



GE Fanuc Automation

Computer Numerical Control Products

Series 15i / 150i – Model A

Operator's Manual (Operation)

B-63324EN-1/01

February 1999

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.

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units.

It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.

Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

CONTENTS

1.	DEFINITION OF WARNING, CAUTION, AND NOTE	s-2
2.	GENERAL WARNINGS AND CAUTIONS	s-3
3.	WARNINGS AND CAUTIONS RELATED TO PROGRAMMING	s-5
4.	WARNINGS AND CAUTIONS RELATED TO HANDLING	s-8
5.	WARNINGS RELATED TO DAILY MAINTENANCE	s-11

DEFINITION OF WARNING, CAUTION, AND NOTE

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

WARNING

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

CAUTION

Applied when there is a danger of the equipment being damaged, if the approved procedure is not observed.

NOTE

The Note is used to indicate supplementary information other than Warning and Caution.

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

GENERAL WARNINGS AND CAUTIONS

WARNING

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
2. Before operating the machine, thoroughly check the entered data. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user..
4. When using a tool compensation function, thoroughly check the direction and amount of compensation. Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
5. The parameters for the CNC and PMC are factory-set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change. Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.
6. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit. Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.
7. The operator's manual and programming manual supplied with a CNC unit provide an overall description of the machine's functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

WARNING

8. Some functions may have been implemented at the request of the machine-tool builder. When using such functions, refer to the manual supplied by the machine-tool builder for details of their use and any related cautions.

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off.

Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.

To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

This section covers the major safety precautions related to programming. Before attempting to perform programming, read the supplied operator's manual and programming manual carefully such that you are fully familiar with their contents.

WARNING

1. Coordinate system setting

If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command.

Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. Positioning by nonlinear interpolation

When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming.

Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Function involving a rotation axis

When programming polar coordinate interpolation or normal-direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely.

Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4. Inch/metric conversion

Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position.

Before starting the machine, therefore, determine which measurement units are being used.

Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

WARNING**5.Constant surface speed control**

When an axis subject to constant surface speed control approaches the origin of the workpiece coordinate system, the spindle speed may become excessively high. Therefore, it is necessary to specify a maximum allowable speed. Specifying the maximum allowable speed incorrectly may damage the tool, the machine itself, the workpiece, or cause injury to the user.

6.Stroke check

After switching on the power, perform a manual reference position return as required. Stroke check is not possible before manual reference position return is performed. Note that when stroke check is disabled, an alarm is not issued even if a stroke limit is exceeded, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

7. Tool post interference check

A tool post interference check is performed based on the tool data specified during automatic operation. If the tool specification does not match the tool actually being used, the interference check cannot be made correctly, possibly damaging the tool or the machine itself, or causing injury to the user.

After switching on the power, or after selecting a tool post manually, always start automatic operation and specify the tool number of the tool to be used.

8. Absolute/incremental mode

If a program created with absolute values is run in incremental mode, or vice versa, the machine may behave unexpectedly.

9.Plane selection

If an incorrect plane is specified for circular interpolation, helical interpolation, or a canned cycle, the machine may behave unexpectedly. Refer to the descriptions of the respective functions for details.

10.Torque limit skip

Before attempting a torque limit skip, apply the torque limit. If a torque limit skip is specified without the torque limit actually being applied, a move command will be executed without performing a skip.

11.Programmable mirror image

Note that programmed operations vary considerably when a programmable mirror image is enabled.

WARNING

12.Compensation function

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.

Before issuing any of the above commands, therefore, always cancel compensation function mode.

WARNINGS AND CAUTIONS RELATED TO HANDLING

This section presents safety precautions related to the handling of machine tools. Before attempting to operate your machine, read the supplied operator's manual and programming manual carefully, such that you are fully familiar with their contents.

WARNING

1.Manual operation

When operating the machine manually, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and feedrate have been specified correctly. Incorrect operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

2.Manual reference position return

After switching on the power, perform manual reference position return as required. If the machine is operated without first performing manual reference position return, it may behave unexpectedly. Stroke check is not possible before manual reference position return is performed. An unexpected operation of the machine may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3.Manual numeric command

When issuing a manual numeric command, determine the current position of the tool and workpiece, and ensure that the movement axis, direction, and command have been specified correctly, and that the entered values are valid. Attempting to operate the machine with an invalid command specified may damage the tool, the machine itself, the workpiece, or cause injury to the operator.

4.Manual handle feed

In manual handle feed, rotating the handle with a large scale factor, such as 100, applied causes the tool and table to move rapidly. Careless handling may damage the tool and/or machine, or cause injury to the user.

WARNING**5.Disabled override**

If override is disabled (according to the specification in a macro variable) during threading, rigid tapping, or other tapping, the speed cannot be predicted, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

6.Origin/preset operation

Basically, never attempt an origin/preset operation when the machine is operating under the control of a program. Otherwise, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the tool, or causing injury to the user.

7.Workpiece coordinate system shift

Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.

If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

8.Software operator's panel and menu switches

Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.

Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

9.Manual intervention

If manual intervention is performed during programmed operation of the machine, the tool path may vary when the machine is restarted. Before restarting the machine after manual intervention, therefore, confirm the settings of the manual absolute switches, parameters, and absolute/incremental command mode.

10.Feed hold, override, and single block

The feed hold, feedrate override, and single block functions can be disabled using custom macro system variable #3004. Be careful when operating the machine in this case.

11.Dry run

Usually, a dry run is used to confirm the operation of the machine. During a dry run, the machine operates at dry run speed, which differs from the corresponding programmed feedrate. Note that the dry run speed may sometimes be higher than the programmed feed rate.

WARNING**12.Cutter and tool nose radius compensation in MDI mode**

Pay careful attention to a tool path specified by a command in MDI mode, because cutter or tool nose radius compensation is not applied. When a command is entered from the MDI to interrupt in automatic operation in cutter or tool nose radius compensation mode, pay particular attention to the tool path when automatic operation is subsequently resumed. Refer to the descriptions of the corresponding functions for details.

13.Program editing

If the machine is stopped, after which the machining program is edited (modification, insertion, or deletion), the machine may behave unexpectedly if machining is resumed under the control of that program. Basically, do not modify, insert, or delete commands from a machining program while it is in use.

WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. Memory backup battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high-voltage circuits (marked  and fitted with an insulating cover).

Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.

If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.

Refer to the Chapter IV-1 for details of the battery replacement procedure.

WARNING**2. Absolute pulse coder battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing the batteries, be careful not to touch the high-voltage circuits (marked  and fitted with an insulating cover).

Touching the uncovered high-voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.

If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.

When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.

Refer to the Chapter IV-1 for details of the battery replacement procedure.

WARNING**3.Fuse replacement**

For some units, the chapter covering daily maintenance in the operator's manual or programming manual describes the fuse replacement procedure.

Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.

For this reason, only those personnel who have received approved safety and maintenance training may perform this work.

When replacing a fuse with the cabinet open, be careful not to touch the high-voltage circuits (marked  and fitted with an insulating cover).

Touching an uncovered high-voltage circuit presents an extremely dangerous electric shock hazard.

TABLE OF CONTENTS

DEFINITION OF WARNING, CAUTION, AND NOTE.....	2
GENERAL WARNINGS AND CAUTIONS	3
WARNINGS AND CAUTIONS RELATED TO PROGRAMMING.....	5
WARNINGS AND CAUTIONS RELATED TO HANDLING.....	8
WARNINGS RELATED TO DAILY MAINTENANCE	11
□ GENERAL.....	21
1 GENERAL.....	23
1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL	25
1.2 NOTES ON READING THIS MANUAL	27
II OPERATION	28
1 GENERAL.....	30
1.1 MANUAL OPERATION	31
1.2 TOOL MOVEMENT BY PROGRAMING - AUTOMATIC OPERATION.....	33
1.3 AUTOMATIC OPERATION	35
1.4 TESTING A PROGRAM.....	37
1.4.1 Check by Running the Machine	37
1.4.2 How to View the Position Display Change without Running the Machine	39
1.5 EDITING A PART PROGRAM	40
1.6 DISPLAYING AND SETTING DATA.....	41
1.7 DISPLAY.....	45
1.7.1 Program Display	45
1.7.2 Current Position Display	47
1.7.3 Operator Message Display.....	48
1.7.4 Alarm Display	49
1.7.5 Parts Count Display, Run Time Display.....	50
1.7.6 Graphic Display	51
1.8 DATA INPUT /OUTPUT	52
2 OPERATIONAL DEVICES	53
2.1 SETTING AND DISPLAY UNITS.....	55
2.1.1 9.5"/10.5" LCD Unit	55
2.1.2 MDI unit.....	56
2.1.3 MDI Unit (Full-keyboard)	57
2.2 EXPLANATION OF THE KEYBOARD	58
2.3 OPERATION SCREEN DISPLAY	61
2.4 OPERATIONS SUPPORTED BY EACH OPERATION SCREEN	62

2.5	FUNCTION KEYS AND SOFT KEYS	63
2.5.1	General Screen Operations	63
2.5.2	Function Keys.....	64
2.5.3	Soft Keys	65
2.5.4	Key Input and Input Buffer.....	65
2.5.5	Chapter selection keys	66
2.5.5.1	Position.....	66
2.5.5.2	Program	68
2.5.5.3	Offset/setting.....	69
2.5.5.4	System.....	71
2.5.5.5	Messages	73
2.5.5.6	Drawing	73
2.5.6	Key Input and Input Buffer.....	74
2.5.6.1	Inserting characters	74
2.5.6.2	Deleting characters	75
2.5.7	Warning Messages	76
2.6	\ f E •	77
2.7	EXTERNAL I/O DEVICES	78
2.7.1	FANUC Handy File	80
2.7.2	FANUC Floppy Cassette	81
2.7.3	FANUC FA Card.....	82
2.8	POWER ON/OFF	83
2.8.1	Turning on the Power.....	83
2.8.2	Power Disconnection.....	84
3	MANUAL OPERATION	85
3.1	MANUAL REFERENCE POSITION RETURN	86
3.2	JOG FEED	88
3.3	INCREMENTAL FEED	90
3.4	Manual Handle Feed	91
3.5	MANUAL FEED IN A SPECIFIED DIRECTION	93
3.6	MANUAL NUMERIC COMMAND	95
3.7	MANUAL ABSOLUTE ON AND OFF	104
3.8	EFFECT OF MANUAL INTERVENTION	111
3.9	THREE-DIMENSIONAL HANDLE FEED	112
3.9.1	Handle feed interruption in the longitudinal direction of the tool	113
3.9.2	Handle feed in the transverse direction of the tool	117
3.9.3	Rotational handle feed around the center of the tool tip.....	122
3.10	CHANGING TOOL LENGTH COMPENSATION IN THE LONGITUDINAL DIRECTION OF THE TOOL	126
3.11	Tool Holder Offset	127
3.12	128
3.13	PRECAUTIONS DURING 3-DIMENSIONAL HANDLE FEED	129
3.14	DISPLAYING THE COORDINATES OF THE TOOL TIP	133

3.15	Displaying Pulse Values and Amount of Movement by Manual Interrupt	135
3.16	MANUAL INTERRUPTION FUNCTION FOR THREE-DIMENSIONAL COORDINATE CONVERSION	138
4	AUTOMATIC OPERATION	140
4.1	DNC OPERATION	141
4.2	MEMORY OPERATION	142
4.3	MDI OPERATION	143
4.4	SELECTION OF EXECUTION PROGRAMS	145
4.4.1	Program Number Search.....	145
4.4.2	Search by sequence number	146
4.4.3	Rewind	147
4.5	EXECUTING AUTOMATIC OPERATION	148
4.6	STOP/TERMINATION OF AUTOMATIC OPERATION	149
4.7	PROGRAM RESTART	152
4.8	OUTPUT OF PROGRAM RESTART M, S, T AND B CODES	166
4.9	Block Restart	170
4.10	TOOL WITHDRAWAL AND RETURN	180
4.11	RETRACE	193
4.12	ACTIVE BLOCK CANCEL	204
4.13	MANUAL HANDLE INTERRUPT	207
4.14	MANUAL INTERVENTION AMOUNT RETURN DURING AUTOMATIC OPERATION	211
4.15	SIMULTANEOUS AUTOMATIC AND MANUAL OPERATION	212
5	TEST OPERATION	215
5.1	MACHINE LOCK AND AUXILIARY FUNCTION LOCK	216
5.2	DRY RUN	218
5.3	SINGLE BLOCK	220
6	SAFETY FUNCTIONS	222
6.1	EMERGENCY STOP	223
6.2	OVERTRAVEL	224
6.3	STROKE CHECK	226
6.4	STROKE LIMIT CHECK PRIOR TO PERFORMING MOVEMENT	232
7	ALARMS AND SELF DIAGNOSTIC FUNCTIONS	235
7.1	INTERNAL POSITION COMPENSATION DATA DISPLAY FUNCTION	236
7.2	COORDINATE SYSTEM RELATED DATA DISPLAY	245
7.3	PERIODIC MAINTENANCE SCREEN	246
7.4	MAINTENANCE INFORMATION SCREEN	258

7.5	SYSTEM LOG SCREEN	259
7.5.1	Displaying the contents of memory	262
7.5.2	262
8	EDITING PROGRAMS	1
8.1	INPUTTING AND OUTPUTTING PROGRAMS	3
8.1.1	Inputting programs.....	3
8.1.2	OUTPUTTING PROGRAMS	12
8.1.3	Collating programs.....	21
8.1.4	Keys and program encryption	25
8.1.5	Stored program lengths and number of registerable programs	29
8.2	Cursor	30
8.2.1	Cursor/editing units/words	30
8.2.2	Positioning the cursor	31
8.3	Program Number Search	33
8.4	Word Search/Address Search	35
8.4.1	Word search	35
8.4.2	Address search	36
8.4.3	Search repeat function.....	37
8.5	Inserting words and blocks	38
8.6	Changing words and blocks	39
8.6.1	Overwriting words.....	39
8.6.2	Replacing words and character strings.....	39
8.6.3	Replacing addresses.....	43
8.7	DELETING WORDS AND BLOCKS	46
8.8	CUTTING AND PASTING	47
8.8.1	Basic operations	47
8.8.2	Copying, moving, and deleting words and blocks.....	49
8.9	CANCELING THE EFFECTS OF EDITING WORDS AND BLOCKS	51
8.10	COPYING PROGRAMS	53
8.11	MERGING PROGRAMS	54
8.12	DELETING PROGRAMS	55
8.13	RESTORING A DELETED PROGRAM	57
8.14	BACKGROUND EDITING	58
8.15	EDITING TWO PROGRAMS SIMULTANEOUSLY	61
8.16	EDITING PROGRAMS IN OPERATION	63
8.17	CONDENSING THE PROGRAM MEMORY	64
8.18	CREATING PROGRAMS USING THE MDI PANEL	66
8.19	PLAYBACK	67
8.20	AUTOMATICALLY INSERTING SEQUENCE NUMBERS	72
8.21	STAMPING THE MACHINING TIME	73
9	SETTING AND DISPLAYING DATA	76
9.1	DISPLAYING AND SETTING TOOL OFFSET VALUES	77
9.2	DISPLAYING AND SETTING WORKPIECE ORIGIN OFFSETS	84

9.3	DISPLAY AND SETTING OF SETTING PARAMETERS	89
9.4	DISPLAYING AND SETTING CUSTOM MACRO VARIABLE VALUES.....	95
9.5	INPUT/OUTPUT OPERATION OF CUSTOM MACRO COMMON VARIABLES ...	99
9.6	DISPLAYING AND SETTING PARAMETERS	104
9.7	DISPLAYING AND SETTING PITCH ERROR COMPENSATION DATA.....	110
9.8	Manual Tool Length Measurement	112
9.9	MANUAL WORKPIECE ORIGIN SETTING	116
9.10	Tool Length/Workpiece Origin Measurement.....	122
9.11	MENU SWITCHES	143
9.12	DISPLAYING AND SETTING TOOL LIFE MANAGEMENT DATA	145
9.13	DISPLAYING AND SETTING THE SOFTWARE OPERATOR'S PANEL.....	157
9.14	DISPLAYING AND SETTING TOOL OFFSET DATA BASED ON TOOL NUMBERS	162
9.15	FLOATING REFERENCE POSITION SETTING	176
9.16	SERVO SPINDLE SCREEN.....	178
9.17	COMMUNICATION SETTING SCREEN.....	179
9.18	HIGH-SPEED HIGH-PRECISION MACHINING SETTING SCREEN	183
10	DISPLAY	192
10.1	PROGRAM DISPLAY.....	193
10.2	PROGRAM AND SEQUENCE NUMBER DISPLAY, STATUS DISPLAY, AND DATA SETTING AND INPUT/OUTPUT OPERATION WARNING DISPLAY	200
10.3	PROGRAM AND SEQUENCE NUMBER DISPLAY.....	201
10.4	STATUS DISPLAY AND DATA SETTING AND INPUT/OUTPUT OPERATION WARNING DISPLAY.....	202
10.5	CURRENT POSITION DISPLAY	206
10.5.1	Presetting the current position	209
10.5.2	Presetting a workpiece coordinate system.....	211
10.5.3	213
10.5.4	215
10.6	OPERATOR MESSAGE DISPLAY.....	218
10.7	ALARM MESSAGE DISPLAY	220
10.8	TOOL PATH DRAWING.....	222
10.9	Operating monitor screen.....	235
10.10	DISPLAY AND SETTING OF THE PARTS COUNTS, OPERATING TIME, AND CLOCK	237
10.11	BACKGROUND DRAWING.....	241
10.12	FLOPPY DIRECTORY SCREEN	260
10.13	MEMORY CARD SCREEN.....	267
11	CLEARING THE SCREEN	274
11.1	CLEARING THE SCREEN	275
11.1.1	Screen clear function	275

11.1.2 Automatic screen clear function.....	275
12 INPUTTING/OUTPUTTING DATA.....	276
12.1 FILES	278
12.2 SPECIFYING PARAMETERS REQUIRED FOR INPUT/OUTPUT	281
12.2.1 Setting parameter screen	281
12.2.2 Communication setting screen.....	283
12.3 INPUTTING AND OUTPUTTING PROGRAMS.....	286
12.3.1 Outputting Programs	286
12.3.2 Program Inputting	293
12.4 INPUTTING AND OUTPUTTING SYSTEM PARAMETERS.....	298
12.4.1 Outputting System Parameters	298
12.4.2 Inputting System Parameters.....	300
12.5 INPUTTING AND OUTPUTTING TOOL OFFSET DATA.....	302
12.5.1 Outputting Tool Offset Data	302
12.5.2 Inputting Tool Offset Data.....	304
12.6 INPUTTING AND OUTPUTTING WORKPIECE ZERO POINT OFFSET DATA...306	
12.6.1 Outputting Workpiece zero point offset data.....	306
12.6.2 Inputting System Parameters.....	308
12.7 INPUTTING AND OUTPUTTING CUSTOM MACRO VARIABLE	310
12.7.1 Outputting Cuctom Macro Variable.....	310
12.7.2 Inputting Cuctom Macro Variable	312
12.8 INPUTTING AND OUTPUTTING THREE-DIMENSIONAL ERROR COMPENSATION DATA	314
12.8.1 Outputting Three-dimensional Error Compensation Data.....	314
12.8.2 Inputting Three-dimensional Error Compensation Data	316
12.9 INPUTTING AND OUTPUTTING TOOL OFFSET DATA BY TOOL NUMBER318	
12.9.1 Outputting Tool Offset Data by Tool Number.....	318
12.9.2 Inputting Tool Offset Data by Tool Number	320
12.10 INPUTTING AND OUTPUTTING FIXTURE OFFSET DATA	322
12.10.1.....	Outputting Fixture Offset Data 322
12.10.2.....	Inputting Fixture Offset Data 324
12.11 INPUTTING AND OUTPUTTING ROTARY HEAD DYNAMIC TOOL OFFSET DATA326	
12.11.1.....	Outputting Rotary Head Dynamic Tool Offset Data 326
12.11.2.....	Inputting Rotary Head Dynamic Tool Offset Data 328
12.12 INPUTTING AND OUTPUTTING SYSTEM CONFIGURATION DATA	330
12.12.1.....	Outputting System Configuration Data 330
12.12.2.....	Inputting System Configuration Data 332
12.13 INPUTTING AND OUTPUTTING DATA USING THE FLOPPY DIRECTORY SCREEN	334
12.14 INPUTTING AND OUTPUTTING DATA USING THE MEMORY CARD SCREEN335	
12.15 External I/O Equipment control.....	336
13 AXIS CONTROL	337
13.1 AXIS CONTROL BY THE PMC.....	338
13.2 MIRROR IMAGE	340
14 ABSOLUTE-POSITION DETECTION.....	342

IV MAINTENANCE..... 346

1 INVESTIGATION OF TROUBLE GENERATION SITUATION. 348

1.1 DISPLAY OF SYSTEM CONFIGURATION SCREEN.....349

APPENDIX..... 351

T GENERAL

1

GENERAL

About this manual

This manual consists of the following parts:

T GENERAL

..Describes chapter organization, applicable models, related manuals, and notes for reading this manual.

