annualed sinulation (Section 20.5.5)
F	
	Manager and an in a fully submatic manager
FPAUIU	Messages concerning fully automatic process
	determination (Section 26.3.5) [C/Y]
-M-	
MACHPLAN	Messages concerning machining surface setting
	(Section 26.3.4)
MPROC	Messages concerning process specification
	(Section 26.3.5) [C/Y]
-N-	
NCOUT	Messages concerning NC data creation (Section
	26.3.6)
- R -	
RGMILLIB	Messages concerning the milling library
	(Section 26.3.7) [C/Y]
RGSTCOM	Messages concerning part file reading (Section
	2.3.8)
RGSTCOND	Messages concerning cutting condition setting
	(Section 26.3.9)
RGSTTOOL	Messages concerning tool/tooling data setting
	(Section 26.3.10)
-S-	
SYMGVPTN	Messages concerning groove figure creation
	(Section 26.3.2) <i>IC/Y1</i>
SYMHLPTN	Messages concerning hole figure creation

(Section 26.3.2)

-340-
SYMMLPTN	Messages concerning rectangular	/circular figure
	creation (Section 26.3.2)	[C/Y]
–No abbreviation–		
Messages output durin	ng setup (Section 26.3.11 (1))	
Messages concerning	figure selection for modification (S	lection 26.3.11
(2))18.		[C/Y]
Messages concerning	machining profile selection (Section 2	6.3.11 (3))
		[C/Y]

26.3.2 Messages related to blank and parts figure creation

[Abbreviation CAPBLANK] ----- Messages related to blank figure creation

Abbreviation-number	Message (Explanation)
[Error]	
CAPBLANK-E0101	Cannot delete blank figure.
CAPBLANK-E0102	Data is not put to memory.
[Warning]	
CAPBLANK-W0004	Data is empty(Outer,Length,Base-point).
CAPBLANK-W0005	Data is empty(Inner).
CAPBLANK-W0007	Data is not correct.
CAPBLANK-W0008	Machining data is defined for a modified blank figure. [C/Y]
	The machining data may become unusable due to a machining profile
	modification. So, be sure to check the machining data before creating NC
	data.

[Abbreviation CAPSYMT] ----- Messages related to parts figure creation

Abbreviation-number	Message (Explanation)
[Error]	
CAPSYMT-E0001	Parameter-error is occured.
CAPSYMT-E0002	System-error is occured.
CAPSYMT-E0009	No supported error is occured.
CAPSYMT-E0010	Data is end.
CAPSYMT-E0011	Element is over maximum number.
CAPSYMT-E0012	Already figure data is exist.
CAPSYMT-E0013	Figure data is not found.
CAPSYMT-E0014	C.F data is not found.
CAPSYMT-E0015	PCF data is not found.
CAPSYMT-E0016	Element data is not found.
CAPSYMT-E0017	PEL data is not found.
CAPSYMT-E0018	Additional attribute data is not found.
CAPSYMT-E0019	Data is not found.

CAPSYMT-E0020	Data is using now.	
CAPSYMT-E0021	Reference figure data is not found.	
CAPSYMT-E0062	Not necessary modifier is specified.	
CAPSYMT-E0105	Duplicated open error is occuerd.	
CAPSYMT-E0106	Modified open error is occuered for not existed figure.	
CAPSYMT-E0107	Added open error is occuered for existed figure.	
CAPSYMT-E0110	Next element is not found at current position.	
CAPSYMT-E0111	Start-point of element is not exist.	
CAPSYMT-E0112	Error information is not exist.	
CAPSYMT-E0200	GP(ID) is not deleted.	
CAPSYMT-E0201	GP(ID) is not opened.	
CAPSYMT-E0202	GP(ID) is not copyed.	
CAPSYMT-E0203	GP(ID) is not registered.	
CAPSYMT-E0204	GP(ID) is not written.	
CAPSYMT-E0205	Data is not read.	
CAPSYMT-E0206	End process is not normal.	
CAPSYMT-E0230	PPTN function error	
CAPSYMT-E0500	Command argument pointer is NULL.	
CAPSYMT-E0501	Symbolic figure number is illegal.	
CAPSYMT-E0502	Symbolic figure is not exist.	
CAPSYMT-E0503	Symbolic figure is not created at new open.	
CAPSYMT-E0504	Symbolic figure is not modified at open.	
CAPSYMT-E0506	It is impossible to move pointer(PGD) to start-point.	
CAPSYMT-E0507	It is impossible to move pointer(PGD) to next element.	
CAPSYMT-E0508	It is impossible to move pointer(PGD) to previous element.	
CAPSYMT-E0509	It is impossible to move pointer(PGD) to previous/next element.	
CAPSYMT-E0510	It is impossible to set data of parts position.	
CAPSYMT-E0511	Element table data is not got.	
CAPSYMT-E0512	Error is occured in alarm diagnosis.	
CAPSYMT-E0513	It is impossible for alarm diagnosis.	
CAPSYMT-E0514	Error code is occured in the reference to figure.	
CAPSYMT-E0515	It is impossible to update element in the reference to figure.	
CAPSYMT-E0517	Figure is not closed.	
CAPSYMT-E0518	Process specification data error	[C/Y]
	Process specification data cannot be read.	
[Warning]		

[waning]	
CAPSYMT-W0050	Figure is parallel.
CAPSYMT-W0051	Figure is not crossed.
CAPSYMT-W0052	Figure is tangent.
CAPSYMT-W0053	Out of used modifier is specified.
CAPSYMT-W0054	Not decided by specified modifier.
CAPSYMT-W0055	Figure is a concentric circle.
CAPSYMT-W0056	Point is included in circle.
CAPSYMT-W0057	Point is exist on circumference.

CAPSYMT-W0058	Figure is same figure.
CAPSYMT-W0059	Circle is included by another circle.
CAPSYMT-W0060	Radius is zero or negative value.
CAPSYMT-W0061	Data is over the limit.
CAPSYMT-W0063	Figure is exist in line.
CAPSYMT-W0064	Corner-R value is zero or negative.
CAPSYMT-W0065	Chamfer value is zero or negative.
CAPSYMT-W0066	Corner-R is not inserted.
CAPSYMT-W0067	Chamfer is not inserted.
CAPSYMT-W0068	Grooving is not inserted.
CAPSYMT-W0069	Corner data is not created at grooving.
CAPSYMT-W0070	Threading is not inserted.
CAPSYMT-W0071	Necking is not inserted.
CAPSYMT-W0072	Corner data is not created at necking.
CAPSYMT-W0074	Element is not decided.
CAPSYMT-W0099	Calculated error
CAPSYMT-W0100	Element type is not suitable.
CAPSYMT-W0101	Previous tangent is specified in spite of undefined C.F.
CAPSYMT-W0102	Element definiton data is lack.
CAPSYMT-W0103	End-point of element is not found.
CAPSYMT-W0104	Modifier for decided end-point is not suitable.
CAPSYMT-W0108	Element error at start-point is occured.
CAPSYMT-W0109	Previous element is not found at current position.
CAPSYMT-W0400	Specified element is not inserted at current position.
CAPSYMT-W0401	Specified element is not deleted.
CAPSYMT-W0402	Start-point is not deleted.
CAPSYMT-W0404	Grooving element is not inserted at 1st element.
CAPSYMT-W0405	Thread element is not inserted at 1st element.
CAPSYMT-W0406	Necking element is not inserted at 1st element.
CAPSYMT-W0407	Corner element is not inserted at 1st element.
CAPSYMT-W0408	Corner value is empty.
CAPSYMT-W0409	Figure is not defined.
CAPSYMT-W0410	A process-specified figure is deleted. Process specification data is also
	deleted. [C/Y]
	If all entities are deleted while a machining profile that was previously
	created by symbolic input is being edited, any previously created
	machining profile is deleted. This message is displayed for a process-
	specified machining profile.
CAPSYMT-W0411	An edited figure is not a closed figure. [C/Y]
	While a machining profile that was previously created by symbolic input is
	being edited, this message is displayed if a mismatch between the start
	point and end point results from a modification made to a pocket figure,
	island figure, or hollow figure. Make the modifications necessary to match

the start and end points.

CAPSYMT-W0412	Machining data is defined for a modified blank figure.	[C/Y]
	The machining data may become unusable due to a machining p	orofile
	modification. So, be sure to check the machining data before creating	ıg NC
	data.	
	While a machining profile that was previously created by symbolic in	put is
	being edited, this message is displayed if a process is specified for	or the
	machining profile.	

[Abbreviation SYMGVP]	[N] Messages concerning groove figure creation [C/Y]
Abbreviation-number	Message (Explanation)
[Error]	
SYMGVPTN-E0502	The machining surface is incorrect. [C/Y]
	This message is displayed when a selected figure cannot be created using
	the currently selected machining surface.
SYMGVPTN-E0504	Input data is insufficient. [C/Y]
	This message is displayed when all of the input data required for figure
	creation has not been entered.
SYMGVPTN-E0505	Input data is incorrect. [C/Y]
	This message is displayed when the input data is incorrect.
SYMGVPTN-E0506	The angle of a groove is too large. [C/Y]
	This message is displayed when a value greater than 360° is defined for
	the angle of a circular groove on an end face, or for the angle of a groove
	around a circular cylinder.
SYMGVPTN-E0507	The angle of a groove is greater than the pitch angle. [C/Y]
	This message is displayed when a value greater than the pitch angle is
	defined for the angle of a circular groove on an end face, or for the angle
	of a groove around a circular cylinder.
SYMGVPTN-E0508	The total of the angles of multiple grooves exceeds 360°. [C/Y]
	This message is displayed when the total of the defined angles of multiple
	circular grooves on an end face or multiple grooves around a circular
	cylinder exceeds 360°.
SYMGVPTN-E0509	No more than 300 grooves can be defined. [C/Y]
	This message is displayed when more grooves than the maximum number
	that can be created at one time, namely 300, are entered.

[Abbreviation SYMHLPTN] Messages concerning hole figure creation		[C/Y]
Abbreviation-number	Message (Description)	
[Error]		
SYMHLPTN-E0502	The machining surface is incorrect.	[C/Y]
	This message is displayed when a selected figure cannot	ot be created using

the currently selected machining surface.

-344-

SYMHLPTN-E0503	The amount of memory is insufficient.	[C/Y]
	This message is displayed when the amount of memory i	is too small to
	create a figure.	
SYMHLPTN-E0504	Input data is insufficient.	[C/Y]
	This message is displayed when all of the input data requ	ired for figure
	creation has not been entered.	
SYMHLPTN-E0505	Input data is incorrect.	[C/Y]
	This message is displayed when the input data is incorrect.	
SYMHLPTN-E0506	No more than 1000 holes can be defined.	[C/Y]
	This message is displayed when more holes than the max	kimum number
	that can be created at one time, namely 1000, are entered.	

[Abbreviation SYMMLPTN] ----- Messages concerning rectangular/circular figure creation [C/Y]

Abbreviation-number	Message (Description)	
[Error]		
SYMMLPTN-E0502	The machining surface is incorrect.	[C/Y]
	This message is displayed when a selected figure cannot be created	ed using
	the currently selected machining surface.	
SYMMLPTN-E0504	Input data is insufficient.	[C/Y]
	This message is displayed when all of the input data required for	r figure
	creation has not been entered.	
SYMMLPTN-E0505	Input data is incorrect.	[C/Y]
	This message is displayed when the input data is incorrect.	
SYMMLPTN-E0506	A corner R value is too large.	[C/Y]
	In rectangular figure creation, this message is displayed when a c	orner R
	value is equal to or greater than half of the vertical or horizon	ital side
	length.	

26.3.3 Messages related to pre-machining settings

[Abbreviation CHCKANMD] ----- Messages related to chuck data setting for animated simulation

Abbreviation-number	Message (Explanation)
[Error]	
CHCKANMD-E1001	Unsuitable type of the chuck.
	The set data type of a chuck is inappropriate. Re-set it.
CHCKANMD-E1002	Short of Memory.
	Check the state of the system memory, and restart the system.
CHCKANMD-E1003	Unable to open the kept file 'chcktail.chk'. Please make sure of the attribute, etc.
CHCKANMD-E1004	Unable to write in the kept file 'chcktail.chk'. Please make sure of the attribute, etc.
CHCKANMD-E1006	Unable to edit the kept file 'chcktail.chk'. Please make sure of the attribute, etc.

	Check whether the chcktail.ctf file (in the directory where the tooling/tool
	files specified during environment setting are saved) is write-protected.
CHCKANMD-E1005	Data of the appointed number is not.
CHCKANMD-E1007	Unable to delete a data. Please make sure of the attribute, etc.
	Check whether the chcktail.ctf file (in the directory where the tooling/tool
	files specified during environment setting are saved) is write-protected.
CHCKANMD-E1008	Unable to open the partfile.
	Check whether the currently selected part file is write-protected.
CHCKANMD-E1009	Unable to read a data.
CHCKANMD-E1010	The Current Selected Chuck is unsuitable.
	The Current Selected Chuck data, specified as the chuck data used for
	animated simulation, is inappropriate. Check whether the data type and
	figure data are correct.
CHCKANMD-E1012	System error
CHCKANMD-E1013	Unable to get the element of the blank figure.
CHCKANMD-E1016	Unsuitable blank figure.
	Check whether the material figure set up on the Machining Definition
	window is correct. It is likely that the figure is crossed or open.
CHCKANMD-E1014	No blank figure.
CHCKANMD-E1015	Unable to get the blank figure.
	Set the blank figure on the Machining Definition window. The blank
	figure is used to determine where the chuck is to be located for animated
	simulation.
CHCKANMD-E1017	Plural blank figure are selected.
	Check whether the blank material is set up correctly on the Machining
	Definition window.
[Warning]	
CHCKANMD-W2001	Select the edited chuck.
	Select one chuck to be edited from the list.
CHCKANMD-W2002	Unable to edit plural data at the same time.
	Only one chuck data item can be selected at a time. Select one chuck to be
	edited from the list.
CHCKANMD-W2003	The Current Selected Chuck is not.
	The chuck data to be used in animated simulation has not been selected.
CHCKANMD-W2004	Unable to select plural data at the same time.
	Only one chuck data item can be used in animated simulation.
CHCKANMD-W2005	Select some data.
	For erasure or addition to the registration list, select the chuck data from
	the list.
CHCKANMD-W2006	Unable to get the data.
CHCKANMD-W2007	Chuck data 'L' is not. Do you try to set again?
CHCKANMD-W2008	Chuck data 'L1' is not. Do you try to set again?
CHCKANMD-W2009	Chuck data 'W' is not. Do you try to set again?

CHCKANMD-W2010	Chuck data 'W1' is not. Do you try to set again?
	L, L1, W, or W1 has not been set in the chuck data specified to be used in
	animated simulation. If the window is exited with No selected, the Current
	Selected Chuck data will not be written to the part file.
CHCKANMD-W2011	Please select some data to delete.
	For erasure, select the corresponding chuck data from the list. Note,
	however, that data cannot be erased from the Current Selected Chuck list.
CHCKANMD-W2012	No Registered Chuck is.
	No chuck has been set in the Registered Chuck list.
CHCKANMD-W2013	The Current Selected Chuck is unsuitable for the inner diameter of
	the blank figure. Do you try to set again?
	The chuck with inner pawls to be used in animated simulation is larger
	than the inner diameter of the blank figure, and cannot pick up the blank
	figure. Select a chuck smaller than the inner diameter.

26.3.4 Messages concerning machining surface setting

[Abbreviation MACHPLA	AN] Messages concerning machining surface setting	[C/Y]
Abbreviation-number	Message (Description)	
[Error]		
MACHPLAN-E0211	No turning surfaces and end faces can be deleted. machining surfaces (prefixed by * in the list) that were add	Only those ed later can
	be deleted.	[C/Y]
MACHPLAN-E0212	The machining surface contains machining profiles. Befo	ore deleting
	the machining surface, delete those machining profiles.	[C/Y]
MACHPLAN-E0213	The height of a machining surface is too large or too s	mall. The
	entered value was unusable.	[C/Y]

26.3.5 Messages related to machining definition

[Abbreviation with KOC] Wessages concerning process specification	
Abbreviation-number	Message (Explanation)
[Error]	
MPROC-E0001	There is no area that can be machined. Alternatively, no cutting area
	can be determined for a specified tool or cutting condition.
	Check that there is an area that can be machined using a specified
	machining method.
MPROC-E0002	A tool suitable for machining has not been registered as tooling data.
	Add a tool suitable for machining to the tooling data, and rerun.
MPROC-E0003	No cutting condition can be determined automatically, because the
	necessary data has not been registered as cutting condition data.
	Check that the cutting condition data is registered correctly and that all
	necessary data is registered, then rerun.

[Abbreviation MPROC] ----- Messages concerning process specification

MPROC-E0004	Machining is impossible with the specified tool.	
	Check that the specified tool is appropriate for machining.	
MPROC-E0005	A blank figure has not been prepared. Alternatively, a blank f	igure is
	incorrect.	
	Prepare a blank figure correctly, and rerun.	
MPROC-E0006	A parts figure has not been prepared. Alternatively, a parts f	igure is
	incorrect.	
	Prepare a parts figure correctly, and rerun	
MPROC-E0007	No machining profile is created on any machining surfaces.	[C/Y]
	Retry after creating machining profiles.	
MPROC-E0008	On the currently selected machining surface, machining profi	les that
	can be machined by the specified machining method are not cre	ated. [C/Y]
	Select a machining surface on which machining profiles are created	ł.
MPROC-E0009	For a turning surface, C/Y-axis machining data cannot be defin	ed. [C/Y]
	Select a machining surface other than a turning surface.	
MPROC-E0010	The selected figure is not an island figure.	[C/Y]
	Select an island figure.	
MPROC-E0011	The selected figure is not a hollow figure.	[C/Y]
	Select a hollow figure.	
MPROC-E0012	A figure that cannot be machined with the specified machining	method
	was selected.	[C/Y]
	Select a figure that can be machined with the specified method.	
MPROC-E0013	An error occurred during processing of the initialization file.	[C/Y]
	The initialization file does not exist under the Windows director	ry, or a
	system environment problem such as insufficient memory has o	ccurred.
	After checking the program and system status, restart the system.	
MPROC-E0014	An error occurred during processing of the file for au	tomatic
	determination.	[C/Y]
	The file for automatic determination does not exist, or a	system
	environment problem such as insufficient memory has occurred.	After
	checking the program and system status, restart the system.	
MPROC-E0015	The same tool number is assigned to different tools.	[C/Y]
	Check that incorrect tool numbers are not specified.	
MPROC-E0016	On the currently selected machining surface, machining profi	les of a
	specified type are not created.	
	Retry after creating machining profiles.	
MPROC-E0017	The selected figure is not a prepared hole figure.	[C/Y]
	Select a prepared hole figure.	
MPROC-E0018	The selected prepared hole figure is a prepared hole figure of	of other
~ ~ ~ ~ ~	machining data, so that the figure cannot be deleted.	[C/Y]
	Delete the prepared hole figure by modifying the machining data of	defining
	the prepared hole.	U
MPROC-E0019	Machining data is defined for the selected prepared hole fig	ure, so
	that the figure cannot be deleted.	[C/Y]
	Delete the prepared hole figure after deleting the machining data	defined
	for the prepared hole figure.	
	σ	

MPROC-E0020	An attempt was made to delete a prepared hole figure, but machining
	data is defined for the figure. So, the figure cannot be deleted. [C/Y]
	Delete the prepared hole figure after deleting the machining data defined
	for the prepared hole figure.
MPROC-E0021	Machining data is defined for the current prepared hole figure, so that
	the figure cannot be updated. [C/Y]
	Create a prepared hole figure after deleting the machining data defined for
	the prepared hole figure.
MPROC-E0022	The prepared hole figure cannot be created under the specified
	conditions. [C/Y]
	Check whether the dimensions of the prepared hole figure are correct.
MPROC-E0023	Under the specified machining conditions, no cutting start point can be
	determined automatically. [C/Y]
	Check whether the machining conditions are correct.
MPROC-E0024	The selected figure is not a cutting start point. [C/Y]
	Select a cutting start point.
MPROC-E0025	All entities of the selected pocketing figure are open entities, so that
	there is no area that can be cut. [C/Y]
	When all the entities of a pocketing figure are open entities, contouring
	cannot be defined
MPROC-E0026	All entities of the selected nocketing figure are onen entities, and no
	island figure is defined for the figure, so that there is no area that can
	be cut
	Pocketing (side face finishing) cannot be defined for a pocketing figure
	when all entities of the figure are open entities and no island figure is
	defined for the figure
MPROC-E0027	A specified cutting condition is incorrect
	(Denth of cut (tool radius direction) > Tool radius)
	Specify a depth of cut (tool radius direction) smaller than the tool radius
MPROC-E0028	The selected item is not a machining data item so that the item cannot
	he released from the machining group
	Select a machining data item from the process specification list then retry
MPROC-E0029	The selected machining data item cannot be released from the
	machining group.
	No machining data item can be released from a machining group when the
	machining group has only one machining data item registered
MPROC-F0030	Multiple machining data items cannot be released from a machining
MI KOC-20030	
	Multiple machining data items cannot be released from a machining
	aroun. Select only one machining data item from the process specification
	list then rates
MPROC-F0031	In the last line of the process specification list auviliary work connet
MI KUU-LUUJI	be executed
	In the last line of the process specification list sub-liner work such as T
	in the last line of the process specification list, auxiliary work such as 1
	code initialization and release from a machining group cannot be executed.

[Warning]	
MPROC-W0200	The cutting method does not match the tool holder type. [C/Y]
	The method of tapping (floating/rigid) does not match the specified tool
	holder type (floating/rigid). If a mismatch exists, the tool may be damaged
	during machining. Ensure that there is no mismatch.
MPROC-W0201	A modification was made to the first machining data in the machining
	group. Such a modification may invalidate other machining data of
	the machining group. So, be sure to check the other machining data. [C/Y]
	The machining data of the machining group shares the data, including tool
	data, of the first machining data. If a modification is made to data such as
	tool data of the first machining data, the cutting conditions of machining
	data after the first machining data may become unsuitable, or incorrect NC
	data may be created. So, check the machining data following the first
	machining data before creating NC data.

[Abbreviation FPAUTO]	Messages concerning fully automatic process determination [C/Y]
Abbreviation-number	Message (Description)
[Error]	
FPAUTO-E0001	A figure cannot be created for some reason. [C/Y]
	Figure name: "Figure name"
	Data such as figure information and machining information is abnormal,
	so that no figure can be created.
FPAUTO-E0002	A prepared hole cannot be created. [C/Y]
	Figure name: "Figure name"
	An attempt was made to create prepared holes automatically. However,
	the attempt failed because the position of a prepared hole could not be
	determined due to its figure, or because no suitable tool could be found.
FPAUTO-E0003	A tool cannot be found. [C/Y]
	Figure name: "Figure name"
	A tool required for the defined machining is not registered in the tooling
	file.
FPAUTO-E0004	A cutting condition is not registered. [C/Y]
	Figure name: "Figure name"
	A cutting condition for the defined machining is not registered in the
	cutting condition library.
FPAUTO-E0005	The combination of a machining profile and machining type is
	incorrect. [C/Y]
	Figure name: "Figure name"
	Machining cannot be defined by a combination of a machining profile type
	and the machining type information associated with the machining profile.

26.3.6 Messages related to NC data preparation

Abbreviation-number	Message (Explanation)
[Error]	
NCOUT-E0001	The error occurred by the instruction when beginning to program.
	An error occurred when data was output at the start of the program. The
	data (such as O-number) belonging to the program or data common to all
	processes is incorrect. Check the data on the Setting of Pre-machining and
	NC Data Preparation Setting windows.
NCOUT-E0002	The error occurred by the instruction when the program ended.
	An error occurred when data signifying the end of the program was output.
	It is likely that the system environment has a problem, such as insufficient
	memory. Check the program and system state, then restart the system.
NCOUT-E0003	The error occurred by the instruction when the process began(main).
	An error occurred when data (such as cutting condition) belonging to a
	process was output. It is likely that the command data is out of the
	allowable range. Open the Machining Definition window for the process.
	and check the data on it.
NCOUT-E0004	The error occurred by the instruction when the process began(sub).
	An error occurred when data (such as cutting condition) belonging to a
	process was output. It is likely that the command data is out of the
	allowable range. Open the Machining Definition window for the process,
	and check the data on it.
NCOUT-E0005	The error occurred by the instruction when the process ended.
	An error occurred when data signifying the end of the process was output.
	It is likely that the system environment has a problem, such as insufficient
	memory. Check the program and system state, then restart the system.
NCOUT-E0006	The error occurred by instructing in the tool and the coordinates
	system setting.
	An error occurred when tool or coordinate system setting data was output.
	It is likely that the specified data is out of the allowable range. Check the
	tool data indicated on the Machining Definition window for the process in
	which the error occurred, and the tool and tooling files related to that data.
NCOUT-E0007	The error occurred by the machine stating point position instruction.
	An error occurred when the home position data was output. It is likely that
	the specified data is out of the allowable range. Check the Home position
	on the Setting of Pre-machining window.
NCOUT-E0008	The error occurred by the tool exchange position instruction.
	An error occurred when the tool exchange position data was output. It is
	likely that the specified data is out of the allowable range. Check the Index
	position on the Setting of Pre-machining window and the Start position of
	a process in which the error occurred.
	a process in which the error occurred.

-351-

NCOUT-E0009	The error occurred by the tool shape data instruction.
	An error occurred when the tool figure data was output. Check the tool
	data indicated on the Machining Definition window for the process in
	which the error occurred, and the tool file and tooling file for that tool.
NCOUT-E0010	The error occurred the instruction concerning the main axis.
	An error occurred when data such as a rotation speed and cutting speed
	was output. It is likely that the specified data is out of the allowable range.
	Open the Machining Definition window for the process in which the error
	occurred, and check the data indicated on it.
NCOUT-E0011	The error occurred by instructing in the tool exchange position the
	movement.
	An error occurred when a command for moving the tool to the tool
	exchange position was issued. It is likely that the specified data is out of
	the allowable range Check the Index position on the Setting of Pre-
	machining window and the Start position of a process in which the error
	occurred
NCOUT-F0012	The error occurred by instructing in passed point the movement
	An error occurred when a command for moving the tool to the pass
	position was issued. It is likely that the specified data is out of the
	allowable range. Check the pass position for a process in which the error
	anowable range. Check the pass position for a process in which the error
NCOUT E0012	The owner economical by instructing the machine starting point position
NCUUI-E0015	The error occurred by instructing the machine starting point position
	or the turret rotating position.
	An error occurred when the nome of index position data was output. It is
	inkery that the specified data is out of the allowable range. Check the Home
	position and index position on the Setting of Pre-machining window.
NCOUT-E0020	The data when beginning to program is not correct.
	An error occurred when data common to all programs was loaded. Data
	that cannot be recognized is included. Check the data on the Setting of
	Pre-machining and NC Data Preparation Setting windows.
NCOUT-E0021	The data when the process begins is not correct.
	An error occurred when data related to a process was loaded. Data that
	cannot be recognized is included. Check the data on the Machining
	Definition window for the process in which the error occurred.
NCOUT-E0022	The data of the tool and the coordinates system setting is not correct.
	An error occurred when the tool and coordinate system setting data was
	loaded. It is likely that a parameter such as a tool number has not been
	specified or that data that cannot be recognized is included. Check the tool
	data and coordinate system selection data on the Machining Definition
	window for the process in which the error occurred.
NCOUT-E0023	The process data is not correct.
	An error occurred when the process data was loaded. Data that cannot be
	recognized is included. Check whether the relevant Machining Definition
	window is normal.
NCOUT-E0024	The data for the interference check is not correct.
	An error occurred when the interference check data was loaded. Data that

	cannot be recognized is included. Check the tool data on the Machining
	Definition window.
NCOUT-E0025	The tool shape data instruction is not correct.
	An error occurred when the tool figure data was loaded. Data that cannot
	be recognized is included. Open the Machining Definition window, and
	check the tool data for the process in which the error occurred, the tool file
	and tooling file for that tool.
NCOUT-E0026	The machine reference point or the turret rotating point is not correct.
	An error occurred when the home or index position data was loaded. Data
	that cannot be recognized is included. Open the Setting of Pre-machining
	window, and check the Home position and Index position data.
NCOUT-E0030	The data for the roughness processing is not correct.
	An error occurred when the cutting condition data for roughing was
	loaded. Data that cannot be recognized is included. Open the Machining
	Definition window, and check the cutting condition data for the process in
	which the error occurred.
NCOUT-E0031	The data for the semi-finishing processing is not correct.
	An error occurred when the cutting condition data for semi-finishing was
	loaded. Data that cannot be recognized is included. Open the Machining
	Definition window, and check the cutting condition data for the process in
	which the error occurred.
NCOUT-E0032	The data for the finishing processing is not correct.
	An error occurred when the cutting condition data for finishing was
	loaded. Data that cannot be recognized is included. Open the Machining
	Definition window, and check the cutting condition data for the process in
	which the error occurred.
NCOUT-E0033	The data for the hole opening processing is not correct.
	An error occurred when the cutting condition data for hole machining was
	loaded. Data that cannot be recognized is included. Open the Machining
	Definition window, and check the cutting condition data for the process in
	which the error occurred.
NCOUT-E0034	The data for the grooving processing data is not correct.
	An error occurred when the cutting condition data for grooving was
	loaded. Data that cannot be recognized is included. Open the Machining
	Definition window, and check the cutting condition data for the process in
	which the error occurred.
NCOUT-E0035	The data for the threading processing is not correct.
	An error occurred when the cutting condition data for threading was
	loaded. Data that cannot be recognized is included. Open the Machining
	Definition window, and check the cutting condition data for the process in
	which the error occurred.
NCOUT-E0036	The data for the necking processing is not correct.
	An error occurred when the cutting condition data for necking was loaded.
	Data that cannot be recognized is included. Open the Machining
	Definition window, and check the cutting condition data for the process in
	which the error occurred.

NCOUT-E0050	The machining face data cannot be acquired.
	An attempt to obtain the machining plane data failed. It is likely that the
	system environment has a problem, such as insufficient memory. Check
	the program and system state, then restart the system.
NCOUT-E0051	The cutting area cannot be acquired.
	An attempt to obtain a machining area failed. It is likely that no
	machining area was prepared during machining definition. It is also likely
	that the system environment has a problem, such as insufficient memory.
	Check the program and system state, then restart the system.
NCOUT-E0052	The process data cannot be acquired.
	An attempt to obtain the process data failed. It is likely that the system
	environment has a problem, such as insufficient memory. Check the
	program and system state, then restart the system.
NCOUT-E0053	The set up data cannot be acquired.
	An attempt to obtain the setup data failed. It is likely that the system
	environment has a problem, such as insufficient memory. Check the
	program and system state, then restart the system.
NCOUT-E0080	It is not possible to refer to the data of the INI file.
	An attempt to obtain the INI file data failed. It is likely that the INI file is
	not in the Windows directory. It is also likely that the system environment
	has a problem, such as insufficient memory. Check the program and
	system state, then restart the system.
NCOUT-E0081	It is not possible to refer to the data of the NC machine file.
	An attempt to obtain the NC machine file data failed. It is likely that the
	NC machine file is not in the specified directory. It is also likely that the
	system environment has a problem, such as insufficient memory. Check
	the program and system state, then restart the system.
NCOUT-E0082	It failed in the initialization of the POST. Please refer to 'LOG' file for
	detailed. Do vou open 'LOG' file?
	An attempt to initialize the POST section failed. It is likely that files such
	as the NC machine file are not in the specified directory. It is also likely
	that the system environment has a problem such as insufficient memory
	Check the program and system state, then restart the system
NCOUT.F0099	The error which was not able to be expected occurred
	An unexpected error such as a calculation overflow occurred. It is likely
	that there is insufficient memory Check the program then restart the
	system
NCOUT-F0101	The following process is a main process
NCOUT-F0102	The following process is a main process.
NCOUT-F0102	Next there is no process
NCOUI-E0103	These error messages indicate the type of the payt process.
	not usually occur. Once they occur, it is likely that the system environment
	has a problem such as insufficient memory Check the program then
	nas a problem, such as insumcient memory. Check the program, then
	restart the system.

NCOUT-E0111	The specified file name is not correct.
	The specified file name contains an invalid character or symbol. Check
	the file name.
NCOUT-E0113	I fail to remake the cutting area.
	An attempt to re-create a machining area failed. It is likely that the system
	environment has a problem, such as insufficient memory. Check the
	program and system state, then restart the system.
NCOUT-E0114	There is no process data.
	There is no process data. Define the type of machining in the Machining
	Definition section.
NCOUT-E0115	Turning process data does not exist. [C/Y]
	Turning process data does not exist. If there are C/Y-axis machining
	processes only, this message is displayed when the program is executed for
	the first time.
NCOUT-E0297	This NC machine file does not allow C-axis machining. [C/Y]
	The selected NC machine file is inadequate. Select an NC machine file
	that allows C-axis machining.
NCOUT-E0298	This NC machine file does not allow Y-axis machining. [C/Y]
	The selected NC machine file is inadequate. Select an NC machine file
	that allows Y-axis machining.
NCOUT-E0299	"C/Y AXIS MODULE" is not installed.
	The C/Y-axis module is not installed, but a part file created with the C/Y-
	axis module is executed. Install the C/Y-axis module before executing a
	part file created with the C/Y-axis module.
NCOUT-E1001	The data for the machining cannot be acquired.
	An error occurred when the machining condition data was loaded. Data
	that cannot be recognized is included. Open the Machining Definition
	window, and check the machining condition data for the process in which
	the error occurred.
NCOUT-E1002	The error occurred by the interference check.
	The tool will interfere with the workpiece when it approaches the
	machining start position at the beginning of a process or when it escapes to
	the machining start position at the end of a process. Open the Setting of
	Pre-machining window, and check whether the Home and Index positions
	were set to a point inside the workpiece.
	Alternatively, interference occurred between the tool and workpiece during
	grooving. Check an escape amount for grooving.
NCOUT-E1003	The memory does not suffice.
	Check the state of the system memory, and restart the system.
NCOUT-E1004	The data base I.D. is not correct.
	This error is related to the data of the next process. It does not usually
	occur. Once it occurs, it is likely that the system environment has a
	problem, such as insufficient memory. Check the program and system
	state, then restart the system.