U PROGRAMMING

..Describes each function: Format used to program functions in the NC language, characteristics, and restrictions. When a program is created through conversational automatic programming function, refer to the manual for the conversational automatic programming function (Table 1).

V OPERATION

..Describes the manual operation and automatic operation of a machine, procedures for inputting and outputting data, and procedures for editing a program.

W MAINTENANCE

..Describes procedures for replacing batteries.

APPENDIX

..Lists tape codes, valid data ranges, and alarms.

Some functions described in this manual may not be applied to some products. For detail, refer to the DESCRIPTIONS manual(B-63322EN).

This manual does not describe parameters in detail. For details on parameters mentioned in this manual, refer to the manual for parameters (B-63330EN).

This manual describes all optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.

The models covered by this manual, and their abbreviations are:

Applicable product name

Product name	Abbreviations	
FANUC Series 15i□-MA	15i□-MA	Series 15i□
FANUC Series 150i□-MA	150i□-MA	Series 150i□

Special symbols

This manual uses the following symbols:

- .____.Indicates a combination of axes such as X__ Y__ Z
(used in PROGRAMMING.).
- . .Indicates the end of a block.
It actually corresponds to the ISO code LF or EIA code CR.

Related manuals

The table below lists manuals related to MODEL A of Series 15i, and Series 150i. In the table, this manual is marked with an asterisk (*).

Table 11 (a) Related Manuals

Manual name	Specification number	
DESCRIPTIONS	B-63322EN	
CONNECTION MANUAL (Hardware)	B-63323EN	
CONNECTION MANUAL (Function)	B-63323EN-1	
OPERATOR'S MANUAL (PROGRAMMING) for Machining Center	B-63324EN	
OPERATOR'S MANUAL (OPERATION) for Machining Center	B-63324EN-1	
MAINTENANCE MANUAL	B-63325EN	
PARAMETER MANUAL	B-63330EN	

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL

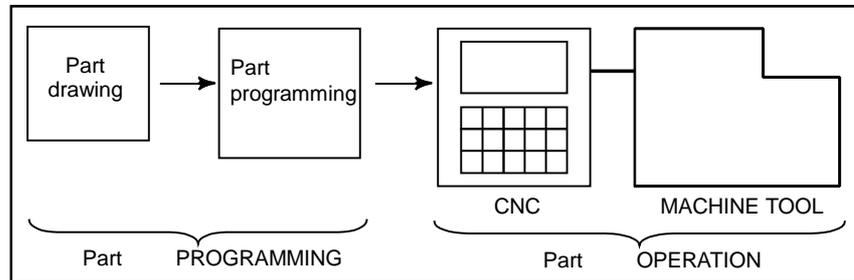
When machining the part using the CNC machine tool, first prepare the program, then operate the CNC machine by using the program.

- (1) First, prepare the program from a part drawing to operate the CNC machine tool.

How to prepare the program is described in the Part .. PROGRAMMING.

- (2) The program is to be read into the CNC system. Then, mount the workpieces and tools on the machine, and operate the tools according to the programming. Finally, execute the machining actually.

How to operate the CNC system is described in the Part .. OPERATION.



Before the actual programming, make the machining plan for how to machine the part.

Machining plan.1. Determination of workpieces machining range

- 2. Method of mounting workpieces on the machine tool

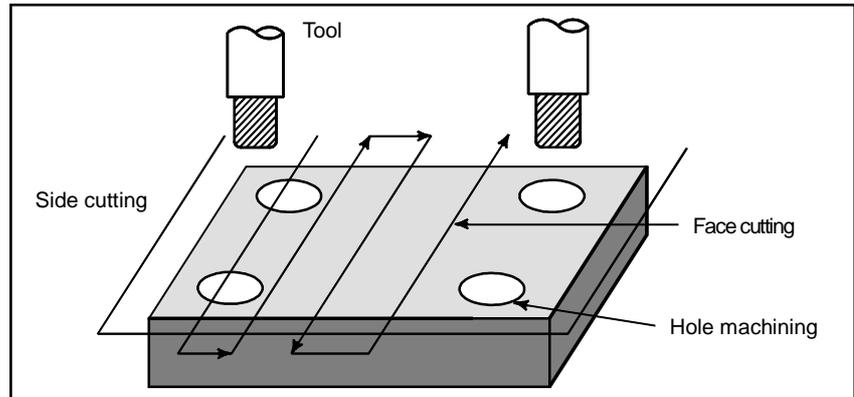
tool

- 3. Machining sequence in every machining process

- 4. Machining tools and machining

Decide the machining method in every machining process.

Machining process \ Machining procedure	P	Q	R
	Feed cutting	Side cutting	Hole machining
P □ Machining method : Rough " " Semi " " Finish			
Q □ Machining tools			
R □ Machining conditions : Feedrate " " Cutting depth			
S □ Tool path			



Prepare the program of the tool path and machining condition according to the workpiece figure, for each machining.

1.2 NOTES ON READING THIS MANUAL

NOTE

- 1 The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations.
This manual generally describes these from the stand-point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.
- 2 Headings are placed in the left margin so that the reader can easily access necessary information. When locating the necessary information, the reader can save time by searching through these headings.
- 3 Machining programs, parameters, variables, etc. are stored in the CNC unit internal non-volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non-volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration.
In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.
- 4 This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
If a particular combination of operations is not described, it should not be attempted.

II OPERATION

1

GENERAL

1.1 MANUAL OPERATION

Explanations

£ Manual reference position return

The CNC machine tool has a position used to determine the machine position.

This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on, the tool is moved to the reference position.

Manual reference position return is to move the tool to the reference position using switches and pushbuttons located on the operator's panel.

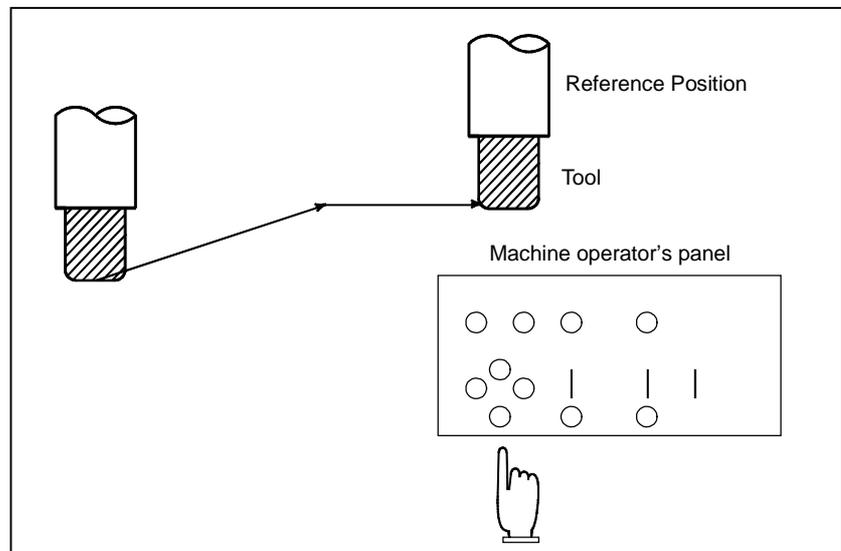


Fig.1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.

This operation is called automatic reference position return (See Section II-6).

£ The tool movement by manual operation

Using machine operator's panel switches, pushbuttons, or the manual handle, the tool can be moved along each axis.

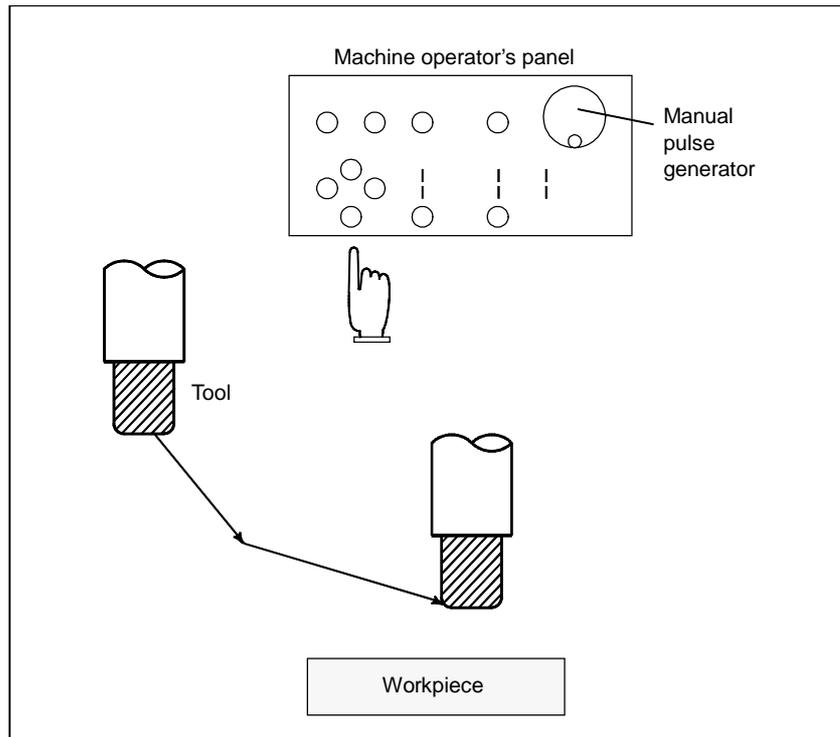


Fig.1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

- (i) Jog feed (See Section III-3.2)
The tool moves continuously while a pushbutton remains pressed.
- (ii) Incremental feed (See Section III-3.3)
The tool moves by the predetermined distance each time a button is pressed.
- (iii) Manual handle feed (See Section III-3.4)
By rotating the manual handle, the tool moves by the distance corresponding to the degree of handle rotation.
- (-) Manual feed at an arbitrary angle (See Section III-3.?)
The tool moves in an arbitrary direction on a plane.
- (-) Manual numeric command (See Section III-3.?)
By entering numeric data from the MDI keyboard, the tool moves through the distance corresponding to the entered data.

1.2 TOOL MOVEMENT BY PROGRAMING – AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program.

It includes memory, MDI and DNC operations. (See Section III-4).

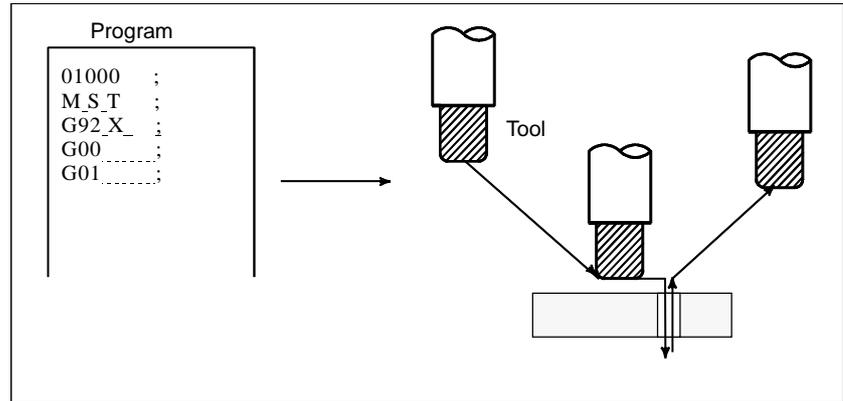


Fig. 1.2 (a) Tool Movement by Programming

Explanations

£ Memory operation

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation.

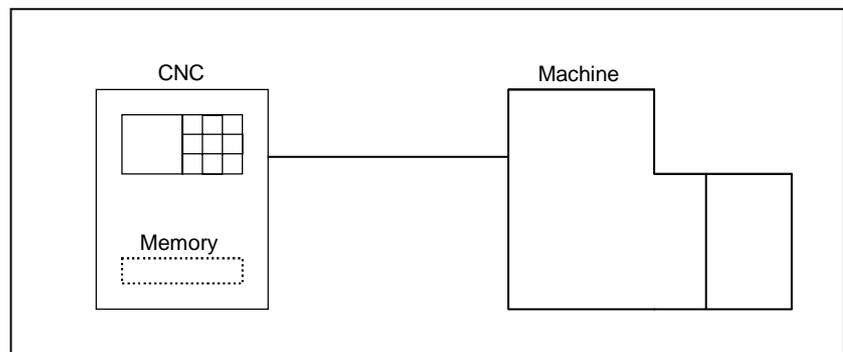


Fig. 1.2 (b) Memory Operation

£ MDI operation

After the program is entered, as an command group, from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation.

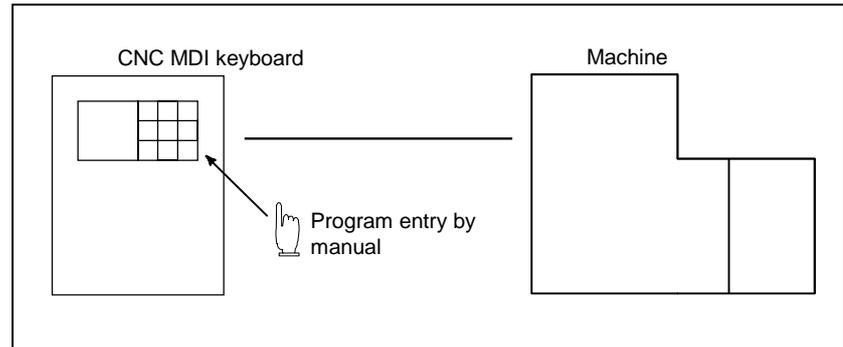


Fig.1.2 (c) MDI operation

£ DNC operation

In this mode of operation, the program is not registered in the CNC memory. It is read from the external input/output devices instead. This is called DNC operation. This mode is useful when the program is too large to fit the CNC memory.

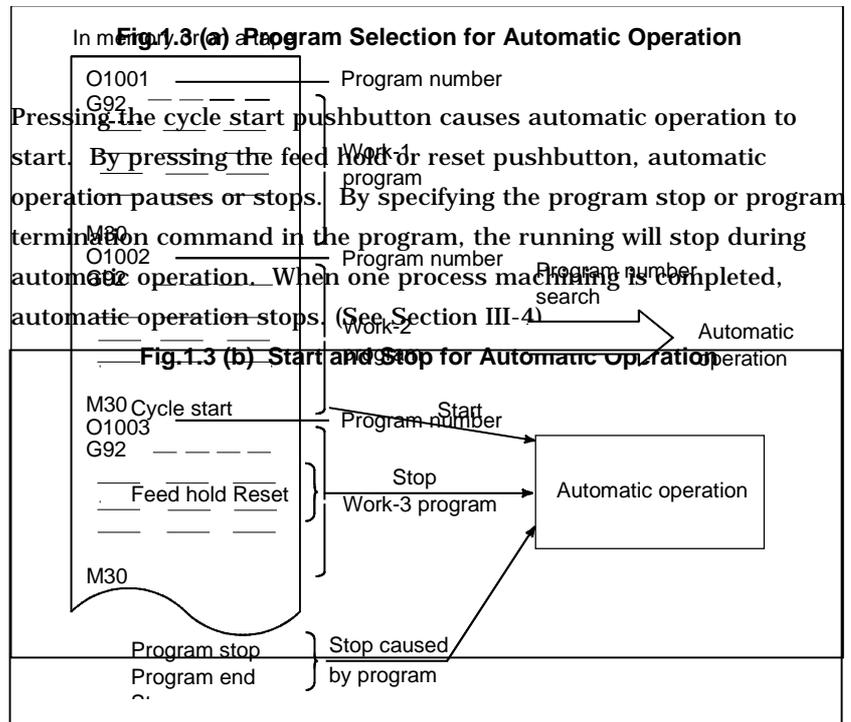
1.3 AUTOMATIC OPERATION

Explanations

£ Program selection

Select the program used for the workpiece. Ordinarily, one program is prepared for one workpiece. If two or more programs are in memory, select the program to be used, by searching the program number (Section III-9.3).

£ Start and stop



£ Restart

You can continue machining of paused workpieces. If, for example, the tool breaks during machining, you can replace the tool, and then restart machining. (Refer to III-?.)

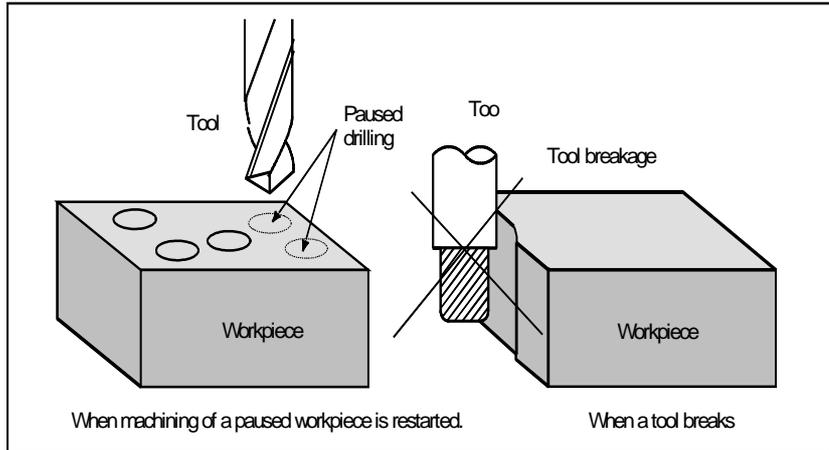


Fig.1.3 (c) Restart on automatic operation

£ Handle interruption

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle. (See Section III-4.8)

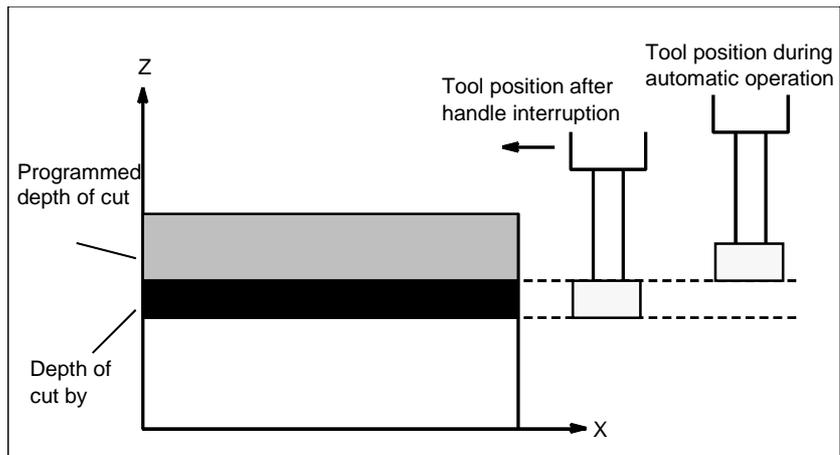


Fig.1.3 (d) Handle Interruption for Automatic Operation

1.4 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed. It checks whether the created program can operate the machine as desired. This check can be accomplished by running the machine actually or viewing the position display change (without running the machine) (See Section III-5).

1.4.1 Check by Running the Machine

Explanations

£ Dry run

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel. (See Section III-5.4)

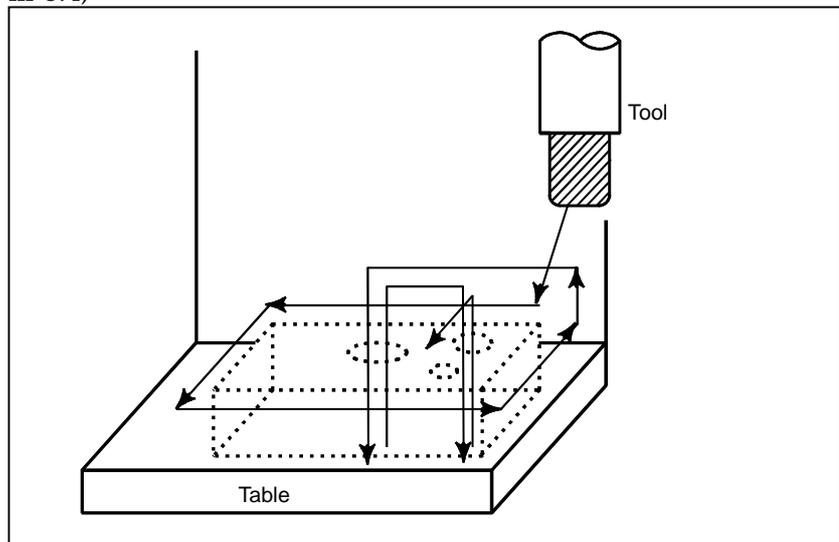


Fig.1.4 (a) Dry run

£ Feedrate override

Check the program by changing the feedrate specified in the program.
(See Section III-5.2)

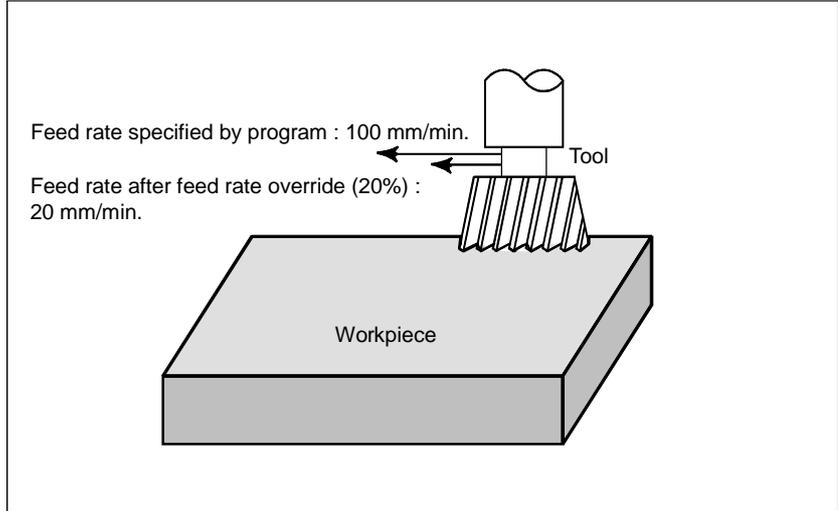


Fig 1.4 (b) Feedrate Override

£ Single block

Each time the cycle start button is pressed, one block of a program is executed, then the tool stops.
A program is checked by executing it on a block-by-block basis. (See Section III-5.5.)