NCOUT-E1005	The data for machining cannot be acquired.
	An error occurred when the machining condition data was loaded. Data
	that cannot be recognized is included. Open the Machining Definition
	window, and check the machining condition data for the process in which
	the error occurred.
NCOUT-E1006	The cutting area cannot be acquired.
	An attempt to obtain a machining area failed. It is likely that the system
	environment has a problem, such as insufficient memory. Check the
	program and system state, then restart the system.
NCOUT-E1099	The error which was not able to be expected occurred.
	An unexpected error, such as a calculation overflow, occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system.
NCOUT-E1101	The clearance amount is not correct.
	A clearance amount for hole machining is incorrect. Check the process
	data for hole machining in the Machining Definition section.
NCOUT-E1102	The cutting amount is not correct.
	A cutting depth for hole machining is incorrect. Check the process data
	for hole machining in the Machining Definition section.
NCOUT-E1103	The feedrate is not correct.
	A feed amount for hole machining is incorrect. Check the process data for
	hole machining in the Machining Definition section.
NCOUT-E1104	The dwell time is not correct.
	The dwell time for hole machining is incorrect. Check the process data for
	hole machining in the Machining Definition section.
NCOUT-E1106	In C-axis machining, the escape angle must be 0 or 180. [C/Y]
	In C-axis fine boring, an incorrect escape angle is specified. Check the
	hole machining process data of the process specification section.
NCOUT-E1107	The angle of the chamfering tool is incorrect. [C/Y]
	In hole chamfering, the angle of the tool used is incorrect. Check the hole
	machining process data of the process specification section.
NCOUT-E1108	Machining is impossible because the tool radius is greater than the
	machining radius. [C/Y]
	Circle cutting or hole contour chamfering is impossible because the radius
	of the tool used is greater than the machining radius. Check the hole
	machining process data of the process specification section.
NCOUT-E1109	An approach or escape radius specified is so large that interference
	with the machining radius occurs. [C/Y]
	An approach or escape radius specified for circle cutting or hole contour
	chamfering is so large that an interference with the hole figure occurs.
	Check the hole machining process data of the process specification section.
NCOUT-E1199	A drill machining ended.
	This error message indicates the end of drilling. This error does not
	usually occur. Once it occurs, it is likely that there is insufficient memory.
	Check the program, then restart the system.

NCOUT-E1200	The error which was not able to be expected occurred.
	An unexpected error, such as a calculation overflow, occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system.
NCOUT-E1201	The area for machining is not correct.
	The machining area is incorrect. Data that cannot be recognized may be
	included. Open the Machining Definition window, and check the program
	for the process in which the error occurred.
NCOUT-E1202	The cutting condition is not correct.
	The machining condition data is inappropriate. Data that cannot be
	recognized may be included. Open the Machining Definition window, and
	check the program for the process in which the error occurred.
NCOUT-E1203	The area for machining cannot be acquired.
NCOUT-E1204	The error which was not able to be expected occurred. (The element is
	not correct.)
	An attempt to obtain a machining area failed. Data that cannot be
	recognized may be included. Open the Machining Definition window, and
	check the program for the process in which the error occurred.
NCOUT-E1205	The feedrate to the roughness cannot be acquired.
NCOUT-E1206	The set feedrate is improper.
	A feed amount is not set for a specific surface roughness. Alternatively, a
	feed amount set for a specific surface roughness is inappropriate. Open the
	Machining Definition window, and check the surface roughness and feed
	amount set for the process in which the error occurred.
NCOUT-E1207	The set machining method is improper.
NCOUT-E1208	The method of machining the set valley is improper.
	This error message indicates the type of a machining method used. This
	error does not usually occur. Once it occurs, it is likely that there is
	insufficient memory. Check the program, then restart the system.
NCOUT-E1240	System error
NCOUT-E1241	System error
NCOUT-E1242	System error
NCOUT-E1243	System error
NCOUT-E1244	System error
NCOUT-E1245	System error
	An unexpected error, such as a calculation overflow, occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system.
NCOUT-E1246	The memory does not suffice.
	Check the state of system memory, then restart the system.
NCOUT-E1247	The area for the machining is not correct.
NCOUT-E1261	The cutting area is not correct.
NCOUT-E1263	The area is not correct.
	The machining area is incorrect. Data that cannot be recognized may be
	included. Open the Machining Definition window, and check the program
	for the process in which the error occurred.

NCOUT-E1260	System error
NCOUT-E1262	System error
NCOUT-E1264	System error
NCOUT-E1280	System error
NCOUT-E1281	System error
NCOUT-E1282	System error
NCOUT-E1300	System error
NCOUT-E1303	System error
NCOUT-E1305	System error
	An unexpected error, such as a calculation overflow, occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system.
NCOUT-E1301	The machining condition is not correct. (Grooving)
	The machining condition for grooving is inappropriate. Data that cannot
	be recognized may be included. Open the Machining Definition window,
	and check the machining condition data for the process in which the error
	occurred.
NCOUT-E1302	There is no area for the machining. (Grooving)
	An attempt to obtain a machining area for grooving failed. Open the
	Machining Definition window, and check the machining area for the
	process in which the error occurred.
NCOUT-E1304	The memory does not suffice.
	Check the state of system memory, and restart the system.
NCOUT-E1306	The element cannot be acquired. (Grooving)
NCOUT-E1308	The kind of the element is not suitable. (Grooving)
NCOUT-E1312	The cutting area is not correct. (Grooving)
	The machining area for grooving is incorrect. Data that cannot be
	recognized may be included. Open the Machining Definition window, and
	check the machining area for the process in which the error occurred.
NCOUT-E1307	The data cannot be acquired. (Grooving)
NCOUT-E1309	The feedrate is improper. (Grooving)
NCOUT-E1310	The kind of machining is not correct. (Grooving)
NCOUT-E1311	The specification of the machining method is not correct. (Grooving)
	The machining condition for grooving is inappropriate. Data that cannot
	be recognized may be included. Open the Machining Definition window,
	and check the machining condition data specified for the process in which
	the error occurred.
NCOUT-E1500	System error
	An unexpected error, such as a calculation overflow, occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system.
NCOUT-E1501	The machining condition is not correct. (Threading)
NCOUT-E1502	The kind of machining is not correct (Threading)

NCOUT-E1503	The machining shape is not correct. (Threading)
	The machining condition for threading is inappropriate. Data that cannot
	be recognized may be included. Open the Machining Definition window,
	and check the machining condition data specified for the process in which
	the error occurred.
NCOUT-E1900	System error
NCOUT-E1902	System error
NCOUT-E1999	System error
	An unexpected error, such as a calculation overflow, occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system.
NCOUT-E1901	The start point is not obtained.
NCOUT-E1904	The end point is not obtained.
NCOUT-E1903	The machining start point is not obtained.
	It is impossible to calculate the start position of the next machining
	process. Alternatively, the Home or Index position has not been set up.
	Open the Machining Definition window and check the machining
	condition data and machining area for the process in which the error
	occurred Also open the Setting of Pre-machining window and check for
	the Home and Index positions
NCOUT_F1905	The tool noth cannot be decided
NCOUT-E1905	The CL data cannot be output
NCOUI-E1700	An unexpected error such as a calculation overflow occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system
NCOUT E1007	The memory does not suffice
NCOUI-E190/	The memory does not suffice.
NCOLT E1009	There is no area for the interference sheet (blank or parts figure)
NCUUI-E1908	The machining area is incorrect. Data that cannot be recognized may be
	included. Open the Machining Definition window, and shack the program
	for the ansacce in which the error accurred
NCOLE E1000	for the process in which the error occurred.
NCOUT-E1909	The movement point is in the area.
	The tool will interfere with the workpiece when it approaches the
	machining start position at the beginning of a process or when it escapes to
	the machining start position at the end of a process. Open the Setting of
	Pre-machining window, and check whether the Home and Index positions
	have been set to a point inside the workpiece.
NCOUT-E1910	The kind of machining is improper.
	This error message indicates the type of a machining method used. This
	error does not usually occur. Once it occurs, it is likely that there is
	insufficient memory. Check the program, then restart the system.
NCOUT-E1950	The tool interfered with a part. [C/Y]
	This message is displayed if a check to see if the tool interferes with a part
	in C/Y-axis machining finds that interference occurs on the path. If the
	use of the interference check function is not specified on the NC creation
	preparation window, no interference check is made, and this message is

-359-

	not displayed.
NCOUT-E2102	An incorrect pocketing figure is specified. [C/Y]
	In pocketing, an incorrect pocketing figure is specified. Check the
	pocketing process data of the process specification section.
NCOUT-E2103	An incorrect cutting start point is specified. [C/Y]
	In pocketing, an incorrect cutting start point is specified. Check the
	pocketing process data of the process specification section.
NCOUT-E2104	An incorrect pocketing figure is specified. [C/Y]
	In pocketing, an incorrect pocketing figure is specified. Check the
	pocketing process data of the process specification section.
NCOUT-E2105	Machining condition data is incorrect. [C/Y]
	In pocketing, an incorrect machining condition is specified. Check the
	pocketing condition of the process specification section.
NCOUT-E2106	Pocketing cannot be performed with a specified condition. [C/Y]
	In pocketing, a specified machining condition is inadequate. Check the
	pocketing condition of the process specification section.
NCOUT-E2107	Finishing cannot be performed with a specified condition. [C/Y]
	In pocket finishing, a specified machining condition is inadequate. Check
	the pocket finishing condition of the process specification section.
NCOUT-E2108	There is no area to be machined. [C/Y]
	In pocketing, there is no area that can be machined with a specified
	machining condition. Check the pocketing condition of the process
	specification section.
NCOUT-E2109	The number of cuts exceeded the limit. [C/Y]
	In pocketing, the number of cuts exceeded a specified limit. Modify the
	pocketing condition of the process specification section (for example, to
	use a tool with a greater maximum depth of cut).
NCOUT-E2110	No cutting start point can be found. [C/Y]
	In pocketing, no cutting start point could be found. Check the pocketing
	condition and data of the process specification section.
NCOUT-E2111	The tool interfered with a part. [C/Y]
	In pocketing, the tool interfered with a part. Check the pocketing
	condition and data of the process specification section.
NCOUT-E2199	System error [C/Y]
NCOUT-E2202	An incorrect contouring figure is specified. [C/Y]
	In contouring, an incorrect contouring figure is specified. Check the
	contouring data of the process specification section.
NCOUT-E2203	An incorrect cutting start point is specified. [C/Y]
	In contouring, an incorrect cutting start point is specified. Check the
	contouring data of the process specification section.
NCOUT-E2204	An incorrect contouring figure is specified. [C/Y]
	In contouring, an incorrect contouring figure is specified. Check the
	contouring data of the process specification section.

NCOUT-E2205	Incorrect machining condition data is specified. [C/Y]
	In contouring, an incorrect machining condition is specified. Check the
	contouring condition of the process specification section.
NCOUT-E2206	Contouring cannot be performed with a specified condition. [C/Y]
	In contouring, a specified machining condition is inadequate. Check the
	contouring condition of the process specification section.
NCOUT-E2207	Finishing cannot be performed with a specified condition. [C/Y]
	In contour finishing, a specified machining condition is inadequate.
	Check the contour finishing condition of the process specification section.
NCOUT-E2208	There is no area to be machined. [C/Y]
	In contouring, there is no area that can be machined with a specified
	machining condition. Check the contouring condition of the process
	specification section.
NCOUT-E2209	The number of cuts exceeded the limit. [C/Y]
	In contouring, the number of cuts exceeded a specified limit. Modify the
	contouring condition of the process specification section (for example, to
	use a tool with a greater maximum depth of cut).
NCOUT-E2210	No cutting start point can be found. [C/Y]
	In contouring, no cutting start point could be found. Check the contouring
	condition and data of the process specification section.
NCOUT-E2211	The tool interfered with a part. [C/Y]
	In contouring, the tool interfered with a part. Check the contouring
	condition and data of the process specification section.
NCOUT-E2299	System error [C/Y]
NCOUT-E2302	An incorrect grooving figure is specified. [C/Y]
	In grooving, an incorrect grooving figure is specified. Check the grooving
	data of the process specification section.
NCOUT-E2303	An incorrect cutting start point is specified. [C/Y]
	In grooving, an incorrect cutting start point is specified. Check the
	grooving data of the process specification section.
NCOUT-E2304	An incorrect grooving figure is specified. [C/Y]
	In grooving, an incorrect grooving figure is specified. Check the grooving
	data of the process specification section.
NCOUT-E2305	Incorrect machining condition data is specified. [C/Y]
	In grooving, an incorrect machining condition is specified. Check the
	grooving condition of the process specification section.
NCOUT-E2306	Grooving cannot be performed with a specified condition. [C/Y]
	In grooving, a specified machining condition is inadequate. Check the
	grooving condition of the process specification section.
NCOUT-E2307	Finishing cannot be performed with a specified condition. [C/Y]
	In groove finishing, a specified machining condition is inadequate. Check
	the groove finishing condition of the process specification section.
NCOUT-E2308	There is no area to be machined.
	In grooving, there is no area that can be machined with a specified
	machining condition. Check the grooving condition of the process
	specification section.

NCOUT-E2309	The number of cuts exceeded the limit. [C/Y]
	In grooving, the number of cuts exceeded a specified limit. Modify the
	grooving condition of the process specification section (for example, to use
	a tool with a greater maximum depth of cut).
NCOUT-E2310	No cutting start point can be found. [C/Y]
	In grooving, no cutting start point could be found. Check the grooving
	condition and data of the process specification section.
NCOUT-E2311	The tool interfered with a part. [C/Y]
	In grooving, the tool interfered with a part. Check the grooving condition
	and data of the process specification section.
NCOUT-E2399	System error [C/Y]
NCOUT-E5001	The error occurred in POST. Please refer to LOG file for detailed. Do
	you open LOG file?
NCOUT-E5004	The memory does not suffice.
	Check the state of system memory, then restart the system.
NCOUT-E5097	The protecting key comes off.
	It is impossible to prepare NC data, because the protect key is detached.
	Attach it, then restart the system.
NCOUT-E5098	The system is a demonstration version.
	It is impossible to output NC data to a file, because this system is a
	demonstration version.
NCOUT-E5099	The error which was not able to be expected occurred.
	An unexpected error, such as a calculation overflow, occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system.
NCOUT-E7001	The blank figure does not exist.
	This program has no blank figure. Enter blank figure data.
NCOUT-E7002	Two or more blank figures exist.
	There is more than one blank figure for this program. This system cannot
	process more than one blank figure. Correct the program.
NCOUT-E7003	The blank figure cannot be acquired.
	A blank figure cannot be obtained. It is likely that the system environment
	has a problem, such as insufficient memory. Check the program and
	system state, then restart the system.
NCOUT-E7004	The blank figure is not correct.
	The blank figure is incorrect. The blank figure must not be crossed or
	open. Correct the program.
NCOUT-E7005	The blank figure for the animation cannot be registered.
	A blank figure for animated simulation cannot be registered. It is likely
	that the system environment has a problem, such as insufficient memory.
	Check the program and system state, then restart the system.
NCOUT-E7006	The turning parts figure does not exist.
	This program has no parts figure. Enter parts figure data.

NCOUT-E7007	Two or more turning parts figure exists.
	There is more than one parts figure for this program. This system cannot
	process more than one parts figure. Correct the program.
NCOUT-E7008	The turning parts figure cannot be acquired.
	Parts figure cannot be obtained. It is likely that the system environment
	has a problem, such as insufficient memory. check the program and system
	state, then restart the system.
NCOUT-E7009	The chuck figure is not correct.
	The chuck figure is incorrect. Check the chuck figure on the Setting of
	Pre-machining window.
NCOUT-E7010	The chuck figure for the animation cannot be registered.
	A chuck figure for animated simulation cannot be registered. It is likely
	that the system environment has a problem, such as insufficient memory.
	Check the program and system state, then restart the system.
NCOUT-E7011	The tailstock figure is not correct.
	The tailstock figure is incorrect. Check the tailstock figure on the Setting
	of Pre-machining window.
NCOUT-E7012	The tailstock figure for the animation cannot be registered.
	The tailstock figure is incorrect. Check the tailstock figure on the Setting
	of Pre-machining window.
NCOUT-E7013	The whole turning parts figure cannot be shown as it has too many
	elements.
	Because a figure consists of too many parts figure entities, it is impossible
	to draw the entire figure. The output NC data is correct, however.
NCOUT-E7014	The whole blank figure cannot be shown as it has too many elements.
	Because a figure consists of too many blank figure entities, it is impossible
	to draw the entire figure. The output NC data is correct, however.
NCOUT-E7099	The error which was not able to be expected occurred.
	An unexpected error, such as a calculation overflow, occurred. It is likely
	that there is insufficient memory. Check the program, then restart the
	system.

26.3.7 Messages concerning the milling library

[Abbreviation RGMILLIB] Messages concerning the milling library [C/Y]		[C/Y]
Abbreviation-number	Message (Description)	
[Error]		
RGMILLIB-E0001	The file cannot be found. Check whether the file name a	nd path name
	are correct, and also whether the drive is ready.	[C/Y]
	The text file format is incorrect.	
RGMILLIB-E0002	The format of the milling library file is incorrect.	A file of an
	incorrect format is identified by the message displayed	l immediately
	before this message.	[C/Y]

RGMILLIB-E0003	There is a file having the same name. [C/Y]
RGMILLIB-E0004	Data to be processed cannot be found.[C/Y]
	Data to be acquired or deleted could not be found. This error may be
	caused by a system environment problem such as there being insufficient
	memory. After checking the system status, restart the system.
RGMILLIB-E0005	The amount of memory is insufficient.[C/Y]
	After checking the system memory status, restart the system.
RGMILLIB-E0007	An unexpected error occurred. [C/Y]
	This error may be caused by a system environment problem such as there
	being insufficient memory. After checking the system status, restart the system.
RGMILLIB-E0008	The disk does not have sufficient free space. [C/Y]
	The disk does not have sufficient free space for writing to a text file.
	Check if the directory of the milling library specified in the operating
	environment setting data has sufficient free space.
RGMILLIB-E0013	An attempt to delete a file failed. Check if the file name and path
	name are correct, if the drive is ready, and if the file is write-
	protected.
RGMILLIB-E0014	An attempt to change the file name failed. [C/Y]
	An attempt to change the file name of the milling library file failed.
	Check if the directory of the milling library specified in the operating
	environment setting data is write-protected.
RGMILLIB-E0015	An attempt to open the file for writing failed. [C/Y]
	An attempt to open the milling library file for writing failed. Check if the
	directory of the milling library specified in the operating environment
	setting data is write-protected, and if the write area has sufficient free
	space.
RGMILLIB-E0023	An attempt to create work data for the milling library failed. [C/Y]
	An attempt to create work data for the milling library failed. The message
	displayed after this message indicates a cause. The failure may have
	occurred as a result of one of the causes listed below.
	(1) The directory of the milling library specified in the operating
	environment setting data does not contain a milling library file
	(XXXXX.MML). Check if a milling library file is contained.
	(2) The format of the milling library file is incorrect.
RGMILLIB-E0024	An attempt to allocate a work data area failed. [C/Y]
	An attempt to allocate a work area for the milling library data failed. The
	message displayed after this message indicates a cause. The failure may
	have occurred as a result of one of the causes listed below.
	(1) The previous work data area still remains for some reason. Restart the system.
	(2) There is a system environment problem such as there being
	insufficient memory. After checking the system status, restart the
	system.

RGMILLIB-E0025	Select only one data item to be modified. [C/Y]
	Multiple milling data items were selected when the user clicked the
	Modify button on the milling data name list window. From the milling
	data name list, select only one milling data item to be modified.
RGMILLIB-E0030	Milling data having the same name has already been created. [C/Y]
	Milling data having the same name was registered when milling data was
	registered or modified. Modify the milling data name.
RGMILLIB-E0031	An attempt to add or modify milling data failed. [C/Y]
	There may be a system environment problem such as there being
	insufficient memory. After checking the system status, restart the system.
RGMILLIB-E0032	Specify a milling data name. [C/Y]
	When milling data was registered or modified, no milling data name was
	specified. Specify a milling data name.
RGMILLIB-E0033	Select only one list for insertion position specification. [C/Y]
	Multiple machining process lists were selected when the user clicked the
	Insert button on the milling process editing window. Select only one
	machining process list for insertion position specification.
RGMILLIB-E0034	An attempt to add or modify milling process data failed. [C/Y]
	There may be a system environment problem such as there being
	insufficient memory. After checking the system status, restart the system.
[Warning]	
RGMILLIB-I0001	The order of center drilling and drilling is reversed. [C/Y]
	Process data editing resulted in an inadequate machining process. Modify
	the machining process.
RGMILLIB-I0002	The order of center drilling and drilling is reversed. Do you wish to
	modify the process? [C/Y]
	When the user clicked the OK button on the milling process editing
	window, an inadequate process was registered. Modify the machining
	process.
RGMILLIB-I0003	The order of preceding machining and roughing is reversed. [C/Y]
	Process data editing resulted in an inadequate machining process. Modify
	the machining process.
RGMILLIB-I0004	The order of preceding machining and roughing is reversed. Do you
	wish to modify the process? [C/Y]
	When the user clicked the OK button on the milling process editing
	window, an inadequate process was registered. Modify the machining
	process.
RGMILLIB-I0005	The order of preceding machining and finishing is reversed. [C/Y]
	Process data editing resulted in an inadequate machining process. Modify
	the machining process.
RGMILLIB-I0006	The order of preceding machining and finishing is reversed. Do you
	wish to modify the process? [C/Y]
	When the user clicked the OK button on the milling process editing
	window, an inadequate process was registered. Modify the machining
	process.

RGMILLIB-I0007	The order of roughing and finishing is reversed.	[C/Y]
	Process data editing resulted in an inadequate machining process.	Modify
	the machining process.	
RGMILLIB-I0008	The order of roughing and finishing is reversed. Do you	wish to
	modify the process?	[C/Y]
	When the user clicked the OK button on the milling process	editing
	window, an inadequate process was registered. Modify the ma	chining
	process.	

26.3.8 Messages related to loading part files

Abbreviation-number	Message (Explanation)
[Error]	
RGSTCOM-E0001	The file is not found. Confirm whether the filename/pathname to be
	correct, and the drive to be ready.
	Check that the file is in the directory where the cutting condition file, too
	file, and tooling file are saved, as specified during environment setting.
RGSTCOM-E0002	The format of a text file is not correct.
	The format of a cutting condition file is incorrect. The file is indicated in a
	message that precedes this message.
RGSTCOM-E003	The same name file found.
RGSTCOM-E004	There is no data.
	Data specified to be obtained or erased is missing. It is likely that the
	system environment has a problem, such as insufficient memory. Check
	the system state, then restart the system.
RGSTCOM-E0005	Insufficient memory.
	Check the state of system memory, then restart the system.
RGSTCOM-E0007	The error not anticipated occurred.
	It is likely that the system environment has a problem, such as insufficien
	memory. Check the program and system state, then restart the system.
RGSTCOM-E0008	No enough capacity in the disk.
	An attempt to save a text file failed, because there is no sufficient free
	space on the disk. Check that there is sufficient free space in the directory
	where the cutting condition file, tool file, and tooling file are saved, as
	specified during environment setting.
RGSTCOM-E0013	Failed in the deletion of the file. Confirm whether the
	filename/pathname to be correct, the drive to be ready, and no read-
	only.
RGSTCOM-E0014	Failed in the change in the file name.
	An attempt to change the file name of a cutting condition file, tool file, or
	tooling file failed. Check whether the directory where the cutting
	condition, tool, and tooling files are saved, as specified during
	environment setting, is write-protected.

RGSTCOM-E0015	Failed in the opening of the file for writing.
	An attempt to open a cutting condition file, tool file, or tooling file for
	write access failed. Check whether the directory where the cutting
	condition file, tool file, and tooling file are saved, as specified during
	environment setting, is write-protected and that there is sufficient write
	area.
RGSTCOM-E0021	Failed in making the work data for tooling file.
	An attempt to create working data for a tooling file failed. A message next
	to this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the tooling file (xxxxx.TSF) is not in the directory
	where the tool and tooling files are saved, as specified during
	environment setting. Check that the tooling file is in that directory.
	(2) It is likely that the format of the tooling file is incorrect.
RGSTCOM-E0022	Failed in the preservation of the tooling file.
	An attempt to save a tooling file failed. It is likely that there is no
	sufficient free space in the directory where the tool and tooling files are
	saved, as specified during environment setting.
RGSTCOM-E0023	Not Found tooling data. Regist tooling data.
	A tool cannot be referenced during machining definition, because there is
	no tooling data. Register tooling data by selecting Setting => Library Data
	Setting => Tool, Tooling, from the menu, then select the target tooling
	data using the Setting of Pre-machining window.
RGSTCOM-E0024	The tooling main file cannot be read.
	An attempt to load a tooling management file failed. A message next to
	this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the tooling management file (TLSTMAIN.TSF) is not
	in the directory where the tool and tooling files are saved, as specified
	during environment setting. Check that the tooling Management file
	is in that directory.
	(2) It is likely that the format of the tooling management file is incorrect.
RGSTCOM-E0031	Failed in making the work data for work material data.
	An attempt to create workpiece material working data failed. A message
	next to this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the workpiece material file (xxxxx.CNA, xxxxx.CDC,
	or xxxxx.CTD) is not in the directory where the cutting condition file
	is saved, as specified during environment setting. Check that the
	workpiece material file is in that directory.
	(2) It is likely that the format of the workpiece material file is incorrect.
RGSTCOM-E0032	Failed in the preservation of the work material file.
	An attempt to save a workpiece material file failed. It is likely that there is
	no sufficient free space in the directory where the cutting condition file is
	saved, as specified during environment setting.
RGSTCOM-E0033	Not Found work material data. Regist work material data.
	Cutting conditions cannot be set up automatically during machining
	definition, because there is no workpiece material data. Register workpiece
	-30/-

material data by selecting Setting => Library Data Setting => Machining Condition from the menu, then select the target material using the Blank Figure window.

RGSTCOM-E0034

[Warning]

The work material main file cannot be read.

An attempt to load a workpiece material management file failed. A message next to this one indicates the cause of the failure. Probable causes are:

- (1) It is likely that the workpiece material management file (CONDMAIN.CCF) is not in the directory where the cutting condition file is saved, as specified during environment setting. Check that the workpiece material management file is in that directory.
- (2) It is likely that the format of the workpiece material management file is incorrect.

RGSTCOM-I0001Not Found tooling data of using part file. If you want to use the tooling
data of using part file, regist and select tooling data.

The tooling data used in a loaded part file cannot be referenced during machining definition, because it is not registered with the system. To use the tooling data used in a part file, it is necessary to register the tooling data by selecting Setting => Library Data Setting => Tool, Tooling from the menu, then select the target tooling data using the Setting of Premachining window. If the tooling data is not registered or selected, the one used before the part file is loaded will be referenced during machining definition.

RGSTCOM-I0002 Not Found tooling data of using part file. If you want to use the tooling data of using part file, regist and select tooling data.

The tooling data used in a part file cannot be referenced during machining definition, because it is not saved in the directory where the tool and tooling files are saved, as specified during environment setting. To use the tooling data used in a part file, it is necessary to register the tooling data by selecting Setting => Library Data Setting => Tool, Tooling from the menu, then select the target tooling data using the Setting of Premachining window. If the tooling data is not registered or selected, the first tooling data saved in the directory where the tool and tooling files are saved will be referenced during machining definition.

RGSTCOM-I0003Not Found work material data of using part file. If you want to use the
work material data of using part file, regist and select work material
data.

The workpiece material data used in a loaded part file cannot be referenced during machining definition, because it is not registered with the system. To use the workpiece material data used in a part file, it is necessary to register the workpiece material data by selecting Setting => Library Data Setting => Machining Condition from the menu, then select the target -368-

material data using the Blank Figure window. If the material data is not registered or selected, the one used before the part file is loaded will be referenced during machining definition.

RGSTCOM-I0004Not Found work material data of using part file. If you want to use the
work material data of using part file, regist and select work material
data.

The workpiece material data used in a part file cannot be referenced during machining definition, because it is not saved in the directory where the cutting condition file is saved, as specified during environment setting. To use the workpiece material data used in a part file, it is necessary to register the workpiece material data by selecting Setting => Library Data Setting => Machining Condition from the menu, then select the target material data using the Blank Figure window. If the workpiece material data saved in the directory where the cutting condition file is saved will be referenced during machining definition.

26.3.9 Messages related to cutting condition setting

[Abbreviation **RGSTCOND**] ----- Messages concerning cutting condition setting

Abbreviation-number	Message (Explanation)
[Error]	
RGSTCOND-E0001	The file is not found. Confirm whether the filename/pathname to be
	correct, and the drive to be ready.
	Check that the file name/path name is correct. Also check that the drive is
	ready.
RGSTCOND-E0002	The format of the text file is not correct.
	The format of a cutting condition file is incorrect. The file is indicated in
	the message that precedes this one.
RGSTCOND-E0003	The same name file found.
RGSTCOND-E0004	There is no data.
	Data to be acquired or erased was found missing. It is likely that the
	system environment has a problem, such as insufficient memory. Check
	the system state, then restart the system.
RGSTCOND-E0005	Insufficient memory.
	Check the state of system memory, and restart the system.
RGSTCOND-E0007	The error not anticipated occurred.
	It is likely that the system environment has a problem, such as insufficient
	memory. Check the system state, then restart the system.
RGSTCOND-E0008	No enough capacity in the disk.
	An attempt to save a text file failed, because there is no sufficient free
	space on the disk. Check that there is sufficient free space in the directory
	where the cutting condition file is saved, as specified during environment
	setting.

RGSTCOND-E0013	Failed in the deletion of the file. Confirm whether the filename/
	pathname to be correct, the drive to be ready, and no read-only.
	Check that the specified file name/path name is correct, that the drive is
	ready, and whether the disk is write-protected.
RGSTCOND-E0014	Failed in the change in the file name.
	An attempt to change the file name of a cutting condition file failed. Check
	whether the directory where the cutting condition file is saved, as specified
	during environment setting, is write-protected.
RGSTCOND-E0015	Failed in the opening of the file for writing.
	An attempt to open a cutting condition file for write access failed. Check
	whether the directory where the cutting condition file is saved, as specified
	during environment setting, is write-protected and whether there is
	sufficient write area.
RGSTCOND-E0021	The area of the work data cannot be secured.
	An attempt to reserve a work area for cutting condition data failed during
	machining definition or cutting condition registration. A message next to
	this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the previous working data area is lingering for some
	reason. Restart the system.
	(2) It is likely that the system environment has a problem, such as
	insufficient memory. Check the system state, then restart the system.
RGSTCOND-E0022	The work material main file cannot be read.
	An attempt to load a workpiece material management file failed. A
	message next to this one indicates the cause of the failure. Probable causes
	are:
	(1) It is likely that the workpiece material management file
	(CONDMAIN.CCF) is not in the directory where the cutting
	condition file is saved, as specified during environment setting.
	Check that the workpiece material management file is in that
	directory
	(2) It is likely that the format of the workpiece material management file
	is incorrect
RGSTCOND-F0023	The work material name file cannot be read
	An attempt to load a workpiece material name file failed. A message next
	to this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the workpiece material name file (xxxxx NCA) is not
	in the directory where the cutting condition file is saved as specified
	during environment setting. Check that the workniece material name
	file is in that directory
	(2) It is likely that the format of the workpiece material name file is
	(2) It is neery that the format of the workprece material name me is
RCSTCOND.F0024	The work material file cannot be conied in a fived directry
	When you selected the Reading of Cutting Condition File option on the
	workniece material addition window and clicked OK the outting
	workpress material addition window, and cheked OK, the cutting

	condition file was not copied to the directory where the cutting condition
	file was saved, as specified during environment setting.
	Check whether the directory where the cutting condition file is saved is
	write-protected and that there is sufficient write area.
RGSTCOND-E0025	Failed in a new making the work material file.
	When you selected the New Making of Cutting Condition File option on
	the material addition window, and clicked OK, the cutting condition file
	was not created in the directory where the cutting condition file was saved,
	as specified during environment setting.
	Check whether the directory where the cutting condition file is saved is
	write-protected and that there is sufficient write area.
RGSTCOND-E0026	It is possible to write the data in the work material main file.
	An attempt to save a workpiece material management file failed. A
	message next to this one indicates the cause of the failure. Probable causes
	are:
	(1) It is likely that there is no sufficient free space in the directory where
	the cutting condition file is saved, as specified during environment
	setting.
	(2) It is likely that the format of the material management file (COND
	MAIN.CCF) is incorrect.
RGSTCOND-E0029	Failed in making the work data of the work material file.
	An attempt to create workpiece material working data failed. A message
	next to this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the workpiece material file (xxxxx.CNA, xxxxx.CDC,
	or xxxx.CTD) is not in the directory where the cutting condition file
	is saved, as specified during environment setting. Check that the
	workpiece material file is in that directory.
DOCTOND DAAA	(2) It is likely that the format of the workpiece material file is incorrect.
KGSTCOND-E0030	Failed in the preservation of the work material file.
	An attempt to save a workpiece material file failed. It is likely that there is
	no sufficient free space in the directory where the cutting condition free is
DOSTONID FAA21	saved, as specified during environment setting.
KGS1CUND-E0051	An attempt to save a workpiece meterial name file failed. A massage next
	to this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that there is no sufficient free space in the directory where
	the cutting condition file is saved as specified during environment
	setting
	(2) It is likely that the format of the workpiece material name file
	(xxxxx.CNA) is incorrect.
	An attempt to discard workpiece material working data failed. A message
	next to this one indicates the cause of the failure. A probable cause is:
	It is likely that the directory where the cutting condition file is saved. as
	specified during environment setting, has no sufficient free space to hold
	the working data text file.