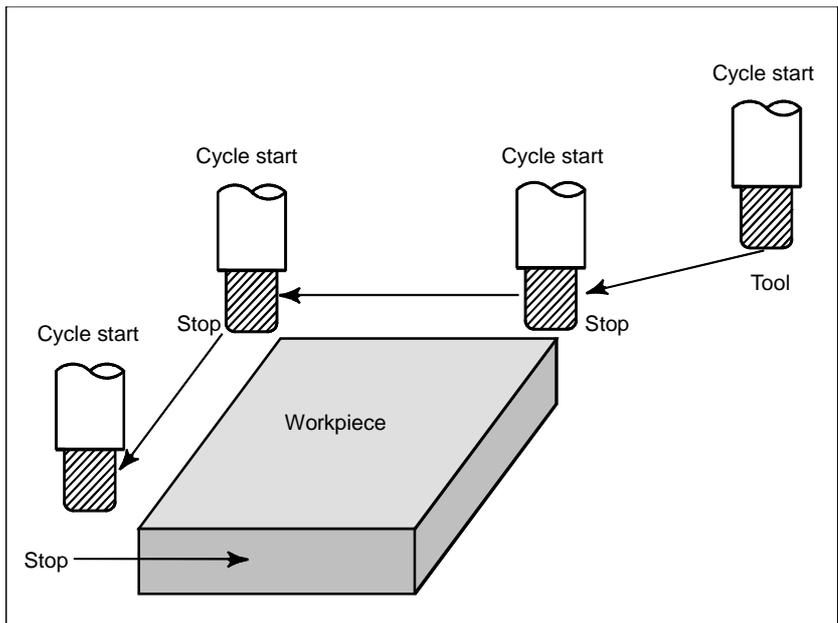


Fig.1.4 (c) Single Block

1.4.2 How to View the Position Display Change without Running the Machine

Explanations

£ Machine lock

The machine tool does not move, and only the position coordinates are updated.

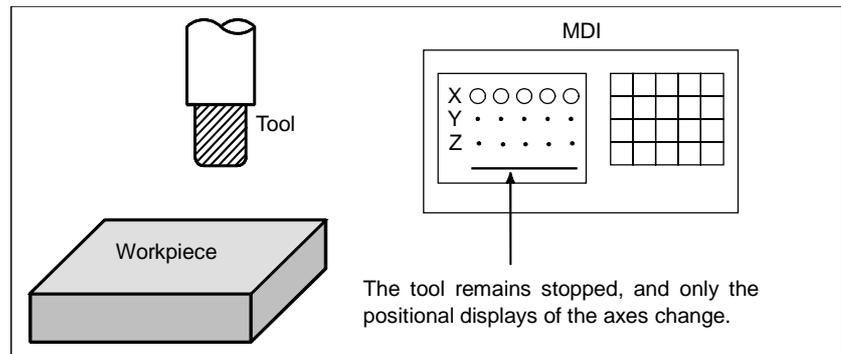


Fig 1.4 (d) Machine Lock

£ Auxiliary function lock

When automatic running is placed into the auxiliary function lock mode during the machine lock mode (See Sections III-5.1), all auxiliary functions (spindle rotation, tool replacement, coolant on/off, etc.) are disabled. (See Section III-5.1)

1.5 EDITING A PART PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the MDI panel (See Section III-9).

This operation can be executed using the part program storage/edit function.

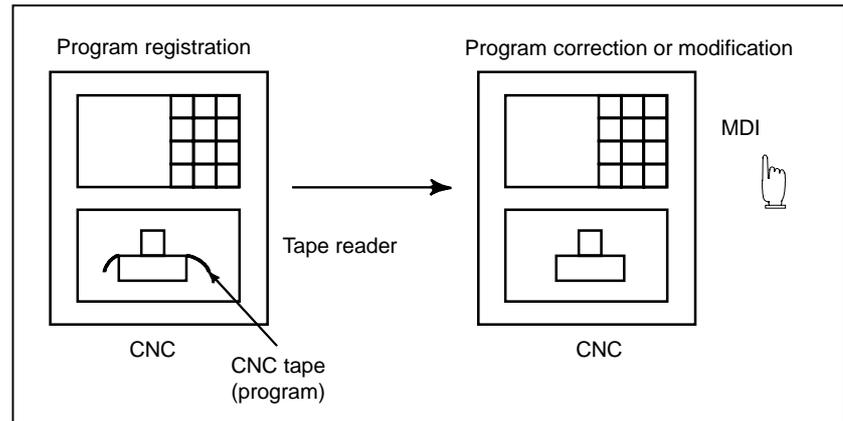


Fig.1.5 (a) Part Program Editing

1.6 DISPLAYING AND SETTING DATA

The operator can display or change a value stored in CNC internal memory by key operation on the MDI screen (See III-11).

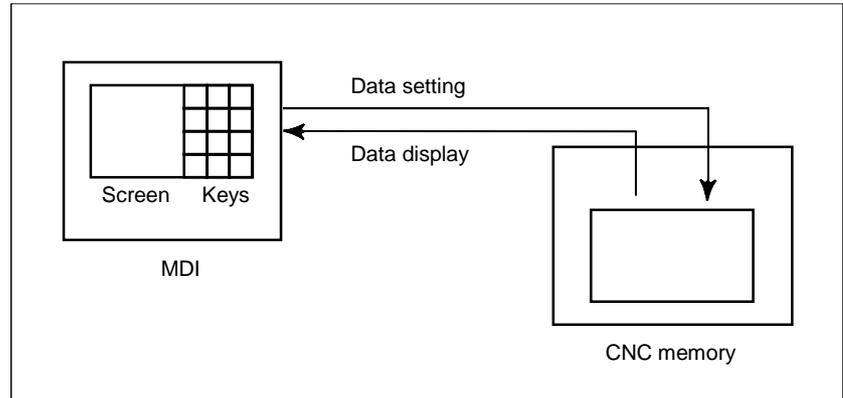


Fig.1.6 (a) Displaying and Setting Data

Explanations

£ Offset value

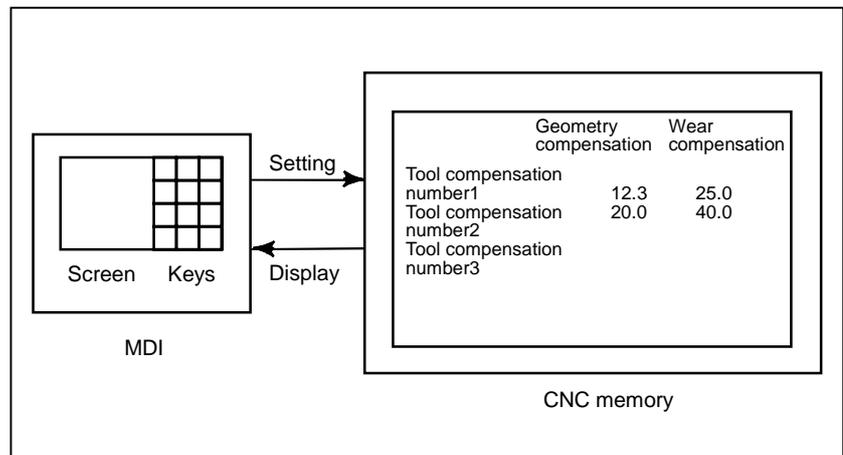


Fig.1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement value depends on the tool dimensions.

By setting tool dimension data in CNC memory beforehand, automatically generates tool routes that permit any tool to cut the workpiece specified by the program. Tool dimension data is called the offset value (See Section III-11.4.1).

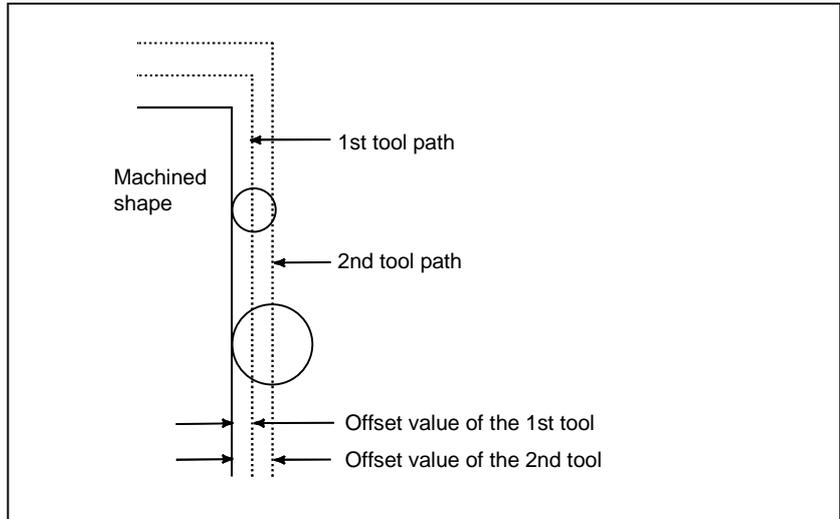


Fig.1.6 (c) Offset Value

£ **Displaying and setting operator's setting data**

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.

For example, the following data can be set:

- . Inch/Metric switching
- . Selection of I/O devices
- . Mirror image cutting on/off

The above data is called setting data (See Section III-11.4.3).

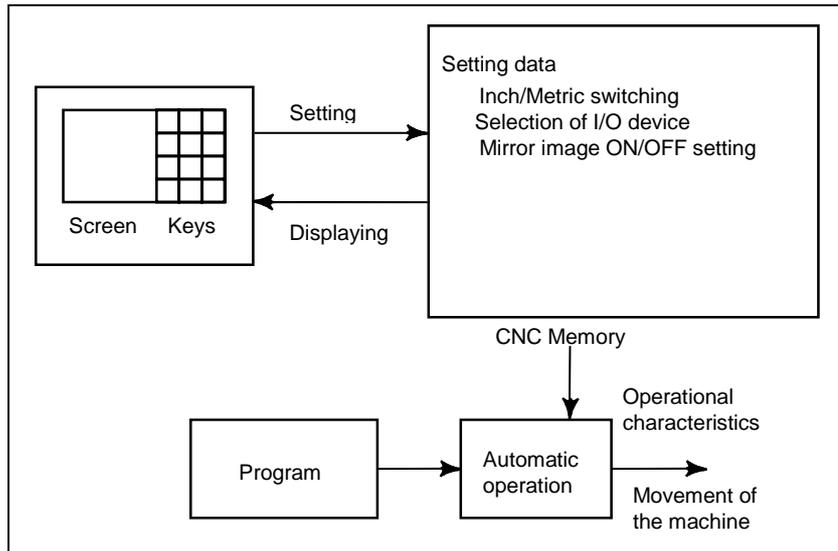


Fig.1.6 (d) Displaying and Setting Operator's setting data

£ **Displaying and setting parameters**

The CNC functions have versatility in order to take action in characteristics of various machines.

For example, CNC can specify the following:

- Rapid traverse rate of each axis
- Whether increment system is based on metric system or inch system.
- How to set command multiply/detect multiply (CMR/DMR)

Data to make the above specification is called parameters (See Section III-11.5.1).

Parameters differ depending on machine tool.

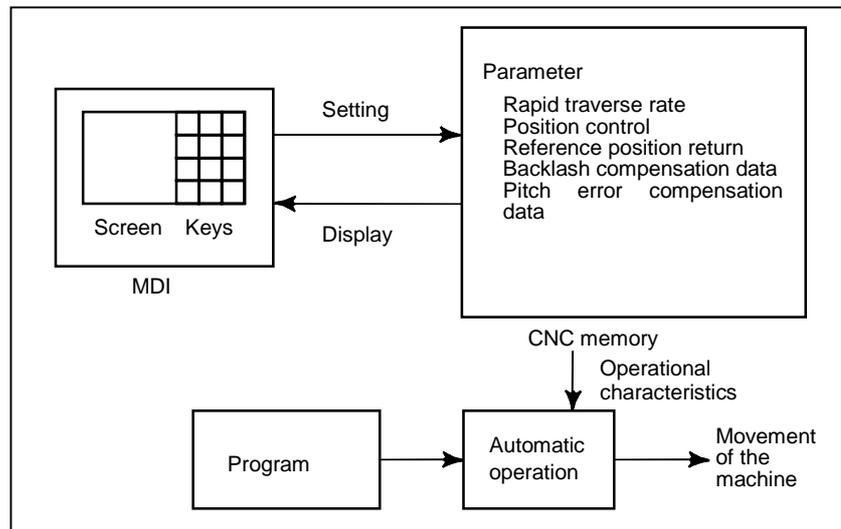


Fig.1.6 (e) Displaying and setting parameters

£ **Data protection key**

A key called the data protection key can be defined. It is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously (See Section III-11).

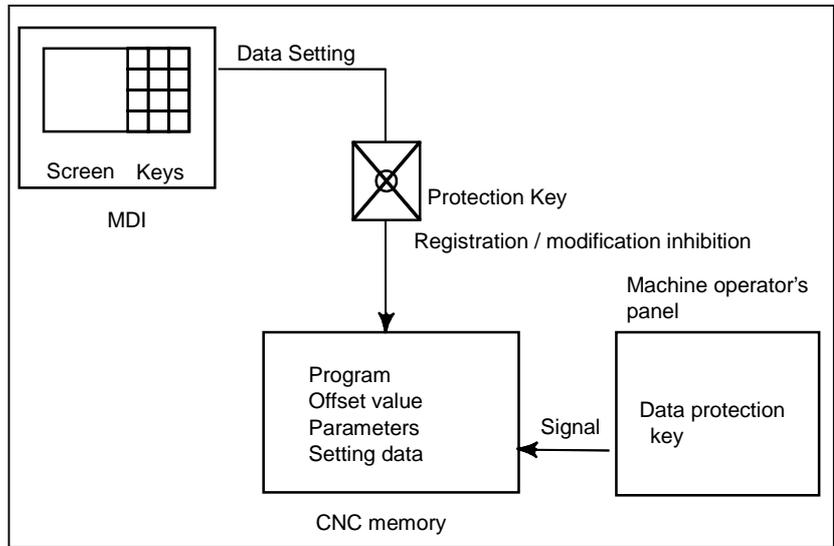


Fig.1.6 (f) Data Protection Key

1.7 DISPLAY

1.7.1 Program Display

The contents of the currently active program are displayed. In addition, the programs scheduled next and the program list are displayed.(See Section III-11.2.1)

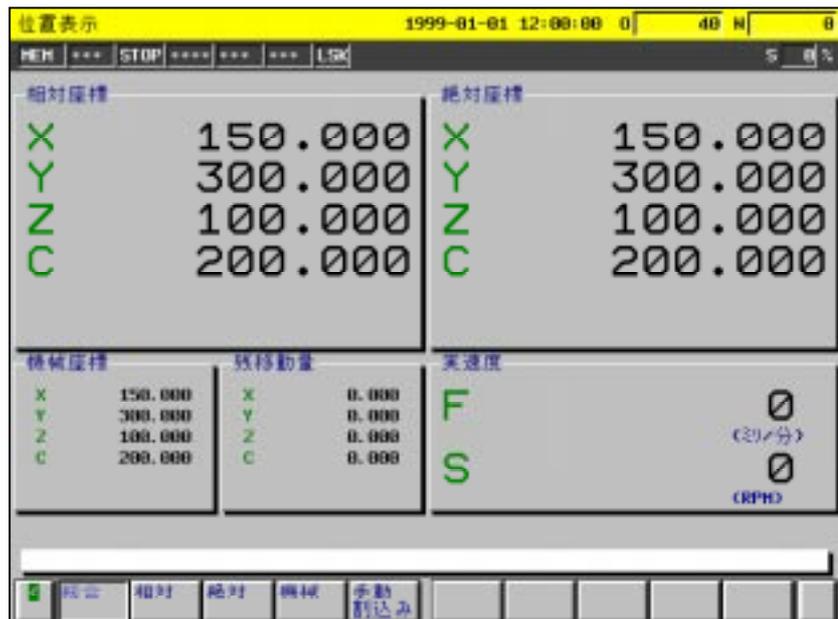
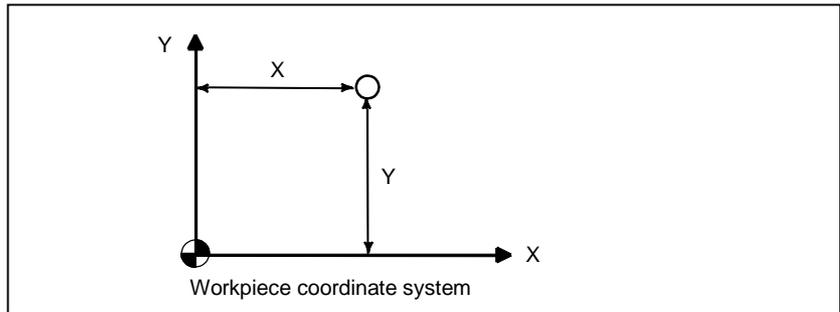
Fig.1.7 (a) Currently executed program



Fig.1.7 (b) Program list

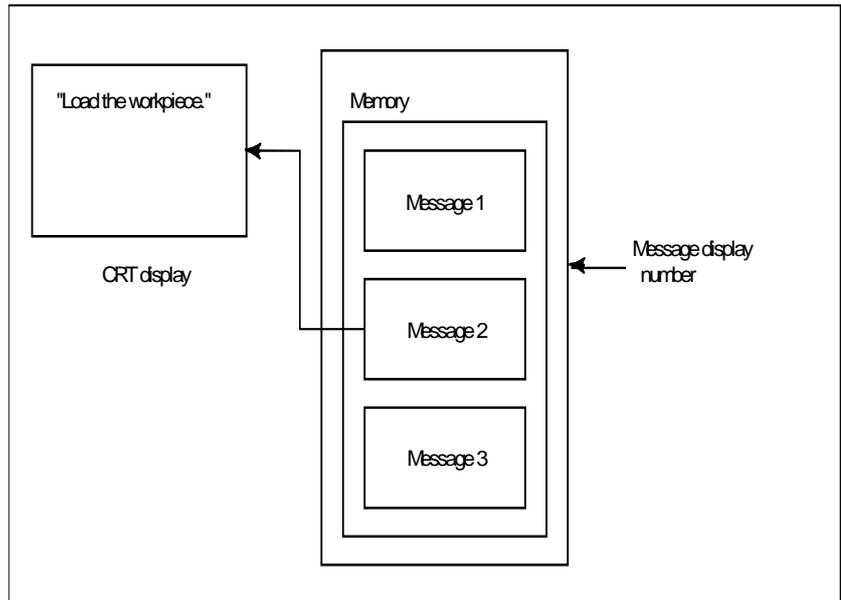
1.7.2 Current Position Display

The current position of the tool is displayed with the coordinate values. The distance from the current position to the target position can also be displayed. (See Section III-11.1.1, 11.1.2, and 11.1.3)



1.7.3 Operator Message Display

Messages stored to memory are displayed when the operator must be instructed during machining. (Refer to ???)