RGSTCOND-E0032	An attempt to discard workpiece material work data failed.
	An attempt to discard workpiece material work data failed. The message
	displayed after this message indicates a cause. The failure may have
	occurred as a result of the following cause:
	When work data is written into a text file, no sufficient write space is
	available under the directory of the cutting condition file specified in the
	operating environment setting data.
RGSTCOND-E0033	Failed in the deletion of machining type data.
	The Erase button was clicked to erase the machining type data from the
	Machining Type Selection window, but it was not erased. It is likely that
	the system environment has a problem, such as insufficient memory.
	Check the system state, then restart the system.
RGSTCOND-E0034	Failed in the addition of machining type list.
	An attempt was made to add a workpiece material machining type to the
	list on the Machining Type Selection window, but it was not added. It is
	likely that the system environment has a problem, such as insufficient
	memory. Check the system state, then restart the system.
RGSTCOND-E0036	The tool type main file cannot be read.
	An attempt to load a tool type management file failed. A message next to
	this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the tool type management file (TOOLMAIN.TLF) is
	not in the directory where the tool and tooling files are saved, as
	specified during environment setting. Check that there is the tool type
	management file in that directory
	(2) It is likely that the format of the tool type management file is
	incorrect
RGSTCOND-E0037	Failed in the acquisition of the finish allowance data.
	An attempt to obtain the finishing allowance data failed when the Tool
	Type Selection window was displayed. It is likely that the system
	environment has a problem such as insufficient memory Check the
	system state then restart the system
RGSTCOND-F0038	Failed in the preservation of the finish allowance data
	An attempt to save the finishing allowance data failed when OK was
	clicked to exit the Tool Type Selection window. It is likely that the system
	environment has a problem such as insufficient memory Check the
	system state then restart the system
RGSTCOND-F0039	Failed in the acquisition of the tool type and tool material name data.
	An attempt to obtain the tool type and tool material name data failed.
	when the cutting condition data was processed. It is likely that the system
	environment has a problem such as insufficient memory Check the
	system state then restart the system
RGSTCOND-E0040	Failed in the deletion of the tool type.
	The Erase button was clicked to erase the tool type data from the Type
	Tool Selection window but it was not erased. It is likely that the system
	environment has a problem such as insufficient memory. Check the
	environment has a problem, such as insufficient memory. Check the

	system state, then restart the system.
RGSTCOND-E0041	Failed in the order change of the tool type.
	On the Tool Type Selection window, an attempt to change the sequence of
	the tool type data failed. It is likely that the system environment has a
	problem, such as insufficient memory. Check the system state, then restart
	the system.
RGSTCOND-E0043	Failed in the acquisition of the tool material data of tool type main file.
	An attempt to obtain the tool material name data from a tool type
	management file failed. A message next to this one indicates the cause of
	the failure. Probable causes are:
	(1) It is likely that the tool type management file (TOOLMAIN.TLF) is
	not in the directory where the tool and tooling files are saved, as
	specified during environment setting. Check that there is the tool
	type management file in that directory.
	(2) It is likely that the format of the tool type management file is
	incorrect.
RGSTCOND-E0044	Failed in the addition of tool diameter - cutting speed list.
	On the Cutting Condition Registration window, an attempt to add the tool
	diameter versus cutting speed data failed. It is likely that the system
	environment has a problem, such as insufficient memory. Check the
	system state, then restart the system.
RGSTCOND-E0045	Failed in the acquisition of the cutting condition data.
	On the Cutting Condition Registration window, an attempt to obtain the
	cutting condition data failed. It is likely that the system environment has a
	problem, such as insufficient memory. Check the system state, then restart
	the system.
RGSTCOND-E0046	Failed in the preservation of the cutting condition data.
	On the Cutting Condition Registration window, an attempt to save the
	cutting condition data failed. It is likely that the system environment has a
	problem, such as insufficient memory. Check the system state, then restart
	the system.
RGSTCOND-E0101	Select the added condition data from the list.
	When the Reading of Cutting Condition File option was selected from the
	workpiece material addition window, and OK was clicked, the desired
	additional cutting condition was not selected. Select it from the list.
RGSTCOND-E0102	Specify the work material name.
	When the New Making of Cutting Condition File option was selected from
	the workpiece material addition window, and OK was clicked, the
	workpiece material name of the cutting condition data to be prepared was
	not specified. Specify the workpiece material name.
RGSTCOND-E0103	Specify the file name.
	When the New Making of Cutting Condition File option was selected from
	the workpiece material addition window, and OK was clicked, the name of
	the cutting condition file to be created was not specified. Specify the file

	name.
RGSTCOND-E0104	An illigal character is included in the file name.
	The name of the cutting condition file contains an invalid character.
	Correct the file name.
RGSTCOND-E0105	Specify the path which the cutting condition data.
	On the workpiece material addition window, when the List Display button
	was clicked, the directory for which the contents are to be listed was not
	specified. Specify the directory.
RGSTCOND-E0106	Select only one list which specifies the insertion position.
	When the Insert button for the tool type data was clicked, it turned out that
	more than one tool type had been selected from the tool type list. Select a
	tool type as the insertion position; do not select more than one.
RGSTCOND-E0107	Select only one machining type which shoud be corrected.
	On the Machining Type Selection window, when the Modify button was
	clicked, it turned out that more than one machining type had been selected.
	Select only one machining type to be modified from the machining type
	list.
RGSTCOND-E0108	Select data which shoud be deleted.
	When the Erase button was clicked, it turned out that the data to be erased
	had not been selected. Select it.
RGSTCOND-E0109	Select a consecutive list.
	When the Change Order button for tool types was clicked, it turned out
	that the tool types had been selected discontinuously. The sequence of two
	or more tool types can be changed simultaneously, only when their tool
	type data is continuous on the list. Select the tool type data that is
	continuous on the list.
RGSTCOND-E0110	Specify the position after moving correctly.
	During tool type sequence change, any of the tool types whose sequence
	was to be changed was selected as the after-change position. Select a tool
	type other than those whose sequence is to be changed, as the after-change
	position.
RGSTCOND-E0111	Select the data to which the order is changed from the list.
	When the Change Order button for tool types was clicked, it turned out
	that the target tool types had not been selected. Select the target tool types
	from the tool type list.
RGSTCOND-E0112	Select only one tool type which shoud be corrected.
	On the Tool Type Selection window, when the Modify button was clicked,
	it turned out that more than one machining type list had been selected.
	Specify only one tool type to be modified from the tool type list.
RGSTCOND-E0113	This Data has already been registered.
	When an attempt was made to register a cutting condition, it turned out
	that the cutting condition for the target tool type and material had already
	been registered. Register a cutting condition for a different tool type and
	material.

RGSTCOND-E0114	Specify the tool material.
	When an attempt was made to add tool type data, it turned out that the
	relevant tool material had not been selected. Select the tool material from
	the tool material list. If it is necessary to add the tool material, select
	Setting => Library Data Setting => Tool, Tooling from the menu.
RGSTCOND-E0115	Specify the tool type.
	When an attempt was made to add tool type data, it turned out that the
	relevant tool type had not been selected. Select the tool type from the tool
	type list.
RGSTCOND-E0116	Select only one data which shoud be corrected.
	When the Modify button for tool diameter versus cutting speed was
	clicked, it turned out that the target data had not been selected, or two or
	more data items had been selected. Select only one data item to be
	modified.

26.3.10 Messages related to tool and tooling data

[Abbreviation RGSTTOOL] ----- Messages concerning tool/tooling data setting

Abbreviation-number	Message (Explanation)
[Error]	
RGSTTOOL-E0001	The file is not found. Confirm whether the filename/pathname to be
	correct, and the drive to be ready.
	Check that the file name/path name is correct and that the drive is ready.
RGSTTOOL-E0002	The format of the text file is not correct.
	The format of a tool or tooling file is incorrect. The file is indicated in the
	message that precedes this one.
RGSTTOOL-E0003	The same name file found.
RGSTTOOL-E0004	There is no data.
	Data to be acquired or erased was found missing. It is likely that the
	system environment has a problem, such as insufficient memory. Check
	the system state, then restart the system.
RGSTTOOL-E0005	Insufficient memory.
	Check the state of system memory, and restart the system.
RGSTTOOL-E0007	The error not anticipated occurred.
	It is likely that the system environment has a problem, such as insufficient
	memory. Check the program and system state, then restart the system.
RGSTTOOL-E0008	No enough capacity in the disk.
	An attempt to save a text file failed, because there was no sufficient free
	space on the disk. Check that there is sufficient free space in the directory
	where the tool and tooling files are saved, as specified during environment
	setting.
RGSTTOOL-E0012	The same tool name has already been registered.
	When an attempt was made to register or modify the data of a tool, it
	turned out that another tool having the same name as the target tool had

	been registered. Change the tool name.
RGSTTOOL-E0013	Failed in the deletion of the file. Confirm whether the filename/
	pathname to be correct, the drive to be ready, and no read-only.
	Check that the specified file name/path name is correct, that the drive is
	ready, and whether the disk is write-protected.
RGSTTOOL-E0015	Failed in the opening of the file for writing.
	An attempt to open a cutting condition file for write access failed. Check
	that the directory where the tool and tooling files are saved, as specified
	during environment setting, is not write-protected and that there is
	sufficient write area.
RGSTTOOL-E0021	The tooling main file cannot be read.
	An attempt to load the tool management file failed. A message next to this
	one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the tooling management file (TLSTMAIN.TSF) is not
	in the directory where the tool and tooling files are saved, as specified
	during environment setting. Check that there is the tooling
	management file in that directory.
	(2) It is likely that the format of the tooling management file is incorrect.
RGSTTOOL-E0024	The tool type main file cannot be read.
	An attempt to load a tool type management file failed. A message next to
	this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the tool type management file (TOOLMAIN.TLF) is
	not in the directory where the tool and tooling files are saved, as
	specified during environment setting. Check that there is the tool
	type management file in that directory.
	(2) It is likely that the format of the tool type management file is
	incorrect.
RGSTTOOL-E0027	The area of the work data cannot be secured.
	An attempt to reserve a work area for tool/tooling data failed during
	machining definition or tool/tooling registration. A message next to this
	one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the previous working data area is lingering for some
	reason. Restart the system.
	(2) It is likely that the system environment has a problem, such as
	insufficient memory. Check the system state, then restart the system.
RGSTTOOL-E0028	Failed in making the work data for tool file.
	An attempt to create working data for a tool file failed. A message next to
	this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the tool file (xxxxx.TLF) is not in the directory where
	the tool and tooling files are saved, as specified during environment
	setting. Check that the tool file is in that directory.
	(2) It is likely that the format of the tool file is incorrect.
RGSTTOOL-E0029	Failed in the acquisition of the tool data.(tool file)
	An attempt to read tool data from a tool file failed. It is likely that the
	system environment has a problem, such as insufficient memory. Check
	the system state, then restart the system.
RGSTTOOL-E0031	Failed in making the work data for tooling file.
--	--
	An attempt to create working data in a tooling file failed. A message next
	to this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that the tooling file (xxxxx.TSF) is not in the directory
	where the tool and tooling files are saved, as specified during
	environment setting. Check that the tooling file is in that directory.
	(2) It is likely that the format of the tooling file is incorrect.
RGSTTOOL-E0032	Failed in the acquisition of the tool data (tooling file)
100110022002	An attempt to read tool data from a tooling file failed. It is likely that the
	system environment has a problem such as insufficient memory Check
	the system state and restart the system
DOSTTOOL FAA3	Failed in the addition of tooling list
K051100L-L0055	When displaying the tool date or tooling list window, an attempt to add
	when displaying the tool data of tooling list window, an attempt to add
	data to the tooling list failed. It is likely that the system environment has a
	problem, such as insufficient memory. Check the system state, and restart
	the system.
RGSTTOOL-E0035	Failed in writing the tool type main file.
	An attempt to save a tool type management file failed. A message next to
	this one indicates the cause of the failure. Probable causes are:
	(1) It is likely that there is no sufficient free space in the directory where
	the tooling and tool files are saved, as specified during environment
	setting.
	(2) It is likely that the format of the tool type management file is
	incorrect
RGSTTOOL-E0036	Failed in the preservation of the tool file.
RGSTTOOL-E0036	Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient
RGSTTOOL-E0036	Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved as
RGSTTOOL-E0036	Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as
RGSTTOOL-E0036	Failed in the preservation of the tool file.An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting.Failed in the preservation of the tooling file
RGSTTOOL-E0036 RGSTTOOL-E0037	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file.
RGSTTOOL-E0036 RGSTTOOL-E0037	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no
RGSTTOOL-E0036 RGSTTOOL-E0037	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are
RGSTTOOL-E0036 RGSTTOOL-E0037	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting.
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name.
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure.
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are:
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: (1) It is likely that there is no sufficient free space in the directory where
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: (1) It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: (1) It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting.
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: (1) It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting. (2) It is likely that the format of the tooling management file
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: (1) It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting. (2) It is likely that the format of the tooling management file (TLSTMAIN TSE) or tooling file (xxxxx TSE) is incorrect.
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting. (2) It is likely that the format of the tooling management file (TLSTMAIN.TSF) or tooling file (xxxxx.TSF) is incorrect.
RGSTTOOL-E0036 RGSTTOOL-E0037 RGSTTOOL-E0038	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting. (2) It is likely that the format of the tooling management file (TLSTMAIN.TSF) or tooling file (xxxx.TSF) is incorrect. Failed in writing the tooling management file failed. A message next to the save a tooling management file failed. A message next to save a tooling file.
RGSTTOOL-E0037 RGSTTOOL-E0038 RGSTTOOL-E0039	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting. (2) It is likely that the format of the tooling management file (TLSTMAIN.TSF) or tooling file (xxxxx.TSF) is incorrect. Failed in writing the tooling management file failed. A message next to this one indicates the cause of the failer.
RGSTTOOL-E0037 RGSTTOOL-E0038 RGSTTOOL-E0039	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the change in the tooling name. An attempt to change the tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: (1) It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting. (2) It is likely that the format of the tooling management file (TLSTMAIN.TSF) or tooling file (xxxxx.TSF) is incorrect. Failed in writing the tooling management file failed. A message next to this one indicates the cause of the failure. Probable causes are: (1) It is likely that the format of the tooling management file (TLSTMAIN.TSF) or tooling file.
RGSTTOOL-E0037 RGSTTOOL-E0038 RGSTTOOL-E0039	 Failed in the preservation of the tool file. An attempt to save a tool file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling file. An attempt to save a tooling file failed. It is likely that there is no sufficient free space in the directory where the tool and tooling files are saved, as specified during environment setting. Failed in the preservation of the tooling name. An attempt to save a tooling name of tooling management and tooling files failed. A message next to this one indicates the cause of the failure. Probable causes are: It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting. (2) It is likely that the format of the tooling management file (TLSTMAIN.TSF) or tooling file (xxxxx.TSF) is incorrect. Failed in writing the tooling management file failed. A message next to this one indicates are: It is likely that there is no sufficient free space in the directory where the tooling and tool files are saved, as specified during environment setting.

RGSTTOOL-E0104

the tool and tooling files are saved, as specified during environment setting.

(2) It is likely that the format of the tooling management file (TLSTMAIN.TLF) is incorrect.

RGSTTOOL-E0040The tooling file cannot be copied in a fixed directry.
When the file name/path name was correct, and the drive was ready, an
attempt to copy a tooling file to the directory where the tool and tooling
files were saved, as specified during environment setting, failed. Check
whether that directory is write-protected and that there is sufficient write
area.RGSTTOOL-E0041Failed in a new making the tooling file.

On the tooling addition window, when the New Making of Tooling File option was selected, and OK was clicked, the tooling file was not created in the directory where the tool and tooling files were saved, as specified during environment setting.

Check whether that directory is write-protected and that there is sufficient write area.

RGSTTOOL-E0101This data is a current tooling data that refer to machining definition.
Do not delete this data.

It is impossible to erase the tooling data that is being referenced by machining definition.

RGSTTOOL-E0102 Select tooling name from list.

On the tooling addition window, when the Reading of Tooling File option was selected, and OK was clicked, it turned out that the tooling data to be added had not been selected. Select the tooling data to be added from a list.

 RGSTTOOL-E0103
 Specify the tooling name.

 On the tooling addition window, when the New Making of Tooling File

option was selected, and OK was clicked, it turned out that the tooling name of the tooling data to be created had not been selected. Select the tooling name.

Specify the file name. On the tooling addition window, when the New Making of Tooling File option was selected, and OK was clicked, it turned out that the name of the tooling file to be created had not been selected. Select the file name.

RGSTTOOL-E0105 An illigal character is included in the file name.

The name of a tooling file contains an invalid character. Specify a valid file name.

RGSTTOOL-E0106 Specify the path which the tooling data.

On the tooling addition window, when the List Display button was clicked, it turned out that the directory for which the contents are to be listed had not been specified. Specify the directory.

RGSTTOOL-E0107 Select only one list from tool data which specifies the insertion position.