1.7.4 Alarm Display

When a trouble occurs during operation, error code and alarm message are displayed on the screen. (See Section III-7.1)
See APPENDIX G for the list of error codes and their meanings

1.7.5 Parts Count Display, Run Time Display

When this option is selected, five types of run time and number of parts are displayed on the screen. (See Section III-11.4.5)

1.7.6 Graphic Display

Programmed tool movement can be displayed on the following planes:

(See Section III-12)

- 1) XY plane
- 2) YZ plane
- 3) XZ plane
- 4) Three dimensional display

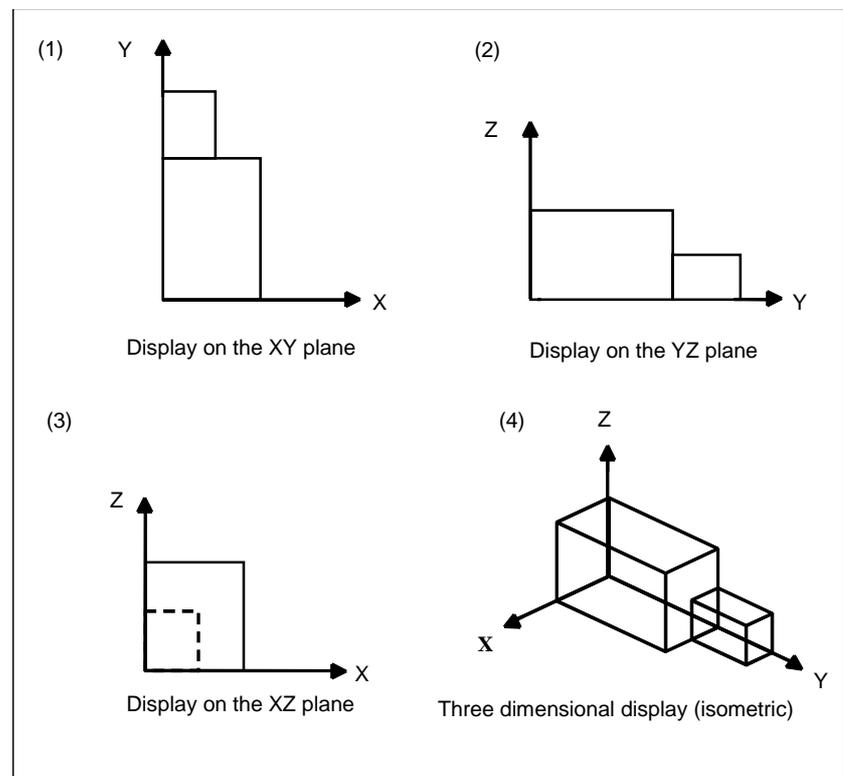
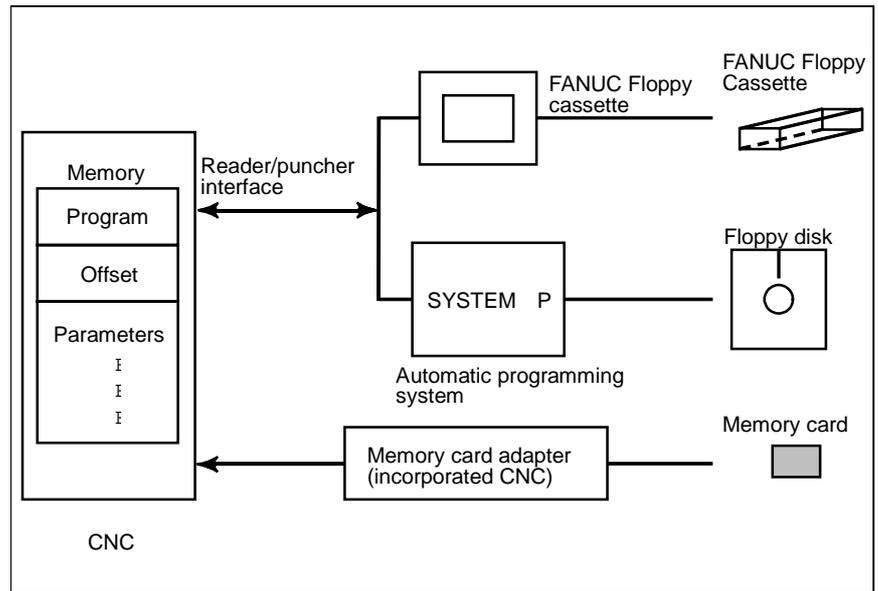


Fig. 1.7 (c) Graphic display

1.8 DATA INPUT /OUTPUT

Programs, offset values, parameters, etc. input in CNC memory can be output to paper tape, cassette, or a floppy disk for saving. After once output to a medium, the data can be input into CNC memory.



(See III-8.)

Fig.1.8 (a) Data Output

2

OPERATIONAL DEVICES

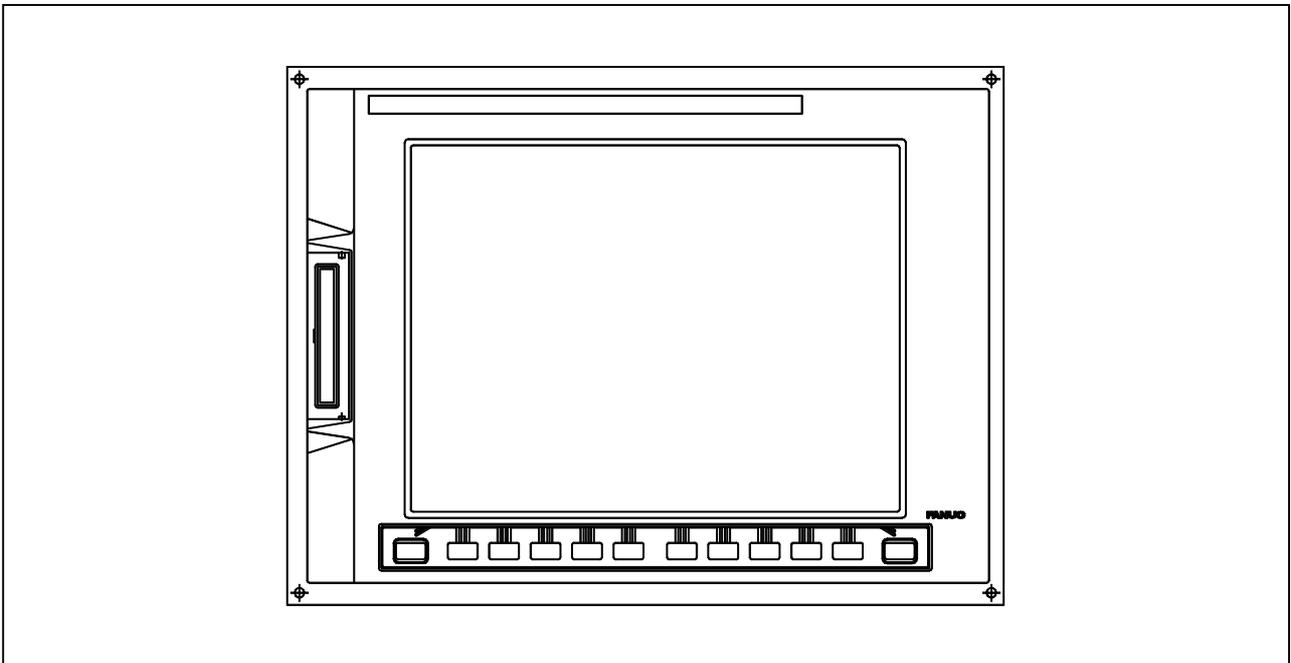
The available operational devices include the setting and display unit attached to the CNC, the machine operator's panel, and external input/output devices such as a PPR, Handy File, Floppy Cassette, and FA Card.

2.1 SETTING AND DISPLAY UNITS

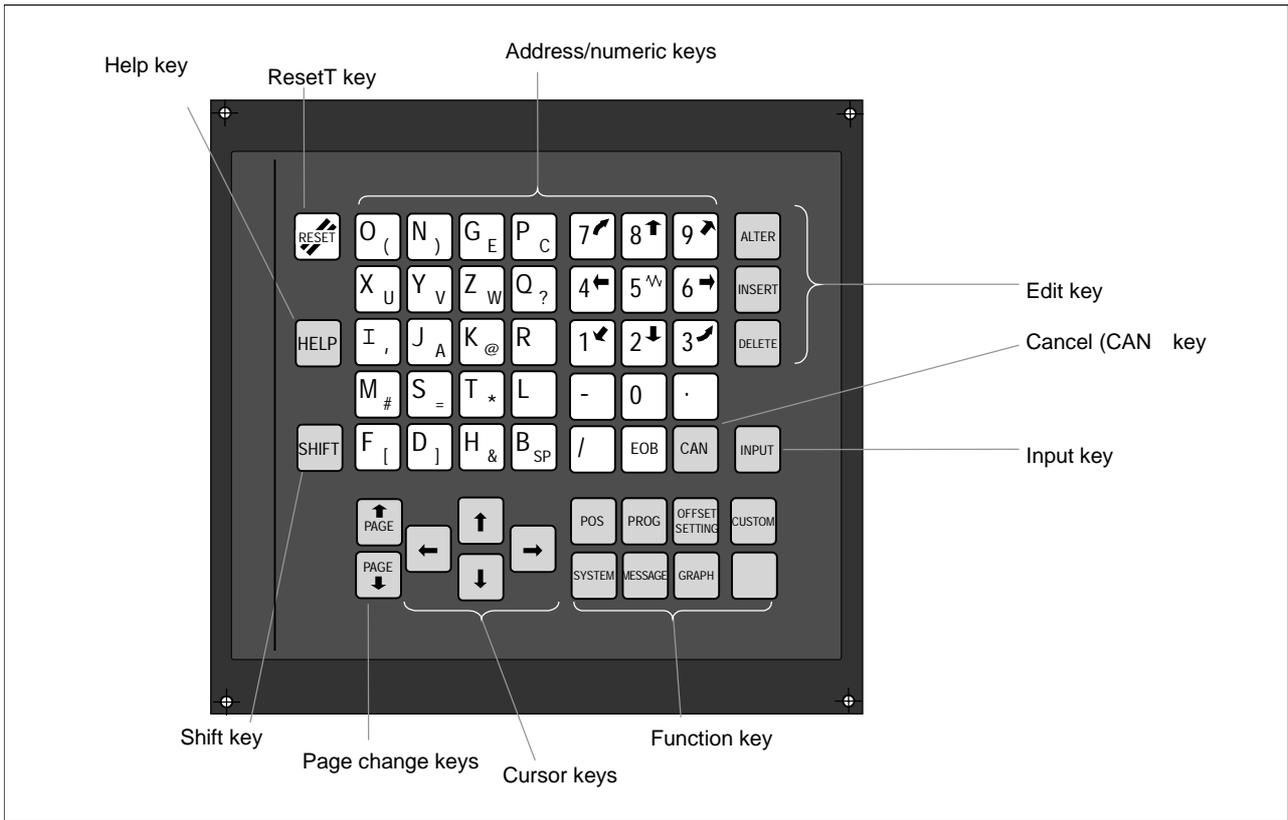
The setting and display units are shown in Subsections 2.1.1 to 2.1.3 of Part III.

9.5"/10.4" LCD unit	III-2.1.1
MDI unit	III-2.1.2
MDI unit (full-keyboard)	III-2.1.3

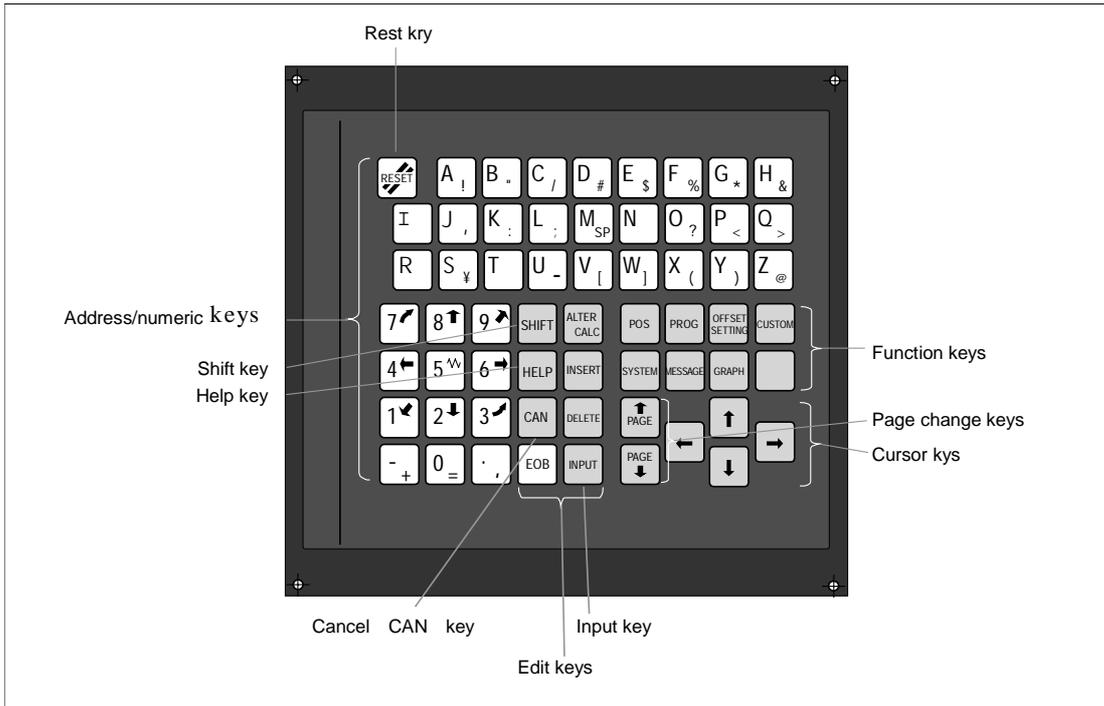
2.1.1 9.5"/10.5" LCD Unit



2.1.2 MDI unit

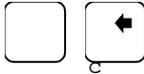
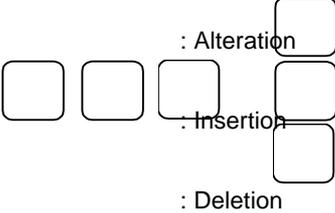


2.1.3 MDI Unit (Full-keyboard)



2.2 EXPLANATION OF THE KEYBOARD

Table2.2 (a) Explanation of the MDI keyboard

Number	Name	Explanation
1	RESET key	Press this key to reset the CNC, to cancel an alarm, etc.
2	HELP key	Press this button to use the help function when uncertain about the operation of an MDI key (help function).
3	Soft keys	The soft keys have various functions, according to the Applications. The soft key functions are displayed at the bottom of the screen.
4	Address and numeric keys 	Press these keys to input alphabetic, numeric, and other characters.
5	SHIFT key 	Some keys have two characters on their keytop. Pressing the <SHIFT> key switches the characters. Special character # is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered.
6	INPUT key 	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the screen. To copy the data in the key input buffer to the offset register, etc., press the key. This key is equivalent to the [INPUT] key of the soft keys, and either can be pressed to produce the same result.
7	Cancel key 	Press this key to delete the last character or symbol input to the key input buffer. When the key input buffer displays >N001X100Z_  and the cancel key is pressed, Z is canceled and >N001X100_ is displayed.
8	Program edit keys 	Press these keys when editing the program. 
9	Function keys 	Press these keys to switch display screens for each function. See III - 2.3 for details of the function keys.

--	--	--

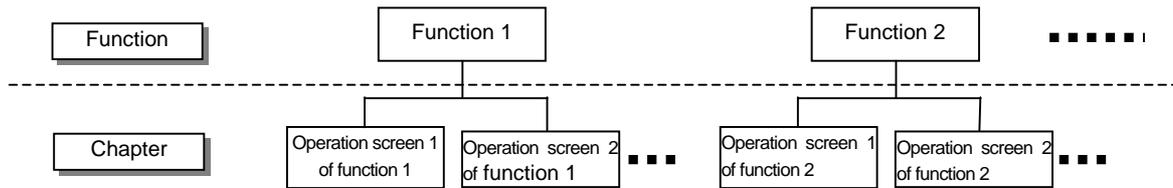
Table2.2 (a) Explanation of the MDI keyboard

Number	Name	Explanation
10	Cursor move keys	<p>There are four different cursor move keys.</p> <ul style="list-style-type: none"> : This key is used to move the cursor to the right or in the forward direction. The cursor is moved in short units in the forward direction. : This key is used to move the cursor to the left or in the reverse direction. The cursor is moved in short units in the reverse direction. : This key is used to move the cursor in a downward or forward direction. The cursor is moved in large units in the forward direction. : This key is used to move the cursor in an upward or reverse direction. The cursor is moved in large units in the reverse direction.
11	Page change keys	<p>Two kinds of page change keys are described below.</p> <ul style="list-style-type: none"> : This key is used to changeover the page on the screen in the forward direction. : This key is used to changeover the page on the screen in the reverse direction.

2.3 OPERATION SCREEN DISPLAY

The operation screens used in the CNC are classified into the following levels and managed hierarchically:

- Function
- Chapter



To display a desired operation screen, select a function then chapter. A function can be selected by using one of the two methods explained below. A desired function can be selected using either method.

- Using a function key on the MDI panel to select a function
When a function is selected using this method, the soft key display automatically changes to the chapter selection menu.
- Selecting a function displayed on a soft key
When a function is selected using this method, the soft key display does not automatically change to the chapter selection menu. To change the soft key display to the chapter selection menu, select the chapter key.

To select a chapter, use one of the following methods:

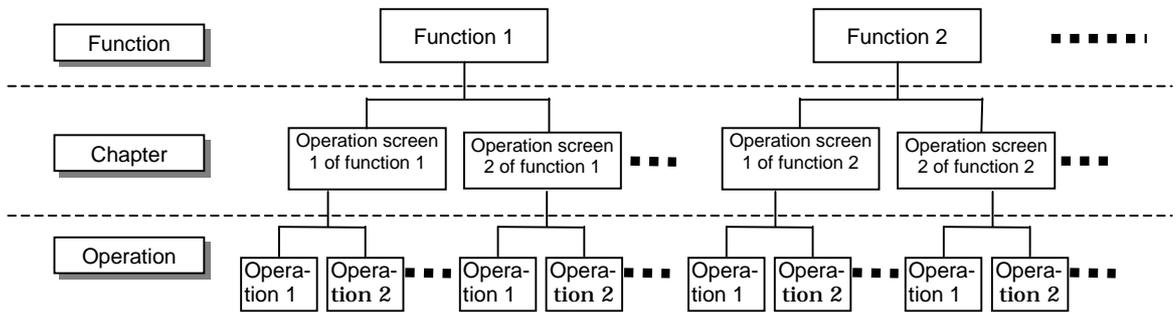
- Pressing a function key on the MDI panel several times repeatedly
Repeatedly pressing a function key displays the chapters of the corresponding function sequentially. After the last chapter is selected, the first chapter is selected again. This method is useful when you want to display all the chapters of a function.
- Selecting from the chapters displayed on the soft keys
Select from the chapter selection menu displayed on the soft keys. This method is useful when you determine the chapter of a target operation screen.

2.4 OPERATIONS SUPPORTED BY EACH OPERATION SCREEN

The operation screens support a wide range of operations.

On the program screen, for example, you can select and search for a desired program. On the setting screens, operations such as altering the settings and outputting the setting data to external input/output devices are possible. The operations that can be performed vary depending on the operation screens.

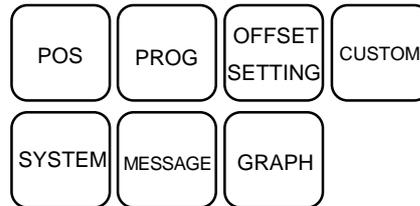
A desired operation can be selected and executed using an operation menu key (soft key) from an operation screen.



2.5 FUNCTION KEYS AND SOFT KEYS

2.5.1 General Screen Operations

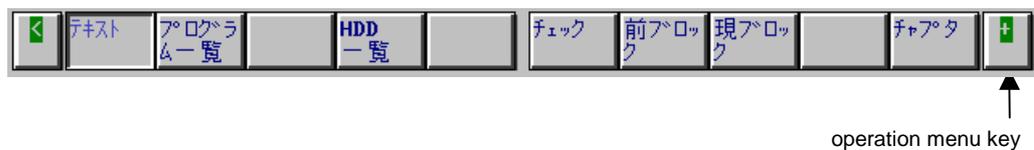
- 1 Press a function key on the MDI panel. The chapter selection soft keys that belong to the selected function appear.



- 2 Press one of the chapter selection soft keys. The screen for the selected chapter appears.
- 3 To perform operations on the displayed screen of the chapter, press the operation menu key.

For example, pressing the  key displays the chapter

selection soft keys shown below. When these soft keys are displayed for the first time, the first chapter is selected. (In this example, "TEXT" is selected.) To display the operation selection menu for this screen, press the operation menu key located at the right end.



- 4 To redisplay the chapter menu keys for chapter selection while the operation selection menu is being displayed, press the chapter key.



The general screen display procedure is explained above. However, the actual display procedure varies from one screen to another. For details, see the description of the individual operations.

2.5.2 Function Keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the MDI panel:



Press this key to display the position screen.



Press this key to display the program screen.



Press this key to display the offset/setting screen.



Press this key to display the system screen.



Press this key to display the message screen.



Press this key to display the graphics screen.

2.5.3 Soft Keys

Instead of the function keys, soft keys can also be used to select the type of screen to be displayed. In addition, soft keys are used for actual operations. The subsequent sections show a function menu and chapter selection menus.

NOTE

The soft keys explained below are not always displayed. Some of the soft keys do not appear depending on the set options.

2.5.4 Key Input and Input Buffer

To select a function with the soft key, press the function selection menu key to change the soft keys to the function selection keys, then press the desired function selection key.

Functions can be selected in any mode.

Detailed function selection is performed with the chapter selection keys.

The following function selection keys are provided:



Function menu key

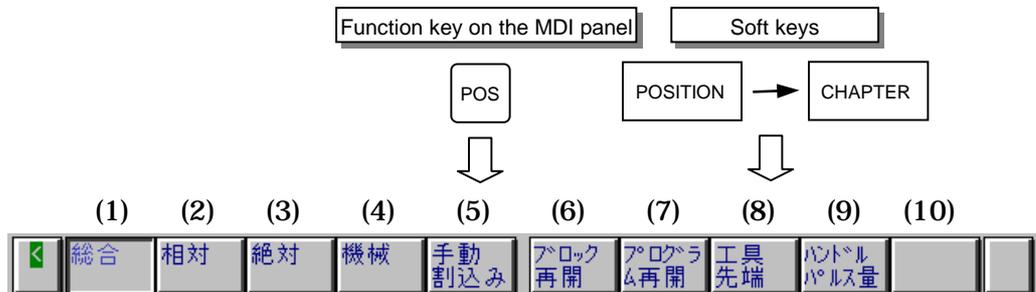
NO.	Function key (MDI panel)	Function selection key (soft key)	Explanation
(1)	POS	POSITION	Current-position information screens indicating absolute coordinates, machine coordinates, relative coordinates, remaining amounts of travel, and so forth can be selected.
(2)	PROG	PROGRAM	The part program display screen, program check screen, and so forth can be selected.
(3)	OFFSET SETTING	OFFSETSETTING	The tool offset screen, workpiece origin offset screen, and so forth can be selected.
(4)	SYSTEM	SYSTEM	The parameter screen, diagnosis screen, PMC screen, and so forth can be selected.
(5)	MESSAGE	MESSAGE	Alarm message and operator message screens can

			be selected.
(6)		GRAPHIC	Graphics screens can be selected.

2.5.5 Chapter selection keys

The function selection keys are used to select large items (functions). Each of them is further divided into smaller items (chapters). These items (chapters) are selected using chapter selection keys. To select a chapter, press the chapter key to display chapter selection keys on the soft keys first, then press the target chapter selection key. Change from one chapter to another can also be performed by pressing a function hard key several times repeatedly. The following sections list the chapters contained in each function.

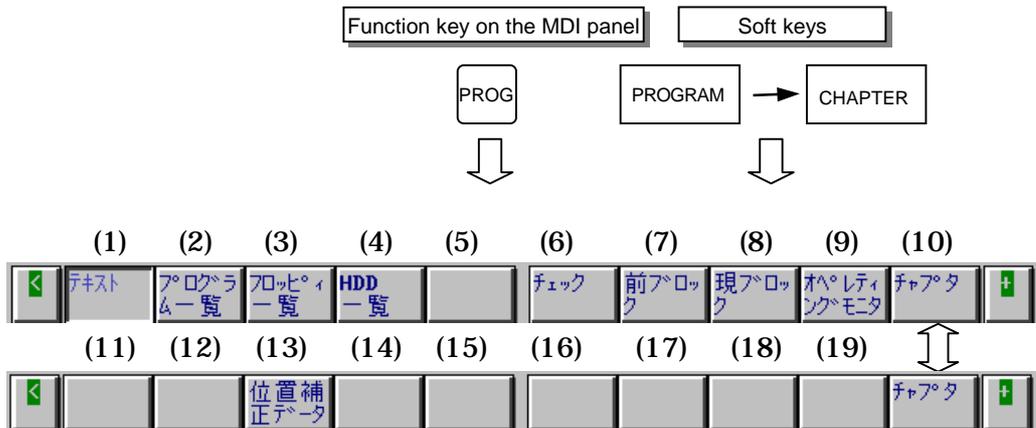
2.5.5.1 Position



NO.	Chapter menu	Explanation
(1)	OVERALL	Selects the overall position display screen.
(2)	RELATIVE	Selects the relative position display screen.
(3)	ABSOLUTE	Selects the position display screen based on the workpiece coordinate system.
(4)	MACHINE	Selects the position display screen based on the machine coordinate system.
(5)	MANUAL OVLAP	Selects the operation screen for performing manual handle feed interrupt operation.
(6)	BLOCK RESTRT	Selects the operation screen for restarting operation from a block in which the previous operation was interrupted.
(7)	PRGRAM RESTRT	Selects the operation screen for restarting the interrupted program operation.

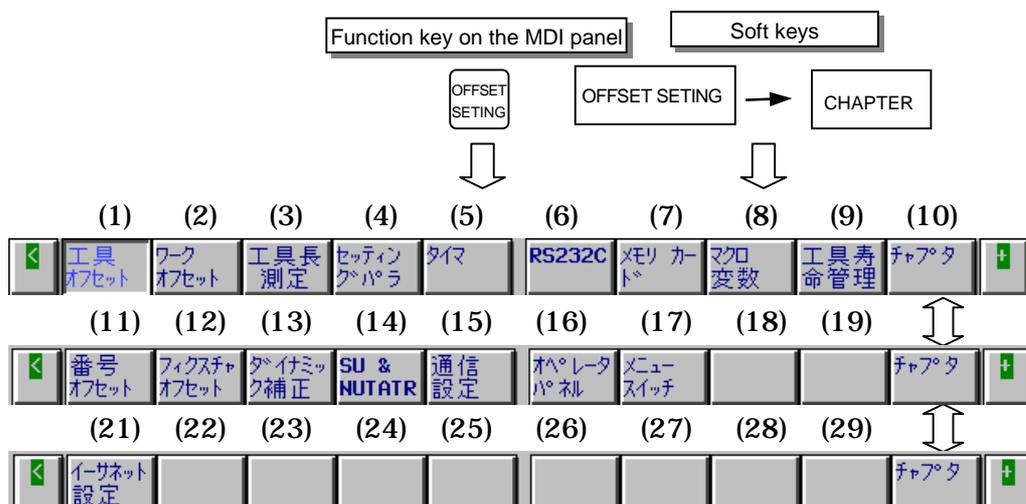
(8)	TOOL HEAD	Displays the absolute coordinates and actual feedrate of the tool tip during handle feed in the longitudinal direction of the tool, handle feed in the transverse direction of the tool, and rotational handle feed around the tool tip.
(9)	HANDLE PULSE	Displays the handle pulse interrupt amount for performing three-dimensional handle feed.

2.5.5.2 Program



No.	Chapter menu	Explanation
(1)	TEXT	Selects the screen for displaying the contents of a currently selected part program.
(2)	DIR. MEMORY	Selects the screen for displaying a list of currently registered part programs.
(4)	HDD DIR.	Selects the screen for displaying a list of files saved on the hard disk of the data server.
(6)	CHECK	Selects the screen for displaying a program, positions, modal information, and so forth at the same time.
(7)	LAST	Selects the screen for displaying the values specified in the block immediately preceding the block currently being executed, and modal values such as G code and F code which were specified in the blocks up to the preceding block.
(8)	ACTIVE	Selects the screen for displaying the values specified in the block currently being executed, and modal values such as G code and F code which were specified in the blocks up to the current block.
(9)	OPE MONTR	Selects the screen for displaying the load (torque) values of the spindle motor and servo motor.
(13)	POS.DATA	Selects the screen for displaying information about position compensation.

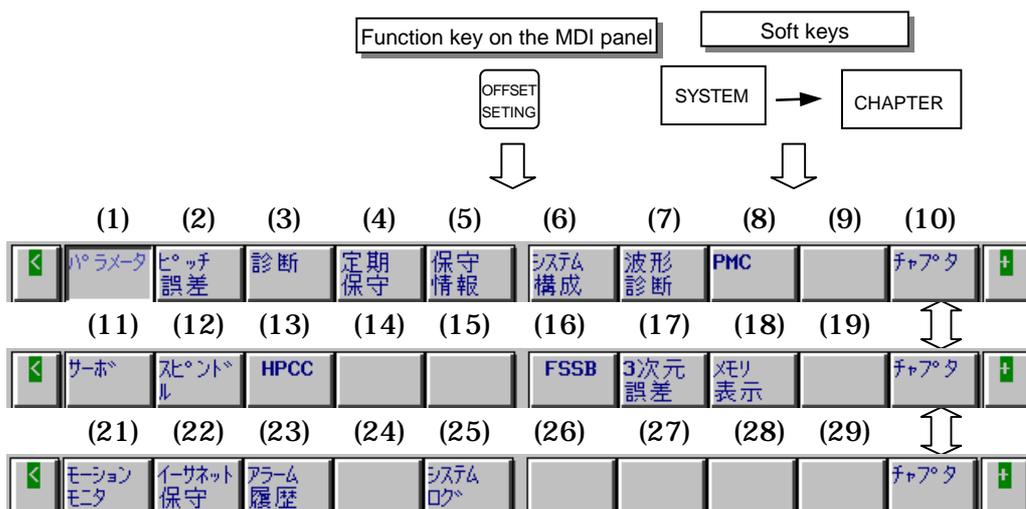
2.5.5.3 Offset/setting



No.	Chapter menu	Explanation
(1)	TOOL	Selects the screen for setting tool offset values.
(2)	WORK OFFSET	Selects the screen for setting workpiece origin offset values. This screen is also used for manual workpiece origin setting and workpiece origin measurement.
(3)	MEASURE_TL	Selects the screen for tool length measurement.
(4)	SETTING PARAMETER	Selects the screen for specifying setting parameters.
(5)	TIMER	Selects the screen for displaying information about the number of machined parts and operation time.
(6)	RS232C	Selects the screen for operating devices connected to the RS-232C interface
(7)	MEMCARD	Selects the screen for memory card operation
(8)	MACRO VAR.	Selects the screen for setting macro variables.
(9)	TOOL LIFE	Selects the screen for making settings related to tool life management.
(11)	T CODE OFFSET	Selects the screen for setting a tool number, pot number, and value for tool offset by tool number.
(12)	FOFS	Selects the fixture offset screen.
(13)	DOFS	Selects the dynamic tool offset screen.
(14)	SU&NUTATR	Selects the SU&NUTATR offset screen.
(15)	COM. SETING	Selects the screen for making RS-232C and RS-422 settings.
(16)	OPERAT PANEL	Selects the screen for using part of the operation switches on the machine operator's panel as soft switches.
(17)	MENU SWITCH	Selects the screen for setting part of the switch signals input from the machine for CNC operation.

No.	Chapter menu	Explanation
(21)	ETHERNET	Selects the screen for making Ethernet board settings.

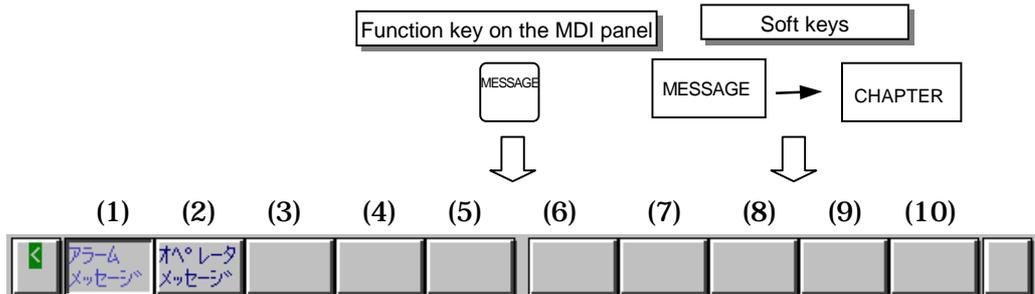
2.5.5.4 System



No.	Chapter menu	Explanation
(1)	PARAMETER	Selects the screen for parameter setting.
(2)	PITCH ERROR	Selects the screen for pitch error compensation setting.
(3)	DIAGNOSIS	Selects the screen for displaying the CNC status information.
(4)	PERIOD MAINTE	Selects the screen for setting the maintenance items to be managed periodically.
(5)	MAINTE INFO	Selects the screen for setting maintenance information.
(6)	SYSTEM CONFIG	Selects the screen for displaying the current system status.
(7)	WAVE DIAGNS	Selects the screen for graphically displaying data such as a servo positional deviation, torque, and machine signals.
(8)	PMC	Selects the screen related to the PMC.
(11)	SERVO	Selects the screen for making servo settings.
(12)	SPINDLE	Selects the screen for making spindle settings.
(13)	HPCC	Selects the screen for setting high-precision contour control.
(16)	FSSB	Selects the screen for setting the high-speed serial servo bus (FSSB: FANUC Serial Servo Bus).
(17)	VOLUMETRIC	Selects the screen for setting three-dimensional error compensation values.
(18)	DISPLY MEMORY	Selects the screen for displaying the current CNC memory contents
(22)	ETHERNET MAINTENANCE	Selects the screen for Ethernet board maintenance and setting.

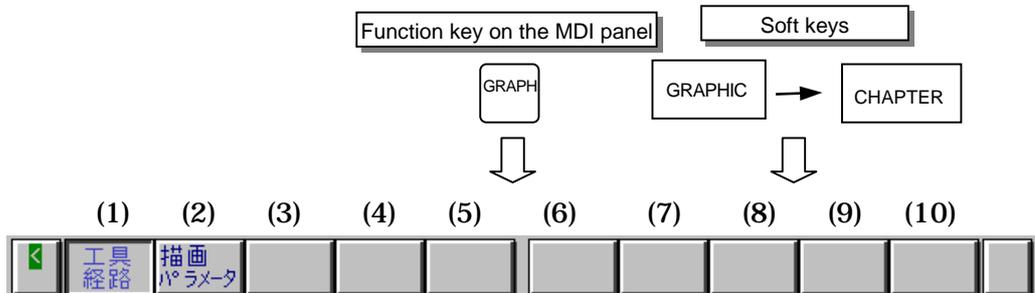
(23)	Alarm History	Selects the screen for displaying the descriptions of alarms issued before
(25)	SYSTEM LOG	Selects the screen for displaying the descriptions of system alarms issued before.

2.5.5.5 Messages



No.	Chapter menu	Explanation
(1)	ALARM	Selects the alarm message screen.
(2)	OPERATOR	Selects the operator message screen.

2.5.5.6 Drawing



No.	Chapter menu	Explanation
(1)	Tool Path	Selects the screen for graphic display of the tool path.
(2)	GRAPH PARAM	Selects the screen for setting graphic display of the tool path.

2.5.6 Key Input and Input Buffer

2.5.6.1 Inserting characters

When an address and a numerical key are pressed, the character corresponding to that key is input once into the key input buffer. The contents of the key input buffer is displayed at the bottom of the screen.

A "_" is displayed at the end of the key input data indicating the input position of the next character.



Fig.2.5.6 (a) Key Input Buffer Display

To input the lower character of the keys that have two

characters inscribed on them, first press the  key and then the key in question.

When the  is pressed, "_" indicating the next

character input position changes to "~". Now lowercase characters can be entered (shift state).

When a character is input in shift status the shift status is

canceled. Furthermore, if  is pressed in shift

status, the shift status is canceled.

Example

Inserting G90 after N001 when "N001X100.OY200.OF100.0;_" is displayed as the contents of the key input buffer:

- Press cursor key . to move the cursor to the following position:
N001_X100.OY200.OF100.0;
- Key in G90.
N001G90_X100.OY200.OF100.0;
- Press cursor key . to move the cursor to the following position:
N001G90X100.OY200.OF100.0;_

2.5.6.2 Deleting characters

Move the cursor to a position next to a character you want to

delete, then press the



When the



is pressed while the shift button is held

down, the contents of the key input buffer are all deleted.

Example

Deleting Y200.0 when "N001X100.0Y200.0F100.0;" is displayed as the contents of the key input buffer:

- Press cursor key < > to move the cursor to the following position:
N001X100.0Y200.0_F100.0;
- Press the <CAN> key six times.
N001X100.0_F100.0;
- Press the cursor key < > to move the cursor to the following position:
N001X100.0F100.0;_

2.5.7 Warning Messages

After a character or number has been input from the MDI

panel, a data check is executed when key  soft key is pressed.

In the case of incorrect input data or the wrong operation a flashing warning message will be displayed on the status display line.



Fig. 8.5.7(a) Warning message display

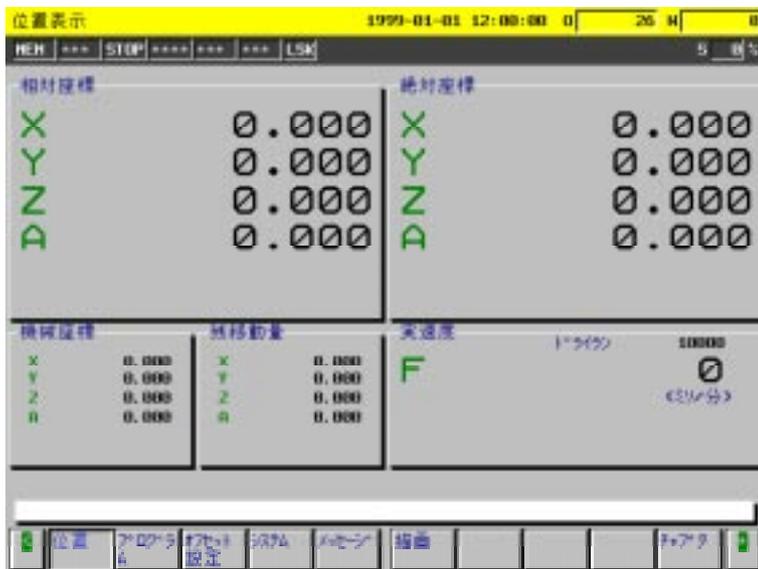
Table 8.5.7(a) Warning Messages

Warning message	Content
FORMAT ERROR	The format is incorrect.
WRITE PROTECT	Key input is invalid because of data protect key or the parameter is not write enabled.
DATA IS OUT OF RANGE	The input value exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.
WRONG MODE	An appropriate mode is not selected.
EDIT REJECTED	It is not possible to edit in the current CNC status.

2.6

\ f Ⓞ •

- ...
- ..
-
-
-
-
-
-



2.6 (a) * Ⓞ F Ⓞ { Ⓞ



2.6 (b) * Ⓞ F p Ⓞ

2.7 EXTERNAL I/O DEVICES

Four types of external input/output devices are available. This section outlines each device. For details on these devices, refer to the corresponding manuals listed below.