	When the Insert option for the tool data was selected, and the New Make
	button for tool data was clicked, it turned out that no tool in the tool data
	list had been selected as the insertion position or that more than one tool
	had been selected. Select only one tool from the list.
RGSTTOOL-E0108	Select only one tool data which shoud be corrected.
	When the Modify button for the tool data or tooling data was clicked, it
	turned out that more than one tool had been selected. Select only one tool
	to be modified from the tool data list or tooling data list.
RGSTTOOL-E0109	Select data which shoud be deleted.
	When the Erase button was clicked, it turned out that data to be erased had
	not been selected. Select the data to be erased.
RGSTTOOL-E0110	Select only one list from tooling data which specifies the insertion posi-
	tion.
	When the Insert option for tooling data was selected, and the New Make
	button for tooling data or the Copy to Tooling Data button for tool data
	was clicked, it turned out that no tooling in the tooling data list had not
	been selected as the insertion position or that more than one tooling had
	been selected. Select only one tooling from the list.
RGSTTOOL-E0111	Select tool from tool data which should be copied in the tooling data.
	When the Copy to Tooling Data button for tool data was clicked, it turned
	out that the tool whose tool data was to be copied had not been selected
	from the tool data list. Select the target tool from the tool data list.
RGSTTOOL-E0112	Specify the tool material name.
	On the tool material addition window, when OK was clicked, it turned out
	that no tool material name had been specified. Specify one.

26.3.11 Other messages

(1) Messages displayed during setup

Message	(Explanation)
---------	---------------

• One or more Visual Basic applications are running. Please close those applications, then choose OK to continue.

Applications created using Visual Basic are running, disabling further processing to occur. If you know which application is the cause of the failure, take measures as directed. If you do not, stop all applications other than Program Manager.

• Could not make the installation path.

It is likely that the drive (for example, c:) specified in the path name does not exist, the target drive is write-protected (for example, because it is CD-ROM), the specified directory (for example, \CAPT) already exists and is write-protected, or the hard disk has no free space.

Check on the above information using File Manager, and specify an appropriate path name.

• Drive <d>: has not enough disk space. Need <size> bytes or more.

The system files are saved on the drive indicated by the path name specified as a location for installation and the Windows system drive. If this message appears, check for free disk space and erase unnecessary files, using File Manager, then specify an appropriate path name.

• Set Disk #<n>.

Insert the <nth> floppy disk in the drive. This message appears also when a floppy disk not specified is inserted.

<filename> is just used by other applications, so not possible to be installed to <pathname>.

One or more other applications are using the indicated file. So, the file cannot be replaced. Stop all applications other than Program Manager, and perform setup again.

• <filename> was not able to be installed to <pathname>.

The probable causes are: The indicated file already exists, and is writeprotected; the indicated file is not supported by Symbolic CAP T or at least its current version (for example, an English file having the same name has been installed for a different application); the hard disk specified by the path name has no free space; and there is insufficient memory.

When you are going to re-install the software in the same directory as before, make sure that the existing files are not write-protected, using File Manager or the like, or rename or erase the existing files if possible. Also check the size of the free space of the hard disk. To check the free space of memory, use, for example, the Help => About Program Manager menu of Program Manager. If other applications are running, stop them.

If it is impossible to solve the problem by any method, it is likely that the system floppy disk is abnormal. In this case, please contact the sales agency from which you purchased your Symbolic CAP T.

- Source File <filename> is not found.
- Source File <filename> is empty.

The system floppy disk is abnormal.

Please contact the sales agency from which you purchased your Symbolic CAP T.

• Protector of the Symbolic CAP T Basic module is not connected. Quit every process and connect the protector after turn off the power.

Connect the correct protector as directed.

If the message appears when the protector is already connected, check for a loose connection, dirty or broken pin. Also make sure that the printer cable is not broken and securely connected.

• <spec no.> spec no. is not correct. The spec no. of software is displayed in the label of the system floppy disk. Confirm it. (e.g. A08B-9310-J500#EN07 for PC FAPT Basic module)

The format of the specification No. entered from the keyboard is incorrect. When gathering a protector into another, it is necessary to enter the specification No. of the associated software. Enter the correct specification No. as directed. • <spec no.> spec no. is not correct or the specified protector is not connected.

The spec no. of software is displayed in the label of the system floppy disk. Confirm it. (e.g. A08B-9310-J500#EN07 for PC FAPT Basic module) If the protector is not connected, quit every process and connect the protector after turn off the power.

The protector that matches the key-entered specification No. is not connected. When gathering a protector into another, it is necessary to enter the specification No. of the associated software and to connect the protector that matches the specification No. Follow the instructions strictly.

• <spec no.> protect key was not possible to be gathered into the protector of Symbolic CAP T Basic module. Following causes are thought. Confirm whether the connected protectors are correct.

+ The key is already gathered to one.

+ Because the key has already gathered in protector other than Symbolic CAP T Basic module, it has not remained any longer.

Confirm the connected protector.

• Error occurred while transferring protect key.

Check the protector for a loose connection, dirty or broken pin. Also make sure that the printer cable is not broken and securely connected. If the printer cable in use is not a brand-name product, the problem may be solved by exchanging it with a brand-name product. If a protector in use is not a product of FANUC, detach it, and see what will happen. Also attach the detached protector to the farthest position from the computer, and see what will happen.

If it is impossible to solve the problem by any method, it is likely that the protector is abnormal. In this case, please contact the sales agency from which you purchased it.

(2)	Messages concer	rning figure selection for modification	[C/Y]
Me	ssage	(Description)	
•	No prepared ho	le figure can be selected.	[C/Y]
		This message is displayed when a prepared selection for modification.	I hole figure is selected in figure
(3)	Messages concer	rning machining profile selection	[C/Y]
Me	ssage	(Description)	
•	Machining data modification is machining data	a is defined for a modified machining profile. made, the machining data may become unsuital before creating NC data.	When a machining profile ble. So, be sure to check the [C/Y]

This message is displayed when a modification is made to a machining profile for which machining data is defined.

27 APPENDIX

27.1 Function Codes

27.1.1 NC data format setting function codes (special functions)

Symbol	Code value	Function	Remark
Nop	0x0000	No function	
FEEDnull	0x0100	Feed output	
FEEDsp	0x0200	Feed output	ISO:NULL
			EIA:Space

27.1.2 NC data format setting function codes (move command words)

Symbol	Code value	Function	Remark
Xmot	0x1000	X-axis output	
Ymot	0x1010	Y-axis output	
Zmot	0x1020	Z-axis output	
Icent	0x1090	Output of arc center (I)	
Jcent	0x10A0	Output of arc center (J)	
Kcent	0x10B0	Output of arc center (K)	
Rzx	0x10D0	Output of ZX-plane arc radius R	
Xref	0x1110	X-axis output of pass position at reference position return	
Yref	0x1120	Y-axis output of pass position at reference position return	
Zref	0x1130	Z-axis output of pass position at reference position return	
Icom	0x1140	Output of cutter compensation vector (I)	
Jcom	0x1150	Output of cutter compensation vector (J)	
Kcom	0x1160	Output of cutter compensation vector (K)	
POL_MOV1_X	0x1220	Plane first axis output during polar	
Y		coordinate interpolation	
POL_MOV2_X	0x1250	Plane second axis output during	
Y		polar coordinate interpolation	

Symbol	Code value	Function	Remark
POL_MOV3_X	0x1280	Plane third axis output during polar	
Y		coordinate interpolation	
POL_CENT1_	0x1320	Arc center first axis output during	
XY		polar coordinate interpolation	
POL_CENT2_	0x1350	Arc center second axis output	
XY		during polar coordinate interpolation	
POL_CIR_R_	0x1380	Arc radius R output during polar	
XY		coordinate interpolation	
CYL_MOV1_Y	0x1400	Plane first axis output during	
Z		cylindrical interpolation	
CYL_MOV2_Y	0x1430	Plane second axis output during	
Z		cylindrical interpolation	
CYL_R_YZ	0x1460	Cylinder radius output during	
		cylindrical interpolation	
CYL_R_ZERO	0x1490	Cylinder radius (R0) output at the	
_YZ		end of cylindrical interpolation	
CYL_CENT1_	0x1500	Arc center first axis output during	
YZ		cylindrical interpolation	
CYL_CENT2_	0x1530	Arc center second axis output	
YZ		during cylindrical interpolation	
CYL_CIR_R_Y	0x1560	Arc radius R output during	
Z		cylindrical interpolation	
Frev	0x1600	Feedrate (feed per rotation) output	
Fmin	0x1610	Feedrate (feed per minute) output	
Pdwell_sec	0x1620	Dwell command output (seconds)	
Pdwell_rev	0x1630	Dwell command output (rpm)	
Xorg	0x17C0	X-axis origin	
Zorg	0x17E0	Z-axis origin	
Zcycl	0x1800	Canned-cycle hole bottom	
		coordinates	
Rcycl	0x1810	Canned-cycle R position	
		coordinates	
Qpeck	0x1820	Depth of cut per canned pecking	
		cycle	
lsft	0x1830	Canned-cycle shift amount (I)	
Jsft	0x1860	Canned-cycle shift amount (J)	
Ksft	0x1850	Canned-cycle shift amount (K)	
Qsft	0x1860	Canned-cycle shift amount (Q)	
Pcycl	0x1870	Canned-cycle dwelling time	
		(specified in seconds) for hole	
		bottom	

Symbol	Code value	Function	Remark
Pcycl_rev	0x1880	Canned-cycle dwelling time	
		(specified in rpm) for hole bottom	
Xthrd_cycl_m	0x1900	Multiple-repetitive-threading-cycle	
ul		X-axis output	
Ythrd_cycl_m	0x1910	Multiple-repetitive-threading-cycle	
ul		Y-axis output	
Zthrd_cycl_m	0x1920	Multiple-repetitive-threading-cycle	
ul		Z-axis output	
lthrd_tapr	0x1930	Threading taper amount (X-axis	
		direction)	
Jthrd_tapr	0x1940	Threading taper amount (Y-axis	
		direction)	
Kthrd_tapr	0x1950	Threading taper amount (Z-axis	
		direction)	
Qthrd_cang	0x1960	Threading cut angle	
Fthrd_lead_X	0x1970	Threading lead (X-axis direction)	
Fthrd_lead_Y	0x1980	Threading lead (Y-axis direction)	
Fthrd_lead_Z	0x1990	Threading lead (Z-axis direction)	
Fhacthrd_	0x19A0	High-precision threading lead (X	
lead_X		axis)	
Fhacthrd_	0x19B0	High-precision threading lead (Y	
lead_Y		axis)	
Fhacthrd_	0x19C0	High-precision threading lead (Z	
lead_Z		axis)	
Athrd_tang	0x19D4	Thread ridge angle	
Kthrd_height	0x19E4	Thread ridge height	
Dthrd_cut	0x19F4	Depth of cut on the first threading	
		cycle	
Qthrd_mincut	0x1A04	Minimum depth of cut on a	
		threading cycle	
Rthrd_fallow	0x1A14	Finishing allowance on a	
		threading cycle	
Fmr	0x1C00	Feed per rotation/minute	
Fthrd_lead	0x1C10	Threading lead	
Fhacthrd_lead	0x1C20	High-precision threading lead	
Pdwell	0x1C30	Dwell command output (seconds	
		or rpm)	
Pcycl_sr	0x1C40	Canned-cycle dwelling time	
		(seconds or rpm) for hole bottom	

		Code value 0x1**a					
а		0	1	2	4	5	6
Output of the same		×	×	×	0	0	0
value as the pr	evious						
Sign output	'+' output	×	×	0	×	×	0
	'-' output	0	×	0	0	×	0

The output format varies according to the first digit of each code value. The meaning of each value is as listed below.

27.1.3 NC data format setting function codes (G codes)

Symbol	Code value	Function	Remark
G04	0x2000	Dwell	
G28	0x2010	Return to the reference position	
G29	0x2020	Return from the reference	
		position	
G30	0x2030	Return to the second, third, or	
		fourth reference position	
G92	0x2040	Coordinate system setting	
G10L2	0x20B0	Workpiece coordinate system	
		setting	
Gi_ret	0x20C0	Return piece to the initial level	
		point.	
Gr_ret	0x20D0	Return to the R-point level	
G_CYL_STRT	0x20E0	Start of cylindrical interpolation	
G_CYL_OFF	0x20F0	End of cylindrical interpolation	
Gthrd_mul	0x2100	Multiple repetitive threading	
		cycle	
G00	0x2200	Rapid traverse	
G01	0x2210	Linear interpolation	
G02	0x2220	Circular interpolation (CW)	
G03	0x2230	Circular interpolation (CCW)	
Gthrd_nrm	0x2260	Threading	
Gthrd_scr	0x2270	Threading (front)	
Gthrd_cycl_nrm	0x2280	Simple threading cycle	
Gthrd_cycl_scr	0x2290	Simple threading cycle (front)	
G19	0x2300	YZ plane selection	
G18	0x2310	ZX plane selection	
G17	0x2320	XY plane selection	
G90	0x2400	Absolute command	
G91	0x2410	Incremental command	
G94	0x2500	Feedrate (minutes)	

Symbol	Code value	Function	Remark
G95	0x2510	Feedrate (rpm)	
G21	0x2600	Metric command	
G20	0x2610	Inch command	
G40	0x2700	Tool-tip radius compensation	
		cancellation	
GcomL_zx	0x2720	Left-side tool-tip radius compen-	
		sation (ZX plane)	
GcomR_zx	0x2750	Right-side tool-tip radius com-	
		pensation (ZX plane)	
G80	0x2A00	Canned cycle cancellation	
G81	0x2A10	Drilling cycle	
G82	0x2A20	Counter-boring	
G83	0x2A30	Peck-drilling cycle	
G84	0x2A40	Tapping cycle	
G84_3	0x2AE0	Reverse rigid tapping	
G85	0x2A50	Boring cycle	
G86	0x2A60	Boring cycle	
G87	0x2A70	Back-boring cycle	
G88	0x2A80	Boring cycle	
G89	0x2A90	Boring cycle	
G73	0x2AA0	Peck-drilling cycle	
G74	0x2AB0	Reverse tapping cycle	
G76	0x2AC0	Fine boring	
G84_2	0x2AD0	Rigid tapping	
G84_3	0x2AE0	Reverse rigid tapping	
Gcyc16	0x2AF0	Canned cycle (reserved)	
Gcyc17	0x2B00	Canned cycle (reserved)	
Gcyc18	0x2B10	Canned cycle (reserved)	
Gcyc19	0x2B20	Canned cycle (reserved)	
Gcyc20	0x2B30	Canned cycle (reserved)	
G97	0x2C00	Constant surface speed control	
		cancellation	
G96	0x2C10	Constant surface speed control	
G54	0x2D00	Selection of workpiece coordinate	
		system 1	
G55	0x2D10	Selection of workpiece coordinate	
		system 2	
G56	0x2D20	Selection of workpiece coordinate	
		system 3	
G57	0x2D30	Selection of workpiece coordinate	
		system 4	

Symbol	Code value	Function	Remark
G58	0x2D40	Selection of workpiece coordinate	
		system 5	
G59	0x2D50	Selection of workpiece coordinate	
		system 6	
Gwcoord7	0x2D60	Selection of workpiece coordinate	
		system 7 (reserved)	
Gwcoord8	0x2D70	Selection of workpiece coordinate	
		system 8 (reserved)	
Gwcoord9	0x2D80	Selection of workpiece coordinate	
		system 9 (reserved)	
Gwcoord10	0x2D90	Selection of workpiece coordinate	
		system 10 (reserved)	
G_POL_STRT	0x2E00	Start of polar coordinate interpolation	
G_POL_OFF	0x2E10	End of polar coordinate interpolation	
G00_G01	0x2F00	Rapid traverse or linear interpolation	
G02_G03	0x2F10	CW or CCW circular interpolation	For CW
			arc,
			G02For
			CCW
			arc,G03
G03_G02	0x2F20	CCW or CW circular interpolation	For CW
			arc,
			G03For
			CCW
			arc,G02
G90_G91	0x2F30	Absolute or incremental command	For
			absolute
			command,
			G90 For
			incremental
			command,
			G91
G41_G42	0x2F40	Left- or right-side tool-tip radius R	For left-
		compensation	side com-
			pensation,
			G41
			For right-
			side com-
			pensation.
			G42

Symbol	Code value	Function	Remark
G42_G41	0x2F50	Right- or left-side tool-tip radius R	For left-side
		compensation	compen-
			sation,G42
			For right-side
			compen-
			sation, G41
G43_G44	0x2F60	Positive- or negative-side tool	For positive-
		length compensation	side com-
			pensation,
			G43
			For negative-
			side com-
			pensation,
			G44
G17_G19	0x2F70	XY, YZ, or ZX plane selection	For XY
			plane, G17
			For YZ
			plane, G18
			For ZX
			plane, G19
G20_G21	0x2F80	Metric or inch command	
G54_G59	0x2FC0	Workpiece coordinate system	Outputs G
		selection	code for a
			selected
			workpiece
			coordinate
			system.
G94_G95	0x2FD0	Feedrate selection	For feed per
			minute, G94
			For feed per
			rotation, G95
G98_G99	0x2FE0	Return position selection	
G96_G97	0x2FF0	Constant surface speed control	
		on/off	

27.1.4 NC data format setting function codes	(integer words)
---	-----------------

Symbol	Code value	Function	Remark
Omain	0x3000	Program number output	
Psub	0x3010	Output of subprogram call block P	
		code	
Nseq	0x3020	Sequence number output	
HcomIng	0x3040	Offset number output	
Pwcoord	0x3050	Workpiece coordinate system	
		value output	
Lcode	0x3060	Output of origin offset fixed L	
		value	
Osub	0x3070	Subprogram number output	
Frapid	0x3090	Rapid traverse F code	
Pthrd_type	0x30C4	Cutting by multiple repetitive	
		threading cycle	
Pthrd_floop	0x30D4	Repetition count for threading	
		cycle finishing	
Pthrd_chamf	0x30E4	Amount of cut by threading cycle	
		(chamfering)	
DH_FIX	0x0100	Output of a fixed value for the	
		word described above	
		The output value is set according	
		to the next int value.	