Table 2.4(a) External I/O device

Device name	Usage	Max. Storage capacity	Reference manual
FANUC Handy File	Easy-to-use, multi function input/output device. It is designed for FA equipment and uses floppy disks.	3600m	B-61834E
FANUC Floppy Cassette	Input/output device. Uses floppy disks.	2500m	B-66040E
FANUC FA Card	Compact input/output device. Uses FA cards.	160m	B-61274E
FANUC PPR	Input/output device consisting of a paper tape reader, tape punch, and printer.	275m	B-58584E

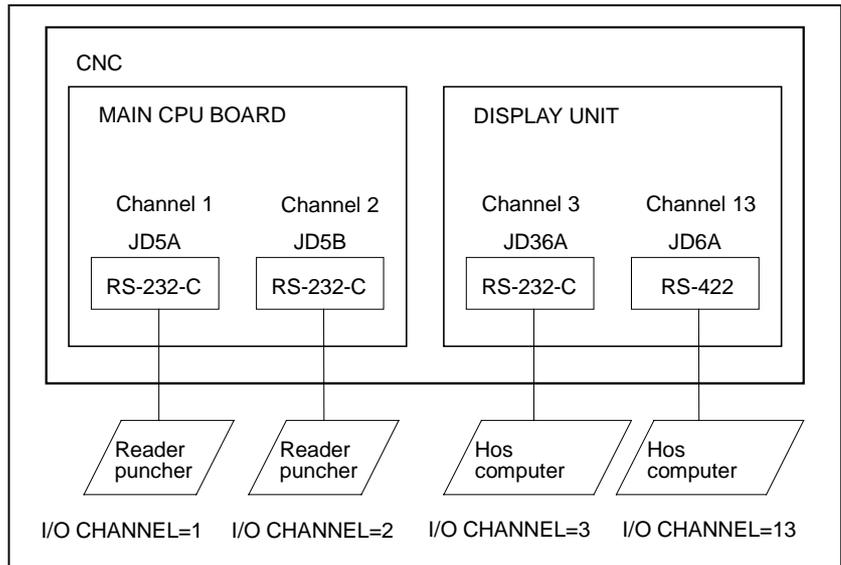
The following data can be input/output to or from external input/output devices:

1. Programs
2. Offset data
3. Parameters
4. Custom macro common variables

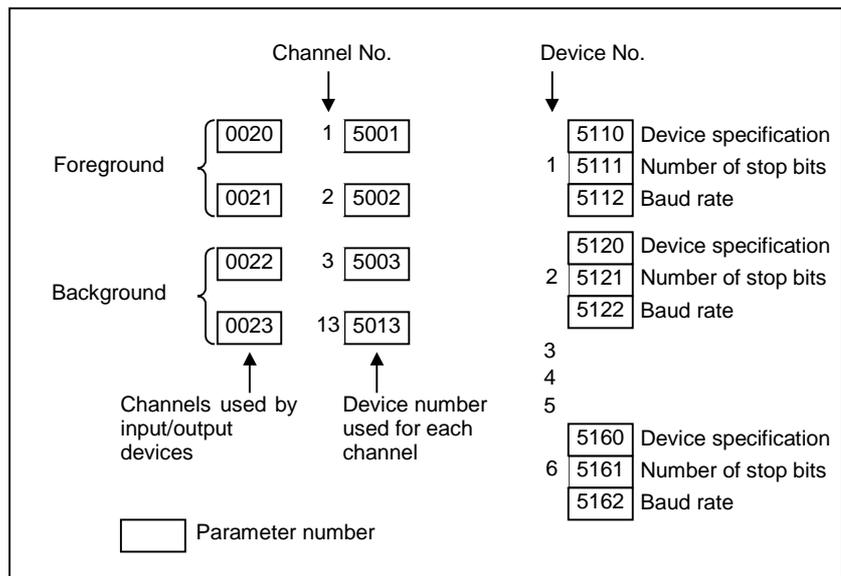
For how data is input and output, see III-8.

Parameter

Before an external input/output device can be used, parameters must be set as follows.



Each external input/output device is temporarily assigned a device number from 1 to 6. When four external input/output devices are used, for example, a device number from 1 to 4 is assigned to each device. After the device numbers are assigned to the external input/output devices, their specifications are set in parameter Nos. 5110 to 5162. The numbers of the external input/output devices attached to channels 1, 2, 3, and 13 are set in parameter Nos. 5001, 5002, 5003, and 5013.

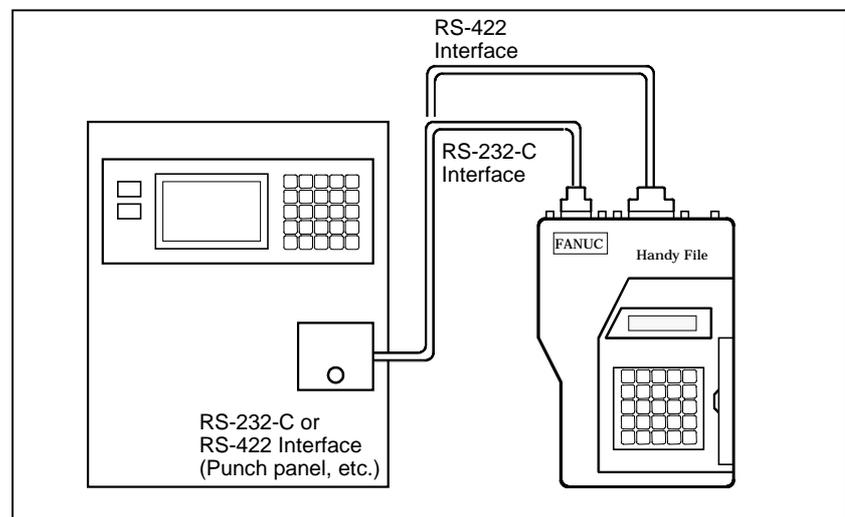


2.7.1 FANUC Handy File

The Handy File is an easy-to-use, multi function floppy disk input/output device designed for FA equipment. By operating the Handy File directly or remotely from a unit connected to the Handy File, programs can be transferred and edited.

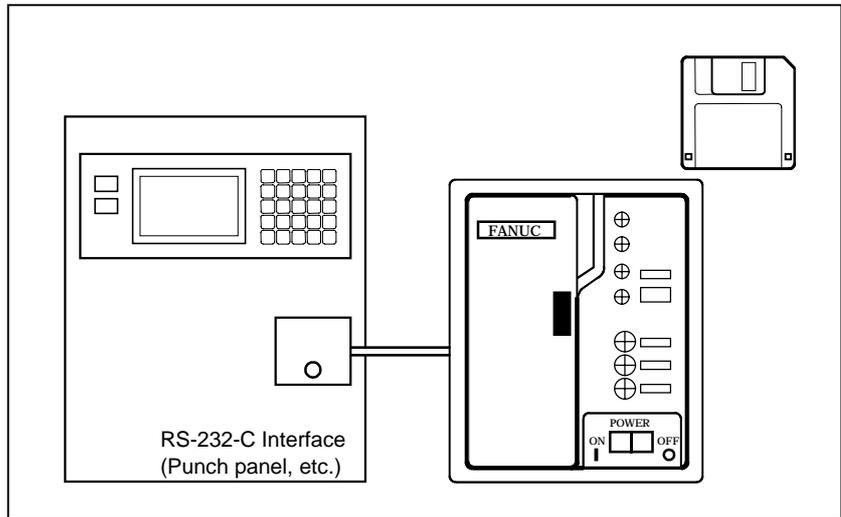
The Handy File uses 3.5-inch floppy disks, which do not have the problems of paper tape (i.e., noisy during input/output, easily broken, and bulky).

One or more programs (up to 1.44M bytes, which is equivalent to the memory capacity of 3600-m paper tape) can be stored on one floppy disk.



2.7.2 FANUC Floppy Cassette

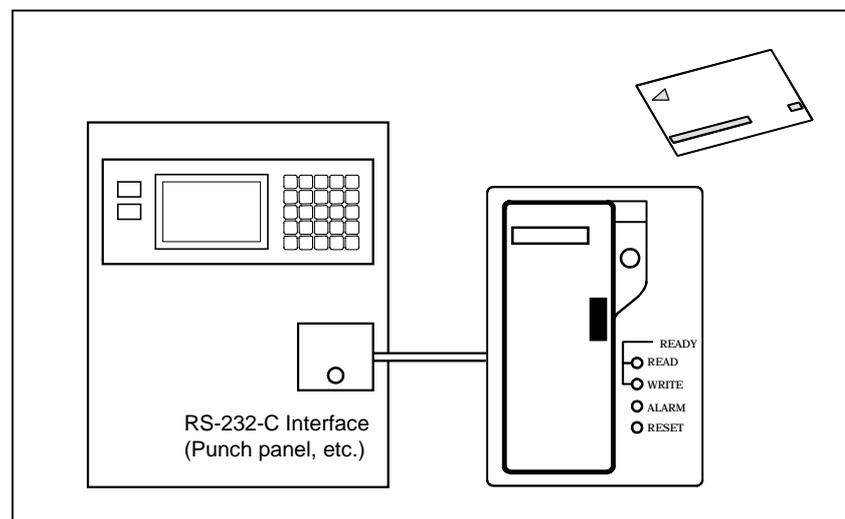
When the Floppy Cassette is connected to the CNC, machining programs stored in the CNC can be saved on a Floppy Cassette, and machining programs saved in the Floppy Cassette can be transferred to the CNC.



2.7.3 FANUC FA Card

An FA Card is a memory card used as an input medium in the FA field. It is compact, but has a large memory capacity with high reliability, and requires no special maintenance.

When an FA Card is connected to the CNC via the card adapter, machining programs stored in the CNC can be transferred to and saved in an FA Card. Machining programs stored on an FA Card can also be transferred to the CNC.



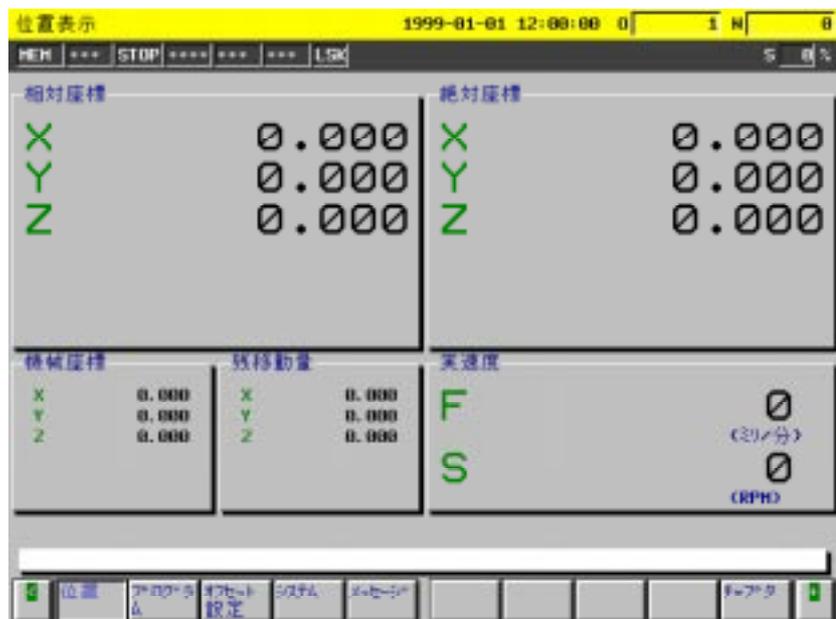
2.8 POWER ON/OFF

2.8.1 Turning on the Power

Procedure of turning on the power

Procedure

- 1 Check that the appearance of the CNC machine tool is normal. (For example, check that front door and rear door are closed.)
- 2 Turn on the power according to the manual issued by the machine tool builder.
- 3 After the power is turned on, check that the position screen is displayed. An alarm screen is displayed if an alarm occurs upon power-on. If the screen shown in Section III-2.5.2 is displayed, a system failure may have occurred.



Screen & Position display

- 4 Check that the fan motor is rotating.

WARNING

Until the positional or alarm screen is displayed at the power on, do not touch them. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

2.8.2 Power Disconnection

Power Disconnection

Procedure

- 1 Check that the LED indicating the cycle start is off on the operator's panel.
- 2 Check that all movable parts of the CNC machine tool is stopping.
- 3 If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.
- 4 Continue to press the POWER OFF pushbutton for about 5 seconds.
- 5 Refer to the machine tool builder's manual for turning off the power to the machine.

NOTE

In the cable connecting the input/output unit and RS-232-C interface, the SG signal and each signal must be paired as shown in the figure below.

RS-232-

C C ^ t F [X - † G Ø o ~ @ P [u E • Ø ~

3

MANUAL OPERATION

MANUAL OPERATION are nine kinds as follows :

3.1 Manual Reference Position Return

3.2 Jog Feed

3.3 Incremental Feed

3.4 Manual Handle Feed

3.5 Manual Absolute ON and OFF

3.6 Tool Axis Direction Handle Feed

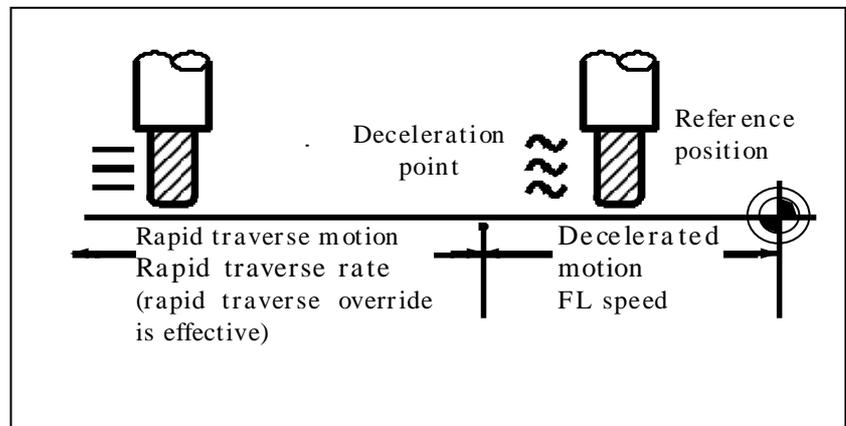
3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :

The tool is moved in the direction specified in parameter ZMI (bit 5 of No. 1006) for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 1420,1421, and 1425).

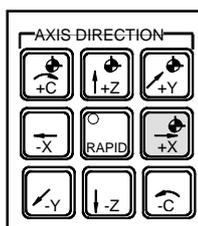
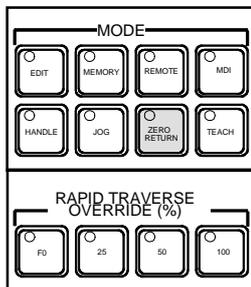
Rapid traverse override is effective during rapid traverse.

When the tool has returned to the reference position, the reference position return completion LED goes on.

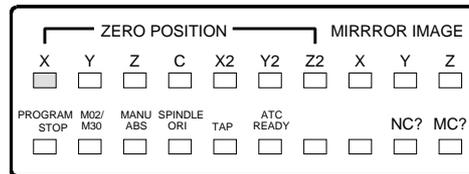


Procedure for Manual Reference Position Return

Procedure



- 1 Press the reference position return switch, one of the mode selection switches.
- 2 To decrease the feedrate, press a rapid traverse override switch. When the tool has returned to the reference position, the reference position return completion LED goes on.
- 3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter.
- 4 Perform the same operations for other axes, if necessary. The above is an example. Refer to the appropriate manual provided by the machinetool builder for the actual operations.



Restrictions

£ Moving the tool again

Once the REFERENCE POSITION RETURN COMPLETION LED lights at the completion of reference position return, the tool does not move unless the REFERENCE POSITION RETURN switch is turned off.

£ Reference position return completion LED

The REFERENCE POSITION RETURN COMPLETION LED is extinguished by either of the following operations:

- Moving from the reference position.
- Entering an emergency stop state.

£ The distance to return to reference position

For the distance (Not in the deceleration condition) to return the tool to the reference position, refer to the manual issued by the machine tool builder.

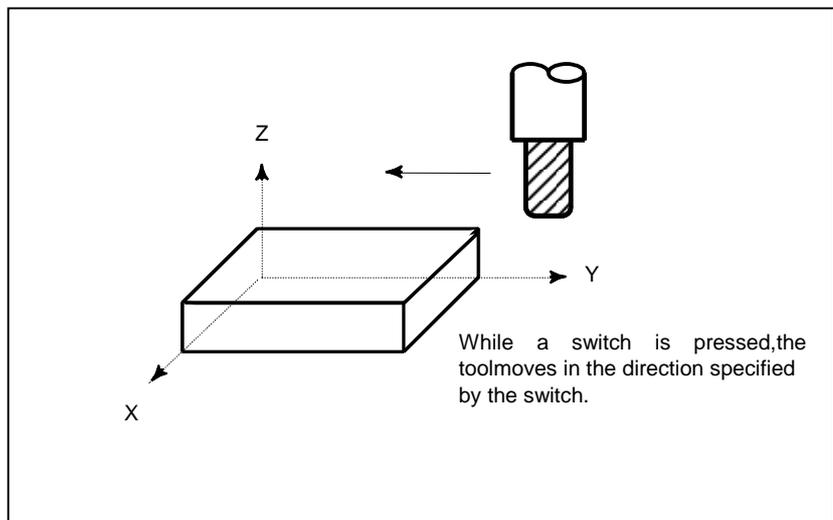
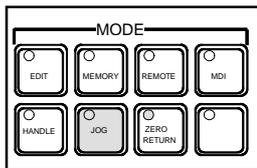
3.2 JOG FEED

In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.

The jog feedrate is specified in a parameter (No.1423)

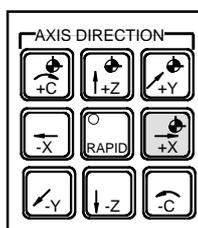
The jog feedrate can be adjusted with the jog feedrate override dial.

Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate (No. 1420) regardless of the position of the jog feedrate override dial. This function is called the manual rapid traverse.



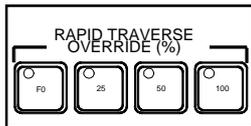
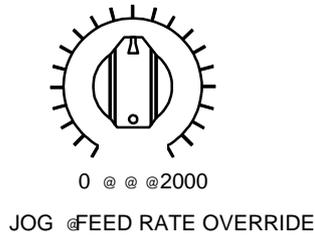
Procedure for JOG feed

Procedure



- 1 Press the jog switch, one of the mode selection switches.
- 2 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in a parameter (No. 1423). The tool stops when the switch is released.
- 3 The jog feedrate can be adjusted with the jog feedrate override dial.
- 4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate while the rapid traverse switch is pressed. Rapid traverse override by the rapid traverse override switches is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.



Limitations

- Acceleration/deceleration for rapid traverse

Feedrate, time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.

- Change of modes

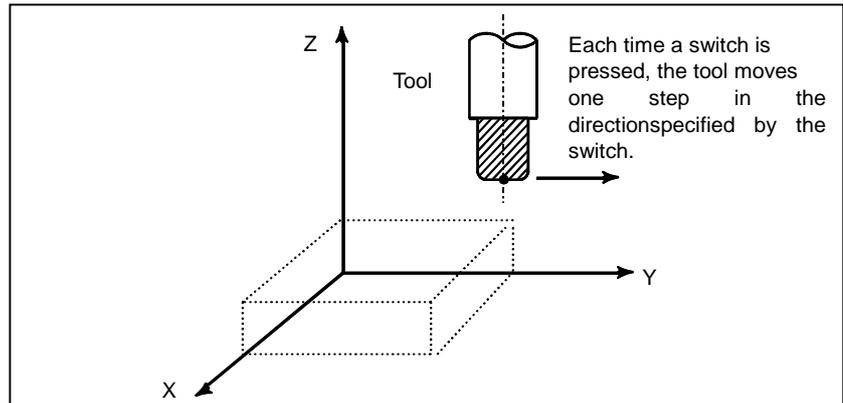
Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

- Rapid traverse before reference position return

When the stored stroke limit option is supported, if reference position return is not performed after the power is turned ON, pressing the RAPID TRAVERSE button does not actuate rapid traverse on the axis to which the reference position return function is programmed; the feedrate remains at the jog feedrate. As the stored stroke limit is effective until reference position return is performed, axis movement at the rapid traverse rate up to the stroke end is prevented. This function can be disabled by setting bit 2 (RPD) of parameter No.1400.

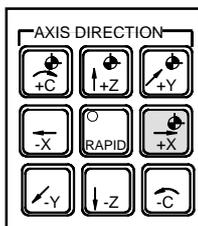
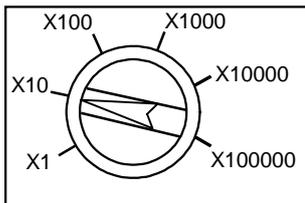
3.3 INCREMENTAL FEED

In the incremental mode(s), pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction.



Procedure for Incremental Feed

Procedure



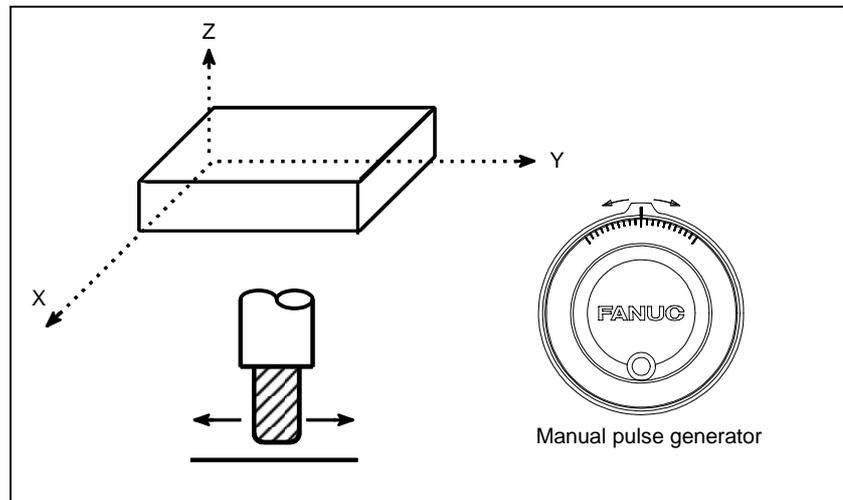
- 1 Press the INC switch, one of the mode selection switches.
- 2 Select the distance to be moved for each step with the magnification dial.
- 3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.
- 4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate. Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

3.4 Manual Handle Feed

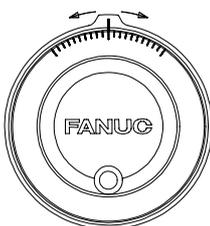
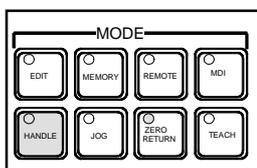
In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.