27.1.5 NC data format setting function codes (special codes)

Symbol	Code value	Function	Remark
Eob	0x4000	Output of NC data with EOB	Outputting only EOB is impossible.
EOB	0x4000	Output of NC data with EOB	Outputting only EOB is impossible.
EobOnly	0x4010	Output of NC data with EOB	Outputting only EOB is possible.
EOBonly	0x4010	Output of NC data with EOB	Outputting only EOB is possible.
OPT_SKIP	0x4020	/EOB output	
Eob_NON	0x4030	NC data output	
EOBnon	0x4030	NC data output	
Eor	0x4040	EOR (%) output	
EOR	0x4040	EOR (%) output	

27.1.6	NC data format setting function codes ((M codes)
--------	---	-----------

Symbol	Code value	Function	Remark
M00	0x5000	Programmed stop	
M01	0x5010	Optional stop	
M02	0x5020	Program end	
M98	0x5030	Subprogram call	
M99	0x5040	Subprogram return	
M06	0x5060	Tool exchange	
Mspndl_sel	0x511F	Spindle range selection	
M05	0x5120	Spindle stop	
M03	0x5130	Spindle normal rotation	
M04	0x5140	Spindle reverse rotation	
Mspndlhalt	0x5150	Spindle pause	
Mslock	0x5160	Spindle orientation stop	
M03_M04	0x5170	Spindle rotation	
M09	0x5300	Coolant off	
M07_M08	0x5310	Coolant on	
Mflood_on	0x5320	Coolant FLOOD	
Mmist_on	0x5330	Coolant MIST	
M_MILLAXES OFF	0x5600	Milling axis off	
M_MILLAXES ON	0x5601	Milling axis on	
M_YAXISOFF	0x5610	Y-axis off	
M_YAXISON	0x5611	Y-axis on	
M_CAXISOFF	0x5620	C-axis off	
M_CAXISON	0x5621	C-axis on	
M_ROTCLAM	0x5630	Rotation first axis clamp on	
PON1			
M_ROTCLAM POFF1	0x5640	Rotation first axis clamp off	

27.1.7 NC data format setting function codes (S codes)

Symbol	Code value	Function	Remark
Srot	0x6000	Spindle rotation command output	
Ssurf	0x6010	Output of S code for surface speed command	
Smaxrpm	0x6020	Output of S code for maximum spindle clamp value	
Spndl_rng	0x6030	Spindle rotation range output	
Srotsur	0x6040	Spindle rotation speed/surface speed command S code	

27.1.8 NC data format setting function codes (T codes)

Symbol	Code value	Function	Remark
TOOL_OFST_	0x7000	Tool offset cancellation	
CAN			
Tsel	0x7010	Output of T code for selection of a	
		tool on tool post 1	
Tfin	0x7040	Output of final T code for tool post	
		1	

27.1.9 NC data format setting function codes (outputting arbitrary character strings)

Symbol	Code value	Function	Remark
Str1	0x8000	Output of arbitrary character	
		string No. 1	
:	:		
Str16	0x800F	Output of arbitrary character	
		string No. 16	
Name	0x8010	Part program name output	
ProcName	0x8020	Machining type name output	

27.1.10	NC data format setting function codes (modal clearing)
---------	--

Symbol	Code value	Function	Remark
Gclr	0x9010	Clearing G-code modal infor-	Following
		mation	this call,
			specify the
			number of
			the group to
			be cleared.
			0xFFFF
			means all
			groups.
McIr	0x9020	Clearing M-code modal	Following
		information	this call,
			specify the
			number of
			the group to
			be cleared.
			0xFFFF
			means all
			groups.
Tclr	0x9030	Clearing T-code modal	
		information	
Sclr	0x9040	Clearing S-code modal	
		information	
Fclr	0x9050	Clears F code modal information	

27.2 Pre/Postprocessor Macro Syntax

27.2.1 General

(1) Outline

The pre/postprocessor macro is used for customizing a postprocessor. The pre/postprocessor macro has the following features:

- Text (ASCII) format
- Syntax similar to C language
- Blanks/tabs can be used between words, and multi-statements are supported.
- Preprocessor pseudo instructions such as #include and #define are supported.
- The concept of variables is employed; both local variables and global variables can be used.
- Arithmetic operations, compare operations, and logical operations are supported.
- The control structure supports condition decision and loop execution.
- Procedures can be defined and subsequently called.

(2) Notation

- [*a*] : Entry *a* can be omitted.
- a|b: Either of entry *a* or *b* must be selected.
- ... : The preceding entry is repeated.

27.2.2 Components

- (1) Character set
 - Uppercase letters (A to Z), lowercase letters (a to z), numeric characters (0 to 9), underscore (_).

Variable names can be created using these characters. The reserved words are also created using these characters. Note that pre/postprocessor macros distinguish between uppercase letters and lowercase letters.

• Blank characters (space, tab, line feed, form feed, comment) These characters are used to delimit constants, variable names, and reserved words from each other in pre/postprocessor macros. All blank characters other than those that do not appear in a character string constant are ignored, such that blank characters can be used to improve the readability of the pre/postprocessor macro. Special characters (, . ; " () [] { } + - * / %
 ! & | = < > #)

When used in a pre/postprocessor macro, each of these special characters has a special function. The use of a special character is described whenever a special character appears in this manual. Special characters other than the above are used only in character string constants, comments, and #include file names.

• Escape sequence (\" \\)

The back slash symbol (\) is used to include a double quotation mark (") or back slash symbol (\) in a character string constant. If a character other than these characters is placed after a back slash symbol, the back slash symbol is ignored and only the character itself becomes effective. When \c is specified, for example, c is assumed.

• Non-display characters

Non-display characters other than tabs, line feeds, and form feeds cannot be used.

• Operators

Operators are special characters used to connect operands in an expression and specify a type of operation. For details, see Section 27.2.5.

(2) Constants

- Integer constant (decimal number, octal number, hexadecimal number)
 - digits
 - 0*odigits*

0xhdigits / 0Xhdigits

Real constant

[digits][.digits][E |e |D |d [+ |-]digits]

• Character string constant

"characters" ["characters"][...]

(3) Variable names and procedure names

Variable names and procedure names consist of one or more letters, numeric characters, and/or underscores, starting with a letter.

No limit is imposed on the length of a name. If a name of more than 31 characters is specified, however, only the first 31 characters are effective as the variable name or procedure name.

The system distinguishes between uppercase and lowercase letters. Therefore, those names that are spelled in the same way but which differ in their capitalization are treated as different variable names or procedure names.

Variable names and procedure names must not duplicate any reserved word.

(4) **Reserved words**

void	int	double	string
function	if	else	for
break	continue	e	

(5) Comments

A comment consists of character strings enclosed within a /*-*/ pair. A comment may consist of more than one line. Nesting is not allowed, however. A comment can be written in any location where blank characters are allowed.

27.2.3 Structure

(1) Basic structure

A pre/postprocessor macro consists of one or more preprocessor pseudo instructions, declare statements, assignment statements, control statements, procedure statements, EL statements, and comment statements. Each of these is summarized below.

• Preprocessor pseudo instruction

Before a pre/postprocessor macro is analyzed, preprocessor pseudo instructions perform processing such as header processing, macro expansion, and condition-based analysis skip processing. Basically, the preprocessor pseudo instructions have functions similar to those of the C preprocessor.

• Declare statement

A declare statement declares a variable by specifying a type name and variable name. When a variable is declared, that variable can simultaneously be initialized to an arbitrary value. The concepts of lifetime and visibility apply to a declared variable.

• Assignment statement

An assignment statement assigns a value to a variable declared with a declare statement. An assignment statement is an assignment expression described as an independent statement. The format of an assignment statement corresponds to the assignment expression description.

• Control statement

A control statement either specifies a conditional expression to select another statement, or controls a loop. A control statement consists of a control part and main part. The control part consists of one statement or multiple statements. Multiple statements are enclosed in braces {}. Statements enclosed in braces are referred to as a control block.

• Procedure definition statement

A procedure definition statement defines a procedure by specifying a type name and procedure name. A procedure definition statement consists of a formal argument part and main -395-

part. In the main part, any statements that can be specified in a pre/postprocessor macro other than procedure definition statements can be described.

• Procedure call statement

A procedure call statement executes a procedure defined with a procedure definition statement.

• EL statement

An EL statement specifies the machining information to be passed to the machining execution function. The format of an EL statement is similar to that for calling a function. Many different EL statements are supported, to be used with various machining information items.

• Comment statement

A comment statement consists only of character strings, and has no effect on the execution. A comment statement, enclosed in a pair of /* and */, can be described at any location where blank characters are allowed.

(2) File configuration

A pre/postprocessor macro can be divided into multiple files. In this case, the pre/postprocessor macro must appear as a single large file having #include pseudo instructions.

(3) Block

In the pre/postprocessor macro, those portions braces {} are called blocks.

Three types of blocks are used. The first is a function block that constitutes the main part of a function statement. The second is a control block that is used when the main part of a control statement includes multiple statements. The third is a procedure block that constitutes the main part of a procedure statement.

Particularly, a function block declares a variable, defining the lifetime and visibility of that variable.

One block can contain multiple blocks. This embedding is referred to as nesting.

Note, however, that the pair of braces used for initialization in an array declaration differs from those used to specify a block.

(4) Lifetime

Any variable declared outside all function blocks has a global lifetime. A variable declared in a function block has a local lifetime within the local block and the block(s) nested in the local block.

(5) Visibility

A variable declared outside all function blocks is visible in the range from the declaration position to the end of the pre/postprocessor macro.

A variable declared in a function block is visible in the range from the declaration position to the end of the local block. A variable declared in an external function block is visible in all blocks nested in the function block. If a nested function block defines the same variable name as that declared in an external function block, the variable declared in the external function block will not be visible in the nested function block. This state is cleared, however, when the internal function block ends.

27.2.4 Declare statement

(1) Type name

A pre/postprocessor macro can handle three types of data:

- int: Integer type
- double: Real type
- string: Character string type

(2) Variable declaration

Before a variable can be used, it must be declared. A variable can be declared by describing a data type followed by a variable name; an additional variable of the same data type can subsequently be declared by specifying a contiguous comma (,) followed by the additional variable name.

A variable must be declared at the beginning of a function block. After a statement other than a declare statement appears in a function block, no further variables can be declared.

Within a single block, any given variable name cannot be declared more than once.

There are two types of variable declarations, simple variable declarations and array declarations:

• Type-name variable-name

Simple variable declaration

• *Type-name variable-name (numeric-expression)*[(*numeric-expression*)...]

Array declaration

By following a *variable-name* with one or more *numeric-expressions* enclosed in parentheses(), an array declaration can specify the number of array elements; all elements are of the type specified by *type-name*.

Let n be the number of *numeric-expressions* that follow *variable-name*. Then, the array is defined as an n-dimensional array; one *numeric-expression* defines the number of elements of one dimension.

Note, however, that no more than ten *numeric-expressions* can be specified. This means that an array having up to ten dimensions can be specified.

From the viewpoint of syntax, the format of a *numeric-expression* must, as a bare minimum, satisfy the *numeric-expression* format -397-

described in Section 27.2.5. Note, however, that a *numericexpression* is used to define the number of array elements, so that one integer constant or an integer expression should be specified.

(3) Initialization

When a variable is declared, the variable can be initialized by specifying an initial value.

- Initialization of a simple variable *Type-name variable-name = expression*
- Initialization of an array

Type-name variable-name (numeric-expression) [(*numeric-expression*)...] = {*expression*,...}

A simple variable declaration can be initialized by describing, after *variable-name*, an equal sign (=) followed by an *expression*. A variable of integer or real type, if not initialized explicitly, assumes an initial value of 0. A variable of character string type, if not initialized explicitly, assumes an initial value of " " (null string).

An array variable declaration can be initialized by describing an equal sign (=) followed by {*expression*,...} after *variable-name* (*numeric-expression*)[(*numeric-expression*),...].

Those *expressions* enclosed in braces {} are used for initialization. At least one *expression* must be specified, but the number of *expressions* must not exceed the number of array elements. Specified values are assigned to the corresponding array elements.

If the number of *expressions* for initialization is less than the total number of array elements, the remaining array elements of integer or real type automatically assume an initial value of 0, while the remaining array elements of character string type assume an initial value of " " (null string). All array elements that are not explicitly initialized assume initial values in the same way.

An error occurs if the number of *expressions* exceeds the total number of array elements.

For details of the specification format of *expressions*, see Section 27.2.5.

27.2.5 Expression and assignment

(1) Expression

Expressions are classified into two types: *numeric-expressions* that are used for obtaining integers or real numbers, and *character-string-expressions* that are used for obtaining character string values.

(a) Numeric expression

A *numeric-expression* consists of one or several operands connected by an operator. Each basic operand is a constant or variable.

Any operand can be enclosed in parentheses (). In this case, the -398-

operand enclosed in parentheses is assumed to be grouped explicitly, and functions as an expression. The format of a constant or variable depends on whether it is an integer or real number, as follows:

- Integer constant: [*Sign*][0][x|X]*integer-part*
- Real constant: [Sign][integer-part][.fraction-part] [e|E|d|D[sign]exponent-part]
- Integer variable: *Variable-name* of int type
- Real variable: Variable-name of double type

The operators used in an expression follow a priority order and associativity as evaluation criteria.

The table below lists the operands, together with their priorities and associativities.

Operator	Type of operator		Associativity	Priority
() []	Expression		\rightarrow	Higher
! –	Unary operator		\leftarrow	
* / %	Multiplication, division			♠
+ -	Addition, subtraction			
< <= > >=	Comparison	Binary operator	\rightarrow	
== !=	Equivalence			↓
&&	Logical AND			•
	Logical OR			
=	Assignment operator		\leftarrow	Lower

(b) Character string expression

A *character-string-expression* represents a character string value with one character string constant or character string variable.

No character string expression can be enclosed in parentheses. The formats of character string constants and character string variables are as follows:

• Character string constant:

"Characters" ["Characters"] [...]

• Character string variable:

Variable of string type

(2) Assignment

An assignment operation loads the evaluation value of the expression on the right side of the assignment operator (=) to the previously declared variable on the left side. Accordingly, the type of the variable on the left side of an assignment operator is related to the expression on the right side, as follows:

- Variable-name of int type = numeric-expression
- *Variable-name* of double type = *numeric-expression*
- Variable-name of string type = character-string-expression

An assignment expression described as an independent statement is referred to as an assignment statement.

The right side of an assignment statement contains an expression, which may include an assignment operation. So, an assignment statement can be written as follows:

Variable-name[= variable-name] = expression;

(3) Referencing variables

To use a variable as a basic operand, the variable must be declared beforehand and must also be visible (Section 27.2.3.(5)). The value of a variable is changed by assignment. The current value of a variable is obtained by referencing it.

When a variable to be referenced is an array variable, the elements of the array to be referenced must be specified using *subscript-expressions* enclosed in brackets [], after the *variable-name*.

 Array-variable-name[subscript-expression][...[subscriptexpression]]

The number of times [*subscript-expressions*] must match the number of array dimensions.

From the viewpoint of syntax, a *subscript-expression* enclosed in brackets [] must be a *numeric-expression* and, as a bare minimum, satisfy the format of *numeric-expressions* described in (1) above.

27.2.6 Control statement

(1) If and else statements

The if and else statements are both conditional branch statements, and have the following format:

if (expression) statement-a; [else statement-b;]

Here, if (*expression*) is referred to as the control part, else is referred to as the auxiliary part, and *statement-a* and *statement-b* are referred to as the main parts. The expression in if (*expression*) is a numeric expression.

If the result of evaluating the control part is true ($\neq 0$), *statement-a* is

-400-

executed. If the result of evaluating the control part is false (= 0), *statement-b* is executed. If, however, the result of evaluating the control part is false (= 0), and else is omitted, no statement is executed and *statement-a* is ignored.

By enclosing the main part of an if statement (and else statement) in braces {}, multiple statements can be contained in the main part, as described below. Such a main part is referred to as a control block.

if(expression) {statement-1;...statement-m;}
 [else {statement-1;...statement-n;}]

(2) For statement

The for statement is used to apply loop control. Its format is as follows:

• for ([multi-expression-1]; [multi-expression-2]; [multi-expression-3]) statement;

The part, for ([multi-expression-1]; [multi-expression-2]; [multi-expression-3]), is referred to as the control part, and statement is referred to as the main part.

Here, a *multi-expression* represents one *numeric-expression*, or *multiple-numeric-expressions* delimited from each other by a comma. When a *multi-expression* is evaluated, the result of evaluating the last *numeric-expression* is returned.

Each *multi-expression* in the control part performs the evaluation described below.

- *Multi-expression-1*: This expression is a loop initialization expression which is executed once before entering a loop. If this expression is omitted, no initialization is performed.
- *Multi-expression-2*: This expression is a loop condition expression which performs evaluation before the execution of one loop. If this expression is omitted, the true condition is always assumed. If the result of evaluation is true (\neq 0), the *statement* is executed, and the loop is repeated. If the result of evaluation is false (= 0), the loop is terminated.
- *Multi-expression-3*: This expression is a loop adjustment expression which is executed after the execution of one loop. If this expression is omitted, no adjustment is performed.