The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. Or the distance the tool is moved when the manual pulse generator is rotated by one graduation can be magnified by 10 times or by one of the two magnifications specified by parameters (No. 1414 and 1418).



Procedure for Manual Handle Feed

Procedure



Manual pulse generator

- 1 Press the HANDLE switch, one of the mode selection switches.
- 2 Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.
- 3 Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.
- 4 Move the tool along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduations.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- When a manual handle feed faster than the rapid traverse rate is specified

When a handle feed faster than the rapid traverse rate is specified, the command pulses up to the permissible flow amount set in parameter No. 1413 are accumulated in the CNC.

- Distance the tool is moved by manual handle feed

The minimum distance the tool is moved when the manual pulse generator is rotated by one gradation is equal to the least input increment. The magnification for the distance the tool is to be moved can be selected from 10 times and the two magnifications specified by parameter Nos. 1414 and 1418.

Restrictions

- Number of MPGs

Up to three manual pulse generators can be connected, one for each axis. The three manual pulse generators can be simultaneously operated.

WARNING

Rotating the handle quickly with a large magnification such as x100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

NOTE

Rotate the manual pulse generator at a rate of five rotations per second or lower. If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after the handle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.

3.5 MANUAL FEED IN A SPECIFIED DIRECTION

The tool can be manually moved in a specified direction on a specified plane. The feedrate and the direction and plane of feed can be changed at any time. Simple cutting can be manually executed on a plane.

Explanation

£ Selecting a plane

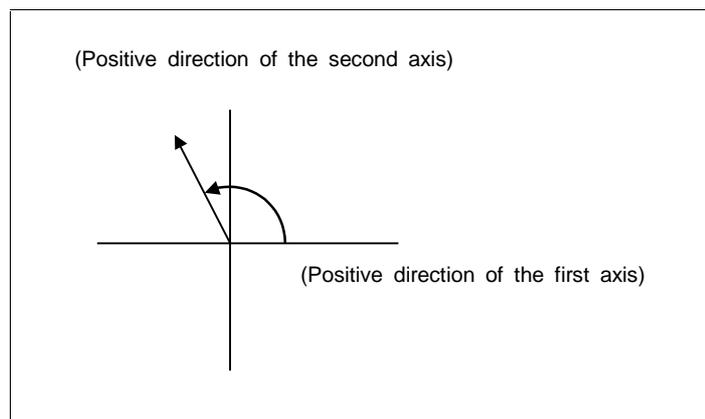
Specify external signals that indicate the first and second axes of the plane on which the tool is manually moved in a specified direction.

£ Specifying the direction of feed

Enter an external input signal that indicates the direction in which the tool is to be manually moved. The direction of feed can be specified in the range of 0 to 360 degrees in 1/16 degrees. The angle is specified as shown below.

If an angle greater than 360 degrees is specified, the CNC unit repeatedly subtracts 360 from the angle until it falls within the range of 0 to 360 degrees.

While the tool is being manually moved in the specified direction, the direction of feed can be changed as required. When the direction is changed, the position check is not executed, but the move command in the new direction is immediately executed.



£ Specifying a feedrate

Using the dial for specifying the jog feedrate, specify the feedrate (as a tangential velocity) at which the tool is manually moved in the specified direction.

Parameter No. 1411 determines the feedrate when the dial is set to the 100% position.

£ Starting and stopping manual feed in the specified direction

Manual feed in the specified direction is executed while the corresponding signals are “1”.

The signal that moves the tool in the specified direction and the signal that moves it in the opposite direction are used in manual feed.

NOTE

- 1 When the mode of manual feed in the specified direction is selected, the tool can be moved only on the selected plane determined by the selected axes.
- 2 Automatic or manual operation cannot be switched when manual feed in the specified direction is executed.

3.6 MANUAL NUMERIC COMMAND

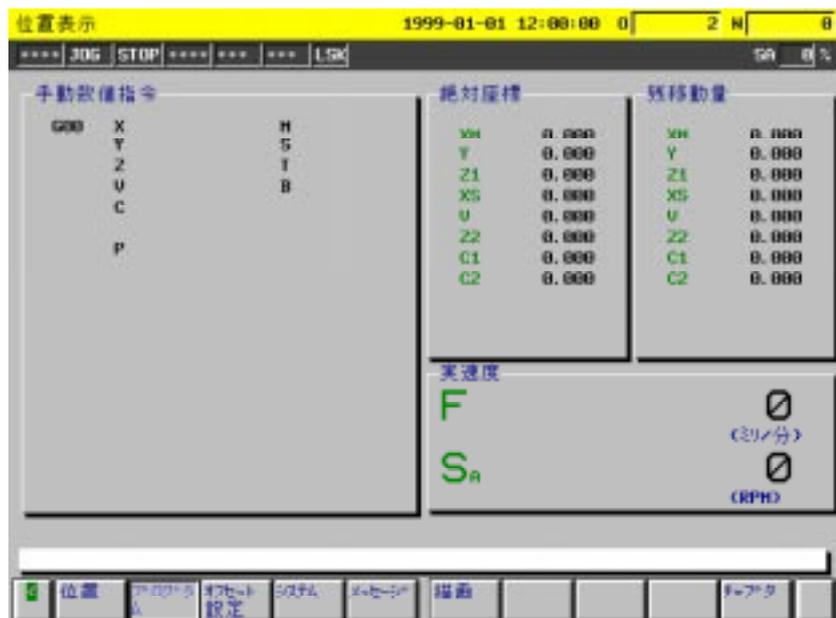
The manual numeric command function allows data programmed by MDI to be executed in the jog mode. Whenever the system is ready for jog feed, a manual numeric command can be executed. The following eight functions are supported:

- (1) Positioning (G00)
- (2) Linear interpolation (G01)
- (3) Automatic reference position return (G28)
- (4) 2nd, 3rd or 4th reference position return (G30)
- (5) M codes (miscellaneous functions)
- (6) S codes (spindle functions)
- (7) T codes (tool functions)
- (8) B codes (second auxiliary function)

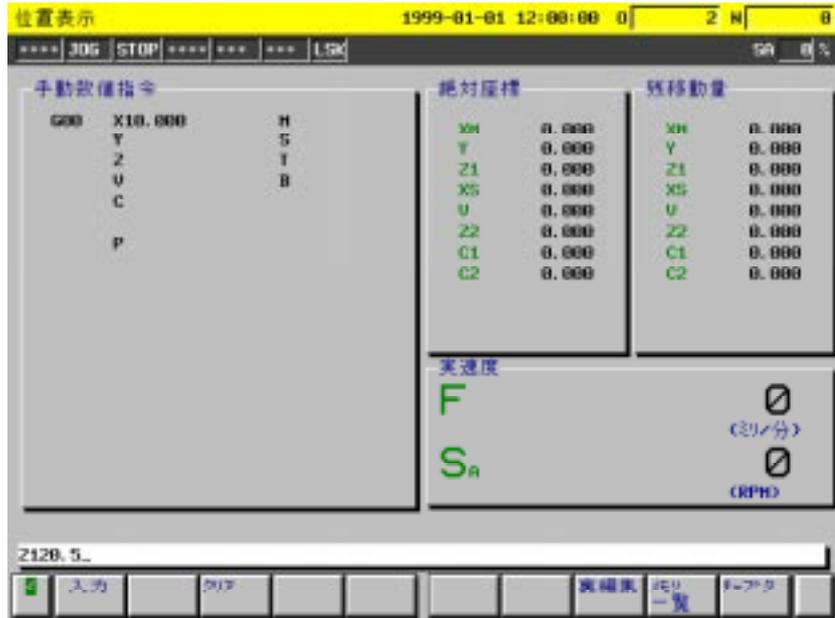
Procedure Manual numeric command

Procedure

- 1 Follow the procedure for displaying the manual numeric command screen to display the manual numeric command screen as follows:



- 2 Press the operation menu key to display the soft keys in the operation selection menu.
- 3 Key in the required command using the address keys and numeric keys on the MDI panel, and press either soft key [ENTER] or [INPUT] to set the data.



The following data can be set:

1. G00: Positioning
2. G01: Linear interpolation
3. G28: Automatic reference point return
4. G30: 2nd, 3rd or 4th reference position return
5. M codes: Miscellaneous functions
6. S codes: Spindle functions
7. T codes: Tool functions
8. B codes: Second auxiliary functions

The set data is held in memory even if the screen or mode is changed.

NOTE

Data cannot be set when an alarm occurs.

- 4 Press the Cycle Start button on the machine operator's panel to start command execution. The status is indicated as "MSTR." The automatic operation signal STL can be set to "1" (ON) by setting bit 2 (STJ) of parameter No. 7001.



When execution is completed, the "MSTR" status indication is cleared from the screen, and automatic signal operation STL is turned OFF. All set data is cleared. G codes are set to G00 or G01 according to the setting of bit 0 (G01) of parameter No. 2401.

NOTE
 If the Cycle Start button is pressed while an alarm has occurred, a "START IMPOSSIBLE" warning is generated, and the entered data cannot be executed.

Explanation
 £ Positioning

The amount of travel is given as a numeric value, preceded by an address such as X, Y or Z. This is always regarded as being an incremental command, regardless of whether G90 or G91 is specified. The tool moves along each axis independently at the rapid traverse rate. Linear interpolation type positioning (when the tool path is linear) can also be performed by setting bit 4 (LRP) of parameter No. 1400.

	Manual rapid traverse selection switch	
	OFF	ON
Feedrate (parameter)	Jog feedrate for each axis (No. 1423)	Rapid traverse for each axis (No. 1420)
Automatic acceleration/deceleration (parameter)	Exponential acceleration/ deceleration in jog feed for each axis (No. 1624)	Linear acceleration/ deceleration in rapid traverse for each axis (No. 1620)
Override	Manual feed override	Rapid traverse override

£ Linear interpolation (G01)

The amount of travel is given as a numeric value, preceded by an address such as X, Y or Z. This is always regarded as being an incremental command, regardless of whether G90 or G91 is specified. Axial movements are always performed in the incremental mode even during scaling or polar coordinate interpolation. In addition, movement is always performed in feed per minute mode, regardless of whether G94 or G95 is specified.

Feedrate (parameter)	Dry run speed (No. 1410)
Automatic acceleration/ deceleration (parameter)	Exponential acceleration/ deceleration in cutting feed for each axis (No. 1622)
Override	Manual feed override

NOTE
 Since the feedrate is always set to the dry run feedrate, regardless of the setting of the dry run switch, the feedrate cannot be specified using F. The feedrate is clamped so that the maximum cutting feedrate, set in parameter No. 1422, is not exceeded.

£ Automatic reference position return (G28)

The tool returns directly to the reference position without passing through any intermediate points, regardless of the specified amount of travel. For axes for which no move command is specified, however, a return operation is not performed.

Feedrate (parameter)	Rapid traverse rate (No. 1420)
Automatic acceleration/ deceleration (parameter)	Linear acceleration/ deceleration in rapid traverse for each axis (No. 1620)
Override	Rapid traverse override

£ 2nd, 3rd or 4th reference position return (G30)

The tool returns directly to the 2nd, 3rd or 4th reference position without passing through any intermediate points, regardless of the specified amount of travel. To select a reference point, specify P2, P3 or P4 in address P. If address P is omitted, a return to the 2nd reference position is performed.

Feedrate (parameter)	Rapid traverse rate (No. 1420)
Automatic acceleration/ deceleration (parameter)	Linear acceleration/ deceleration in rapid traverse for each axis (No. 1620)
Override	Rapid traverse override

NOTE

The 2nd, 3rd or 4th reference position return function is optional. If this option is not selected, the warning "FORMAT ERROR" is generated, and G30 cannot be entered. If neither of P2, P3 or P4 are specified in address P when this option is selected, a "START IMPOSSIBLE" warning is generated, and the entered data cannot be executed.

£ M codes (miscellaneous functions)

After address M, specify a numeric value of no more than the number of digits specified by parameter No. 2030.

NOTE

Neither subprogram calls nor custom macro calls can be performed using M codes.

£ S codes (spindle functions)

After address S, specify a numeric value of no more than the number of digits specified by parameter No. 2031.

NOTE

Subprogram calls cannot be performed using S codes.

£ T codes (tool functions)

After address T, specify a numeric value of no more than the number of digits specified by parameter No. 2032.

NOTE

Subprogram calls cannot be performed using S codes.

£ B codes (second auxiliary functions)

After address B, specify a numeric value of no more than the number of digits specified by parameter No. 2033.

NOTE

Subprogram calls cannot be performed using B codes.

£ Data input

- (1) When addresses and numeric values of a command are types, then soft key [INPUT] is pressed, the entered data is set. In this case, the input key is either least input increment or calculator-type input format according to the setting of bit 0 (DPI) of parameter No. 2400.
The [INPUT] key on the MDI panel can be used instead of soft key [INPUT].
- (2) Commands for multiple addresses can be types successively.
- (3) Key entry is disabled during execution. If soft key [INPUT] or the [INPUT] key on the MDI panel is pressed during execution, an "EXECUTION/MODE SWITCHING IN PROGRESS" warning is generated.
- (4) If input data contains an error, the following warnings may be generated:

Warning	Description
---------	-------------

FORMAT ERROR	<ul style="list-style-type: none">-G code other than G00, G01, G28 and G30 has been entered.-An address other than those displayed on the manual numeric command screen has been entered. TOO MANY DIGITS A value that exceeds the following limitations has been entered: <ul style="list-style-type: none">-Address G: 2 digits-Address P: 1 digit-Axis address: 9 digits-M, S, T, B: The parameter-set number of digits
--------------	---

NOTE

Even when the memory protection key is set, key input can be performed nevertheless.

£ Erasing data

- (1) When soft key [CLEAR] is pressed, followed by soft key [EXEC], all set data is cleared. In this case, however, the G codes are set to G00 or G01 according to the setting of bit 0 (G01) of parameter No. 2401.
Data can also be cleared by pressing the [RESET] key on the MDI panel.
- (2) If soft key [CLEAR] is pressed during execution, an "EXECUTION/MODE SWITCHING IN PROGRESS" warning is generated.

£ Halting execution

If one of the following occurs during execution, execution is halted, and data is cleared in the same way as when soft key [CLEAR] is pressed. The remaining amount of travel is canceled.

- (1) When a feed hold is applied
- (2) When the mode is changed to other than the jog feed mode
- (3) When an alarm is generated
- (4) When a reset or emergency stop is applied

The M, S, T and B functions remain effective even upon the occurrence of the above events, with the exception of (4).

£ Modal information

Model G codes and addresses used in automatic operation are not affected by the execution of commands specified using the manual numeric command function.

£ Jog feed

When the tool is moved along an axis using a feed axis and direction selection switch on the manual numeric command screen, the remaining amount of travel is always shown as "0".

Limitation**£ Constant surface speed control**

S codes cannot be specified in the constant surface speed control mode.

£ M, S, T, B functions

Manual numeric commands can be executed while automatic operation is halted. In the following cases, however, a "START IMPOSSIBLE" warning is output, and command execution is disabled.

- (1) When an M, S, T or B function is already being executed, a manual numeric command containing an M, S, T or B function cannot be executed.
- (2) When an M, S, T or B function is already being executed, and that function alone is specified or a block specifying that function also contains another function (such as a move command or dwell function) which has already been executed, a manual numeric command cannot be executed.

£ Jog feed

When a manual numeric command is specified while the tool is being moved along an axis by using a feed axis and direction selection switch, the axial movement is interrupted, and the manual numeric command is executed. For this reason, the tool cannot be moved along an axis by using a feed axis and direction selection switch during execution of a manual numeric command.

£ Mirror image

Mirror image cannot be applied in the direction of the specified axis movement.

£ Indexing of the index table and chopping

Commands cannot be specified for an axis along which operation is being performed during indexing or chopping.

If such an axis is specified for execution, a "START IMPOSSIBLE" warning is generated.

3.7 MANUAL ABSOLUTE ON AND OFF

Whether the distance the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operator's panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is not added to the coordinates.

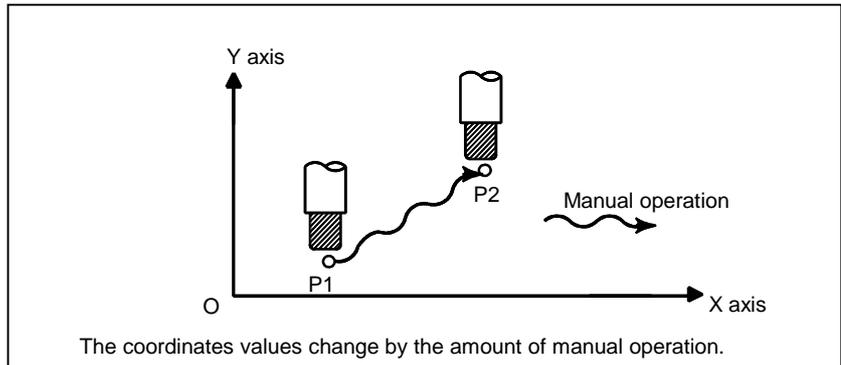


Fig.3.7 (c) Coordinates with the switch ON

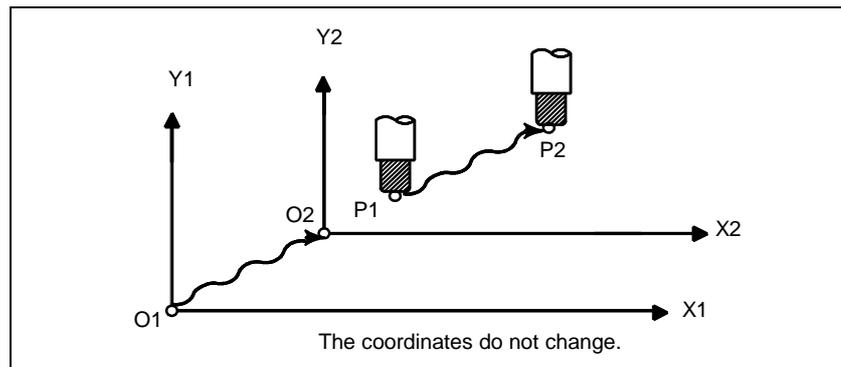


Fig.3.7 (d) Coordinates with the switch OFF

Explanation

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

```
G01G90          X100.0Y100.0F100  ;
                X200.0Y150.0    ;
```

```
X300.0Y200.0 ; .
```

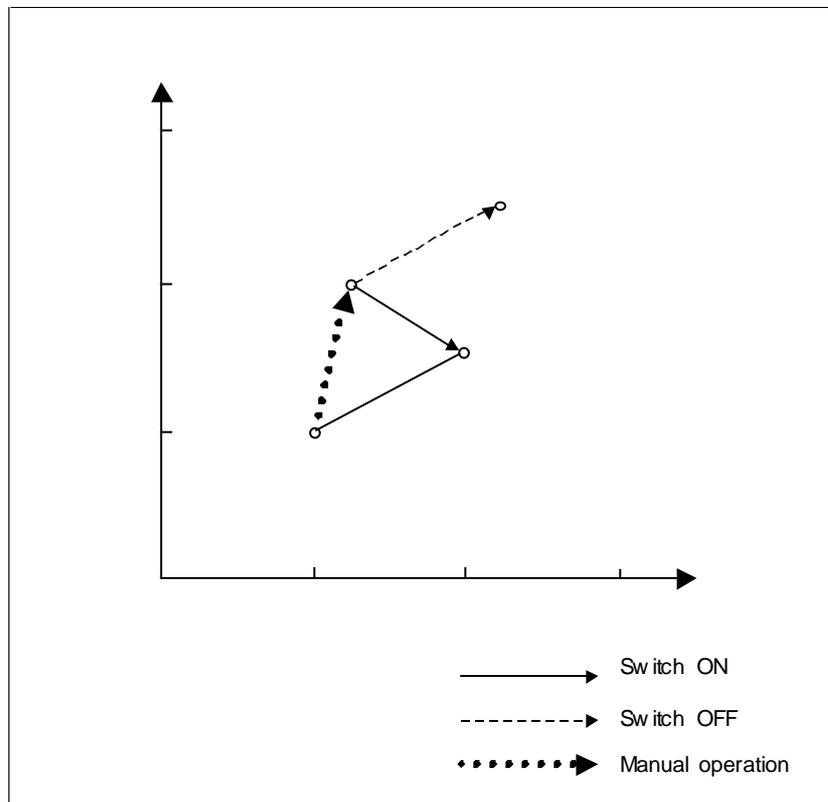
The subsequent figures use the following notation:

- > Movement of the tool when the switch is on
- - - - -> Movement of the tool when the switch is off

The coordinates after manual operation include the distance the tool is moved by the manual operation. When the switch is off, therefore, subtract the distance the tool is moved by the manual operation.