By enclosing the main part of a for statement in braces {}, *multiple-statements* can be specified in the main part, as described below.

for ([multi-expression-1]; [multi-expression-2]; [multi-expression-3]) { statement-1; ...statement-n; }

(3) Break statement

The break statement is an auxiliary statement for loop control which can be described only within the main part. The appearance of this statement causes the loop to stop immediately.

break;

(4) Continue statement

The continue statement is an auxiliary statement for loop control which can be described only within the main part. The appearance of this statement causes the remaining statement or statements in the main part to be ignored, and control is passed to the next iteration.

continue;

27.2.7 Procedure statement

(1) Concept of procedure

Several statements can be combined into a procedure, such that those statements can be executed simply by calling the procedure.

(2) **Procedure type**

Only one procedure type, VOID, can be used for a procedure definition.

(3) Definition format

Procedure-type procedure-name([type-name-expression][, typename expression][...])

{

[declare-statement][...] [assignment-statement | control-statement | EL-statement | function-statement | procedure-call][...]

- }
- [
- The procedure statements in the main part of a procedure definition is not executed during definition. Instead the main part is executed when the procedure is called.
- The *procedure-type* must not be omitted.

(4) Calling a procedure

Procedure-name([expression][, expression][...]);

- An actual argument must be specified as immediate data, an expression, or variable.
- Only input is allowed for immediate data or an expression. Both input and output are allowed for a variable.
- (5) Return value
 - No value is returned. (An argument specified when calling a procedure is used to return a value.)

(6) **Restrictions**

- A procedure statement definition must be described outside all blocks.
- Procedure names that duplicate reserved words (int, double, string, function, and so forth) cannot be used.
- When the name of an EL statement duplicates that of a procedure statement, the procedure statement has the higher priority.
- A procedure name not exceeding 31 characters must be specified. (All characters subsequent to the first 31 characters are ignored.)
- If a procedure name is defined more than once, the most recently made definition replaces all other definition(s).
- A procedure cannot be defined while another procedure is being defined or called.
- While a procedure is being defined, another procedure cannot be called.
- The calling of procedures can be nested up to 50 levels.
- (7) Visibility
 - (a) Visibility of a procedure statement
 - If a procedure definition statement exists in the same file, the procedure statement will be visible from the point at which it is defined until the end of execution.
 - If a procedure definition statement exists in another file, all procedures defined in the file(s) executed before the current file are visible. The procedures in a library that are made visible using the reference option become visible from the start to the end of execution.

(b) Visibility of a variable

- A variable declared in a procedure block is visible within the block, and its lifetime does not extend beyond the block at the time of execution.
- A variable declared outside all blocks can be referenced using a procedure. The global variables in files other than that containing the procedure definition are not visible.

(8) Arguments

- The type (numeric value or character string) of arguments, the number of arguments, dimensions, and array size are checked.
- An error occurs if the number of arguments in an argument list is invalid.

Actual argument	int type	double type	string type
int type	0	double type ↓ int type Cast from int type to double type	Error
double type	int type ↓ double type Cast from double type to int type	0	Error
string type	Error	Error	0

An argument type check is made as indicated in the table below.

(9) Sample

#include "mdfile.h" /* procedure definition */ void proc(int i_1, int i_2, double d_1, double d_2, string s_1[3]) { draw_pen(i_1); draw_pen(i_2); $mot_goabs(d_1, d_2,);$ name_mac($s_1[0]$); i_1 = 3; i_2 = 5; d_1 = 20.0; d 2 = 30.0; s_1[1] = "str_1"; } int wint; double wdbl = 50.0;

-404-

27.2.8 EL statement

The EL statement specifies the machining information (referred to as EL data) to be passed to the machining execution function. The basic format of an EL statement is as follows:

• *EL-statement-name([expression][, expression][...]);*

A range of *EL-statement-names* are supported to match many different machining information items. For each *EL-statement-name*, the type (numeric value or character string) of the expression(s) in parentheses, as well as the number of expressions, are determined. (Note, however, that for some EL statements such as the EL statement for the post function, the type of expression(s) in parentheses and the number of expressions are not fixed.)

This means that the EL statement has a format similar to that of calling functions in C; an *EL-statement-name* is equivalent to a function name, and actual arguments are described within parentheses.

27.2.9 Preprocessor pseudo instruction

The instructions below can be described to perform particular operations before a pre/postprocessor macro is analyzed; the instructions are basically similar those of the C preprocessor.

```
#include <pathname>
#include 'pathname'
#define name sequence_of_tokens
#define name(arg[,...]) sequence_of_tokens
#undef name
#if
```

#ifdef	(Not supported.	Use #if	<pre>defined(xxx).)</pre>
#ifndef	(Not supported.	Use #if	<pre>!defined(xxx).)</pre>
#else			
#elif			
#endif			
defined			

For details, refer to the relevant C language manual.

27.2.10 Restrictions

- Length of a line
 Each line can contain up to 255 characters, including \n at the line end.
- Length of a statement The length of each statement is not limited.
- Length of a variable

The length of each variable is not limited. If a variable name of more than 31 characters is specified, however, the first 31 characters function as a variable name.

- Number of variables The number of variables is not limited.
- Function statement nesting depth Function statements can be nested up to 20 levels.
- Integer/real number value range The same ranges as represented by int and double of C language apply.
- Length of a character string A character string can be up to 255 characters in length.

<A>

Abbreviations of functions 340 About Setup 3 About the protector 8 Activating Dimension Measurement 310 Activating only the File Transmission Tool 250 Activating the File Transmission Manager 250 Activating the File Transmission Tool 249 Activation using the Symbolic CAP T output function 249 Adding a material name 166 Adding cutting condition data 170 Adding movement entities 132 Adding or deleting a tool material name only 204 Adding or inserting the tool data for a tool into the tooling data 199 Adding setting items 246 Addition of Tooling Data 196 Address setting 221 Adjusting the creation method of a fitting curve 156 Angles formed by two lines 316 Animation Screen 122 APPENDIX 382 Application software 247 Arbitrary character string 223 Auxiliary menu 31

<*B*>

BASIC OPERATION 9 Basic Operation 16 Before Trouble Occurs 337 Blank Figure Color/Line Type Modification 58 BLANK FIGURE CREATION 54 Blank Figure Creation 54 Buttons 252

$\langle C \rangle$

CAD data 277 CAD DATA INPUT/OUTPUT 277 CAD Data Input/Output 277 Calculating method of the machining conditions 176 Canceling movement entry editing 136 Cautions 318 Cautions and Restrictions 307 Changing and modifying the machining process 109 Changing Cutting Condition Files 189 Changing icons 335 Changing the contents of the transmission operation 254 Changing the order of the tool types 175 Changing the order of the tools in the tool data 203 Changing the schedule 269 Changing Tooling Files 212 Checking the transmission status 255 Checking tool paths for continuity 135 Chuck figure registration/edit screen 91 Circle\Arc Radius 315 Clear 258 **CNC Specifications 221** Components 393 Condition setting 105 Confirming the version 16 Contents of Symbolic CAP T window 2-2 Contents of the screen 268 Control statement 400 Conversion specifications for DXF file input/output 296 Coordinate system 147 Coordinate system used for tool path editing 128 Copying figures 149 Creating icons 334 Creating polygons 155 **CREATING PROGRAMS 140** Creation method of a fitting curve 156 Customization 242 CUTTING CONDITIONS 162 CUTTING CONDITIONS DATA UTILITY 179

$\langle D \rangle$

Data format that can be transmitted 248 Data setting screen for animated simulation 124 Declare statement 397 Defining new machining process 111 Delete 257 Deleting a material name [Edit Cutting Condition Data] 168 Deleting a Tooling 198 Deleting buttons/spaces 335 Deleting cutting condition data 175 Deleting cutting condition data on a machining type basis 175 Deleting setting items 245 Deleting tool data from the tooling data 202

Index

Deleting tool from the tool data 203 Deleting tool paths 131 Devices to which data can be transmitted 247 **DIMENSION MEASUREMENT 308 Dimension Measurement Function 308** Discarding the settings/returning to the normal state 336 Displaying input information 306 Displaying Measurements 317 Displaying the progress of input processing 305 Displaying the Tool Bar Setting screen 334 Distance between Points and Entity 314 Distance between two points 313 Drawing Format Setting 40 Drawing window 13 Duplicating the tool in the tool data 203 DXF file input 282 DXF File Input/Output 279 DXF file output 289 DXF input conversion specifications 296 DXF output conversion specifications 298

$\langle E \rangle$

Edit menu 28 Editing 128 Editing cutting condition data 169 Editing movement entities 132 EL statement 405 Ending the transmission operation 256 Entity selection 61 Erasing movement entities 133 Error Messages 339 Example of NC Data Output 239 Example of operation 43 Example of referencing NC data formats 225 Example Transmission Procedures 271 Execution of Tool Path Editing 129 Exiting the File Transmission Manager 270 Explanation of each step 42 Expression and assignment 398

$\langle F \rangle$

FAPT LANGUAGE FIGURE INPUT 301

FAPT language figure input 303 FAPT statement types that can be input 301 Feedrate 220 FIGURE CREATION 141 Figure creation layer setting 39 Figure creation mode 19 Figure types that can be input 301 File configuration 242 File menu 26 File operation functions 248 File specification areas 252 FILE TRANSFER 247 File Transmission Tool Screen 251 Fitting curve 156 Flow of Operation 35, 41 Fully automatic determination 112 Function Codes 382 Function Outline 247 Functions 179, 194, 205 Functions of deleting 164 Functions Provided by Menu Bar 26 Functions Provided by Side Menu 19 Functions Provided by the Tool Bar 34 Fundamentals 9

$\langle G \rangle$

G code 222 General 393

$\langle H \rangle$

Handling CAD data efficiently 295 Handling of CAD data by this system 278 Help menu 33 Home position/index position 219

<*I*>

If Trouble Occurs 338 Inputting Cutting Condition Files 184 Inputting Tool/Tooling Files 210 Inserting spaces 335 Inward offset for a corner 160

< L >

i-2

Index

Layer 38 Listing entities 136 Loading a File 139 Location of the division point and the cutting area 101

< M >

M code 222 Machine Specifications 219 MACHINING DEFINITION 93 Machining Definition Screen 94 Machining plane coordinate system 147 Machining process list 130 Major classification of layers 38 Measurable Dimensional Values 309 Measuring Coordinates 311 Menus 251 Menu bar 13 Messages concerning machining surface setting 347 Messages concerning the milling library 363 Messages related to blank and parts figure preparation341 Messages related to cutting condition setting 369 Messages related to loading part files 366 Messages related to machining definition 347 Messages related to NC data preparation 351 Messages related to pre-machining settings 345 Messages related to tool and tooling data 375 Modify menu 27 Modifying cutting condition data 175 Modifying figures by extension/trimming 148 Modifying initial values 243 Modification of the Tooling Data 199 Modifying setting item names 244 Modifying the cutting condition data on a machining type basis 169 Modifying the NC Data Output Format 216 Modifying the operation flow 17 Modifying tool registered in the tooling data 202

< N >

NC Data Format 225 NC data format setting function codes (G codes) 385 NC data format setting function codes (integer words) 389 NC data format setting function codes (M codes) 390 NC data format setting function codes (modal clearing) 392 NC data format setting function codes (move command words) 382 NC data format setting function codes (outputting arbitrary character strings) 391 NC data format setting function codes (S codes) 391 NC data format setting function codes (special codes) 389 NC data format setting function codes (special functions) 382 NC data format setting function codes (T codes) 391 NC DATA PREPARATION 115 NC data preparation data setting screen 121 NC Data Preparation Screen 116 NC Machine File 214 NC machine name 219 NC MACHINE SETTING 214 NC Machine Setting Tool Operation 241

$\langle O \rangle$

Offset 159 Operating Procedure 253 Operation 302 Operation guide 252 Other messages 379 Other Symbolic CAP T icons 9 Other usable devices 2 Outline of Processing for Each Function 257 **Outputting Cutting Condition Files 182** Outputting Tool/Tooling Files 208 Outward offset for a corner 159 Overview 162, 192, 301 Overview of Data 36 Overview of DXF 279 Overview of each function 35 Overview of NC Data Preparation Procedure 215

< P >

PARAMETER SETTING 321
Parameter Setting for Auto Saving 333
Parameter Setting for Blank Figure and Part Figure 324
Parameter Setting for Figure Operation 322
Parameter Setting for Machining Condition Automatic Decision 326
Parameter Setting for Machining Definition 325
Parameter Setting for NC Data Preparation 332
Parameter Setting for Overall Setting of Symbolic CAP T 324
Parameter Setting for Pre-Machining Setting 325
Parameter Setting for Tool Path Creation 331
Parameter setting for view 321

Index

Part program selection 302 PARTS FIGURE CREATION 59 PLOTTING 272 Pre/Postprocessor Macro Syntax 393 PRE-MACHINING SETTINGS 84 Preparing and Displaying NC Data 115 Preprocessor pseudo instruction 405 Prerequisites 1 Printing NC data and a process table 126 Prior to Setting Up (Operating Environment) 1 Procedure 41 Procedure statement 402 Procedures 274

Process Order Editing Screen 119 PROGRAMMING EXAMPLES 41 Programming mode 19 Prompt area 14

$\langle R \rangle$

Reading the latest information about Symbolic CAP T 9 Read-only and read/write 258 Receive 257 Refresh 259 Registering a new tool and adding or inserting the data into the tooling data 200 Registering a new tool and adding or inserting tool data 202 Registering menu items 335 Reinstalling Symbolic CAP T 8 Rename 258 Restrictions 406 <\$\$>

Saving a File 138 SAVING AND LOADING FILES 138 Saving settings/returning to the normal state 336 Selecting a command 254 Selecting an NC Machine File 86 Selecting a tool path for editing 130 Selecting entities 145 Selecting the device and setting the transmission conditions 253 Selecting the Machining Type 95 Selecting Tooling Data 85 Selecting Utility Functions 181, 207 Send 257 Setting items of the chuck 89 Setting menu 32 Setting procedure 260 Setting reception control conditions 266 Settings related to the FAPT language figure input function 304 Setting the Chuck 89 Setting the communication conditions 264 Setting the home position 87 Setting the index position 88 Setting the input conversion method 284 Setting the output conversion method 291 Setting the transmission control conditions 265 Setting Transmission Conditions 260 Setting up a Plotter 272 Setting up an optimal module 6 Setting Up Symbolic CAP T 3 SETUP 1 Side menu 12 Special figure 56 Specifying a radius, distance, and angle 142 Specifying Another Area 107 Specifying files 253 Specifying positions 143 Specifying required functions 223 Specifying start entities 134 Specifying the Cutting Area 97 Specifying the Cutting Area and Condition Data 97 Specifying the device and plotter 263 Spindle speed 220 STANDARD SETTING 319 Standard setting files 263 Standard Setting for Blank Figure and Parts Figure 319 Standard Setting for NC Data Preparation 320 Start-up 302 Starting of Cutting Condition Data 165 Starting of tool data/tooling data editing 195 Starting Symbolic CAP T 9 Starting the Cutting Condition Data Utility 180 Starting the Tool/Tooling Data Utility 206 Starting transmission 255 Status window 15 Storing the tool bar definition file 334 Structure 395 Symbolic figure creation by figure reference 80 Symbolic Figure data editing 78 Symbolic figure data input 62

Symbolic Input Method 61

< T >

Terminating movement entry editing 137

Terminating Symbolic CAP T 9

Terminating tool path editing 131

Timing of NC data format reference 226

Title bar 11

Tool bar 13

TOOL BAR SETTING 334

Tool Bar Setting 334

TOOL DATA AND TOOLING DATA 192

TOOL PATH EDITING 127

Tool Path Editing 127

Tool Path Editing Dialog Box 130

Tool Paths subjecting to editing 127

TOOL/TOOLING DATA UTILITY 205

Total number of axes/number of simultaneously controlled axes 219 Transmission protocols 248 Transmission Schedule Management 268

Transmission to and from the CNC 271

Trimming/Extending movement entities 133

TROUBLESHOOTING 337

Turning Figure Creation 59

Types of Figures 141

 $\langle U \rangle$

Undoing/redoing editing 133

Unit of editing 128

Using the Help function to become familiar with the operating procedure 16

$\langle V \rangle$

View menu 30

 $<\!W\!>$

Whether to use the ATC function 220

World coordinate system 147

Writing NC data to a floppy disk and transferring NC data to a peripheral device 125

σ
5
0
C
Φ
2
_
5
0
10
S
/isi
visi
evisi
Revisi

FANUC Symbolic CAP T Basic Module V1 OPERATOR'S MANUAL (B-62824EN)

				Contents
				Date
				Edition
		 Tool Path Editing function is added. Cutting Condition Data Utility function is added. FAPT Language Figure Input function is added. Tool/Tooling Data Utility function is added. 		Contents
		Nov.,'98	Jul.,'96	Date
		02	0	Edition
- No part of this manual may be reproduced in any form.
- All specifications and designs are subject to change without notice.