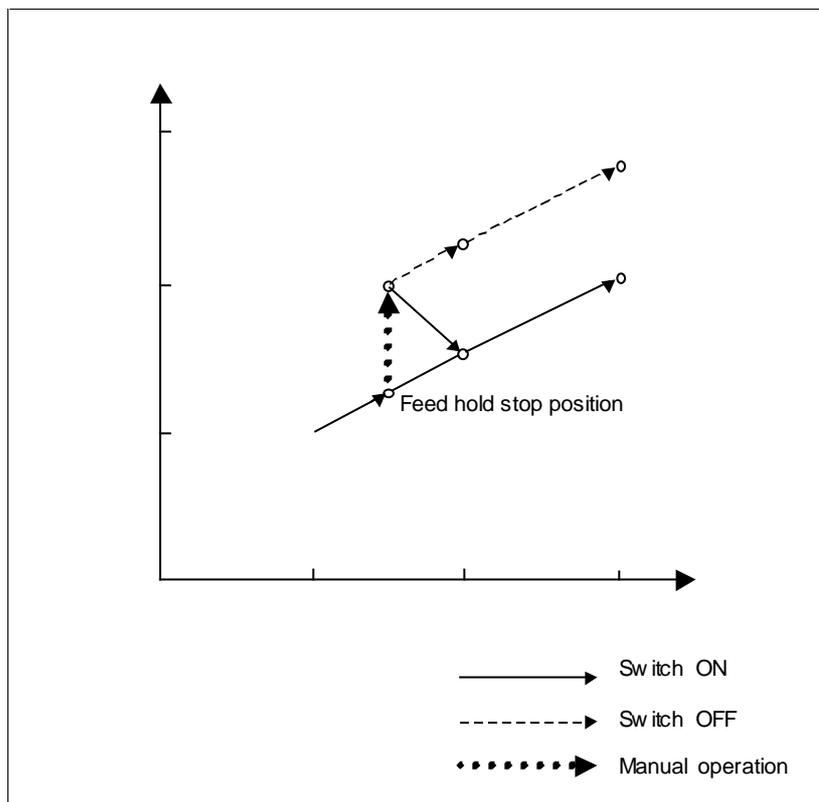
£ Manual operation after the end of block

Coordinates when block . has been executed after manual operation (X-axis +20.0, Y-axis +100.0) at the end of movement of block.



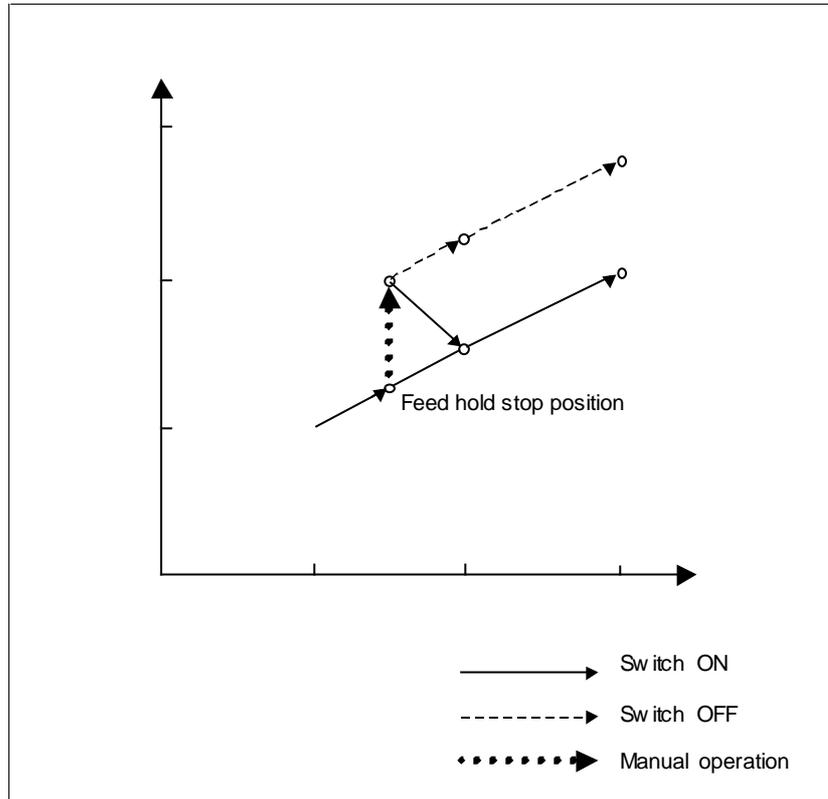
£ Manual operation after a feed hold

Coordinates when the feed hold button is pressed while block . is being executed, manual operation (Y-axis + 75.0) is performed, and the cycle start button is pressed and released.



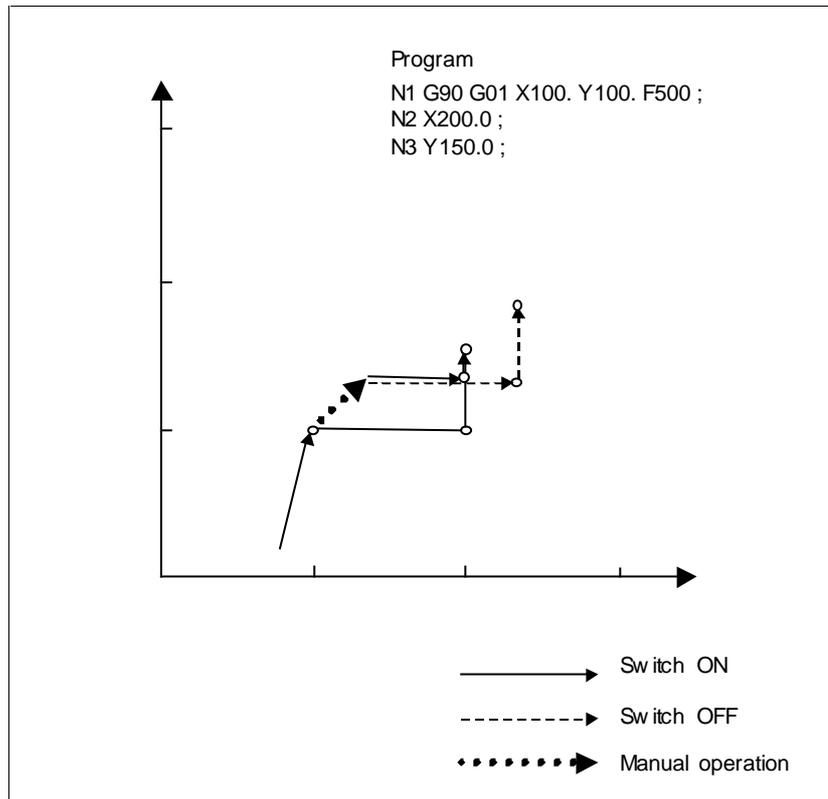
£ **When reset after a manual operation following a feed hold**

Coordinates when the feed hold button is pressed while block . is being executed, manual operation (Y-axis +75.0) is performed, the control unit is reset with the RESET button, and block . is read again



£ When a movement command in the next block is only one axis

When there is only one axis in the following command, only the commanded axis returns.



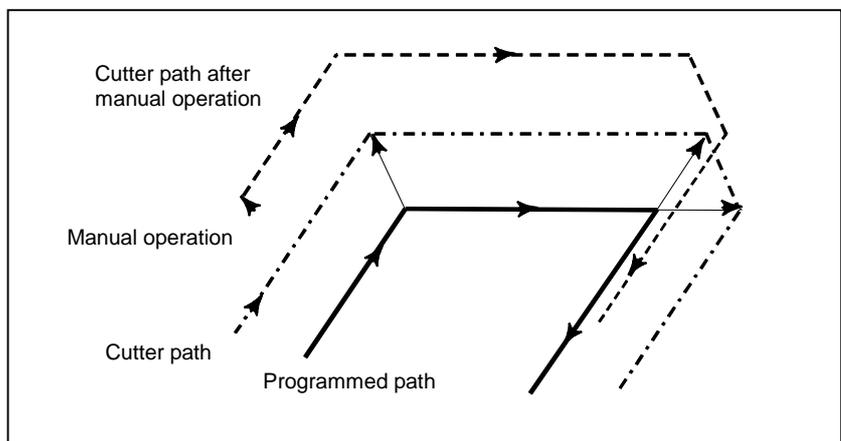
£ **When the next move block is an incremental**

When the following commands are incremental commands, operation is the same as when the switch is OFF.

£ **Manual operation during cutter compensation**

When the switch is OFF

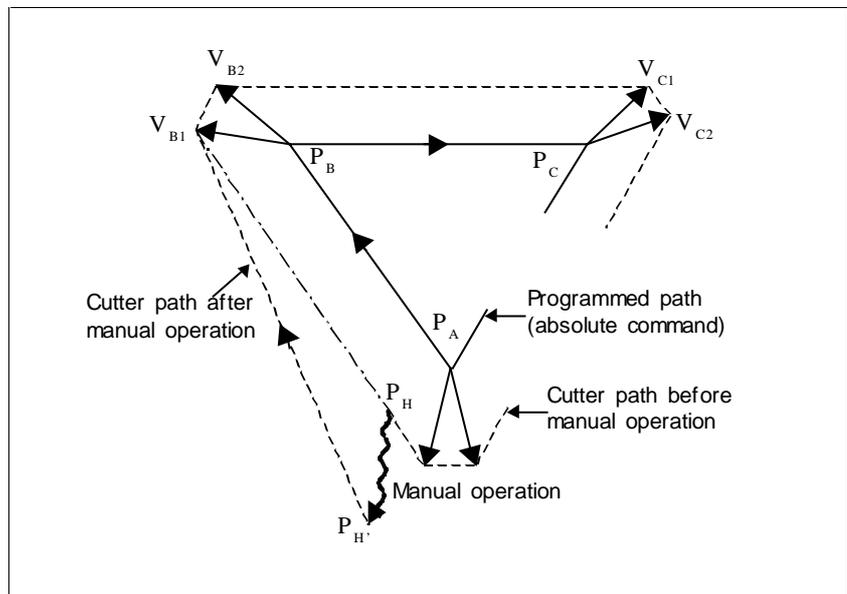
After manual operation is performed with the switch OFF during cutter compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed. The amount of separation equals to the amount that was performed manually.



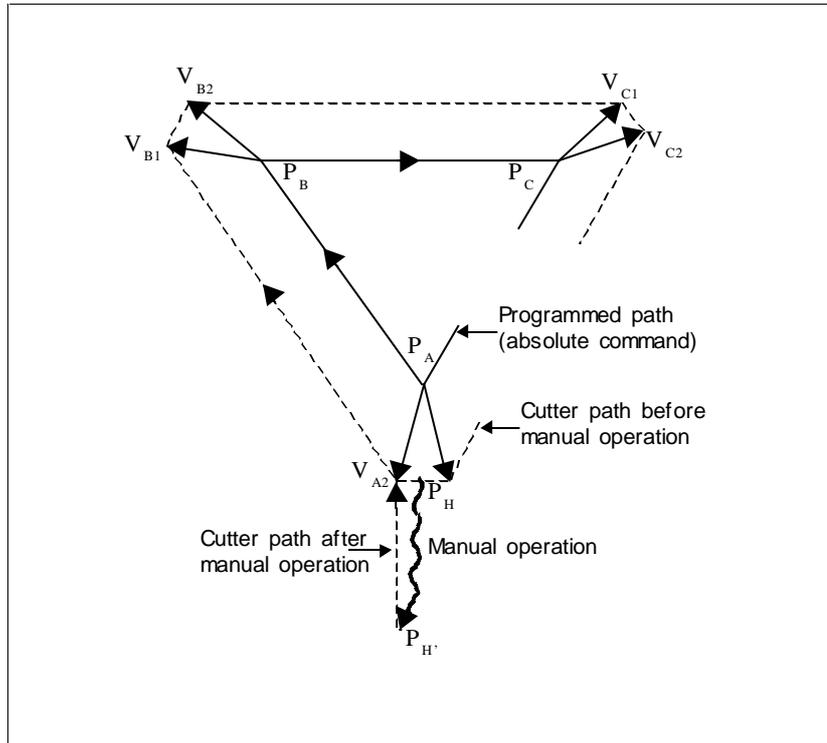
When the switch is ON during cutter compensation

Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the cutter compensation mode will be described.

After manual intervention, the tool moves so that the tool center moves towards the end of the next vector in the program.

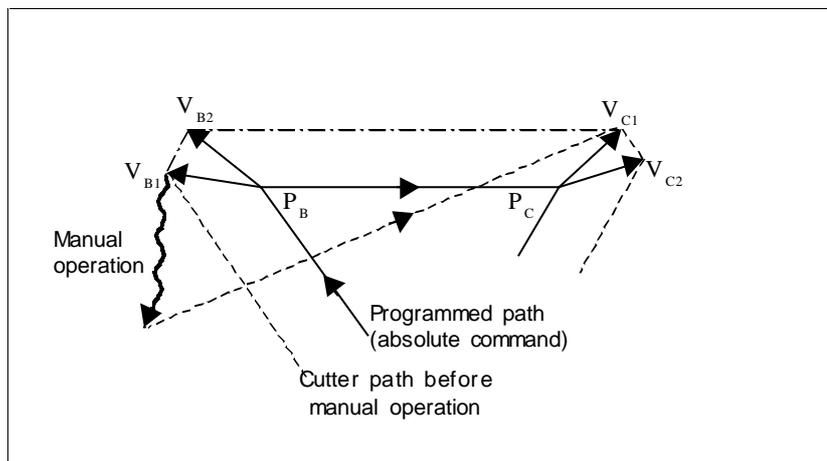


Also, after manual operation is performed during corner movement in cutter compensation mode, the tool moves so that the tool center moves towards the end of the next vector in the program.



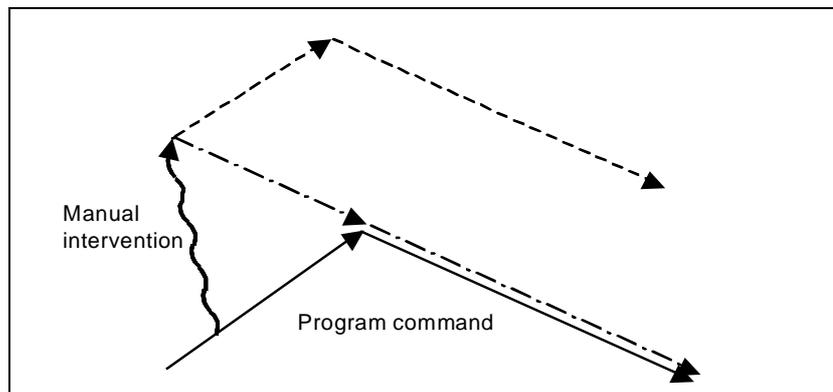
After manual intervention is performed when the execution of a block has been terminated by single block stop, the tool also moves so that the tool center moves towards the end of the next vector in the program.

An MDI operation can also be intervened as well as manual operation. The movement is the same as that by manual operation.



3.8 EFFECT OF MANUAL INTERVENTION

Section 4.7, "Manual Absolute On and Off," describes the effect of manual intervention on the execution of the remaining part of the program. This section describes in detail the relationship between manual intervention and the ABS bit (bit 3) in parameter No. 2409, the manual/ absolute switch, and the G90 and G91 modes.



- (1) When the manual/absolute switch is off: .
- (2) When the manual/absolute switch is on:
 - (a) When the G90 mode is selected: .
 - (b) When the G91 mode is selected:
 - (i) When the INC bit in parameter No. 2409, is set to 1: .
 - (ii) When the INC bit 3 (ABS) is set to 0: .

In this case, the tool moves in the same manner as the operation of the absolute command described in Section 4.7, "Manual Absolute On/Off."

3.9 THREE-DIMENSIONAL HANDLE FEED

Three-dimensional handle feed includes three special handle feed modes, a function for changing the tool length compensation in the longitudinal direction of the tool, and two screen display functions. The three-dimensional handle feed function is used to execute special handle feed for a five-axis diesinking machine. These handle feed modes function even during manual handle interrupt.

-Special handle feed modes

- 1 Mode of handle feed in the longitudinal direction of the tool
 - 2 Mode of handle feed in the transverse direction of the tool
 - 3 Mode of rotational handle feed around the center of the tool tip
- * These three modes are called as special handle feed modes.

-Function for changing the tool length compensation in the longitudinal direction of the tool

-Screen display functions

- 1 Function that displays the coordinate of the tool tip
- 2 Function that displays the number of pulses in a manual interrupt and the traveled distance

The following variables are used in expressions described in the subsequent sections:

Xp, Yp, Zp : Number of handle feed pulses for the axes (integrated value)

Hp : Number of specified handle feed pulses (integrated value)

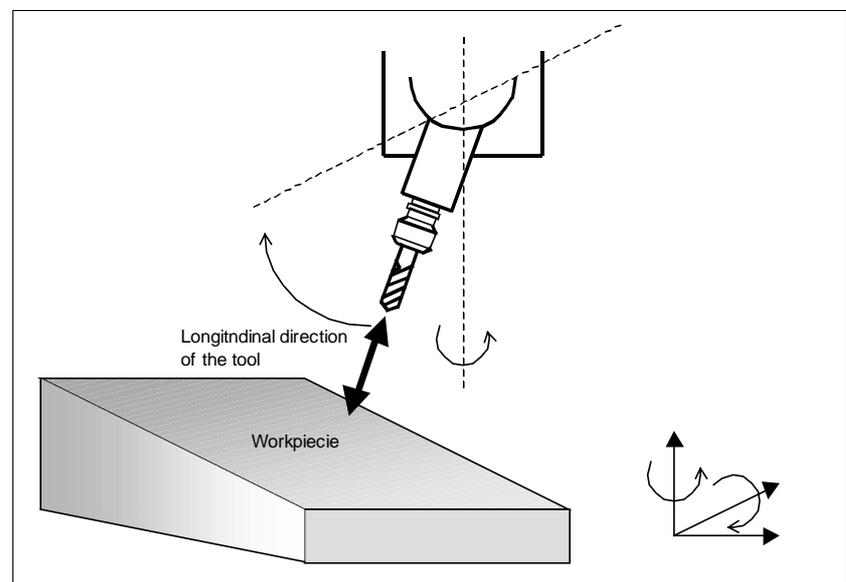
A, B, C : Machine coordinate of the A-, B-, and C-axes

L : Distance from the center of rotation of the tool to the tool tip

3.9.1 Handle feed interruption in the longitudinal direction of the tool

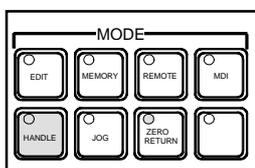
Select the mode of handle feed in the longitudinal direction of the tool and rotate the handle of the manual pulse generator. The tool rotates about the rotation axis, causing the slanted tool to move in the longitudinal direction of the tool by the specified distance.

If the mode is selected in the manual handle interrupt mode in automatic operation, the longitudinal movement by the manual pulse generator is superimposed on the movement by automatic operation.



Tool Axis Direction Handle Feed

Procedure



- 1 Select the HANDLE switch from the mode selection switches.Procedure
- 2 Select the tool axis direction handle feed switch.
- 3 Select the tool axis direction handle feed mode axis as the handle feed axis for the first manual pulse coder (parameter No. 7121).
- 4 When the handle is turned, the tool moves in the tool axis direction by the corresponding distance.

When both the tool axis direction handle feed mode and tool axis right-angle direction handle feed mode are selected, neither mode is set, but the ordinary handle mode is set.

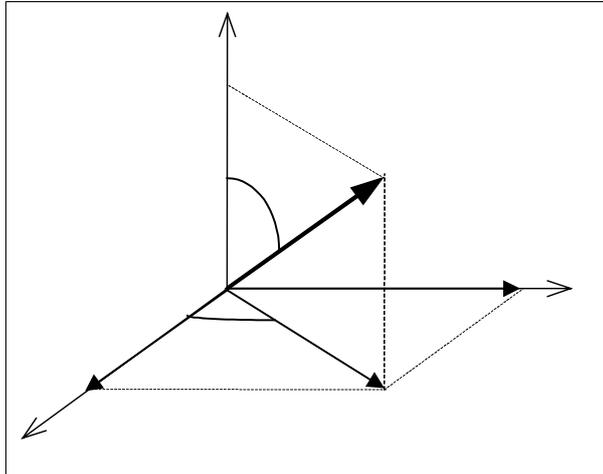
The procedure above is only an example. Refer to the relevant manual published by the machine tool builder for other possible operations.

Explanation

£ Pulses distributed to each axis

When handle feed/interruption in the longitudinal direction of the tool is performed, pulses are distributed to the axes according to the machine configuration as explained below.

£ Rotary axes B and C (with the Z-axis representing the tool axis)

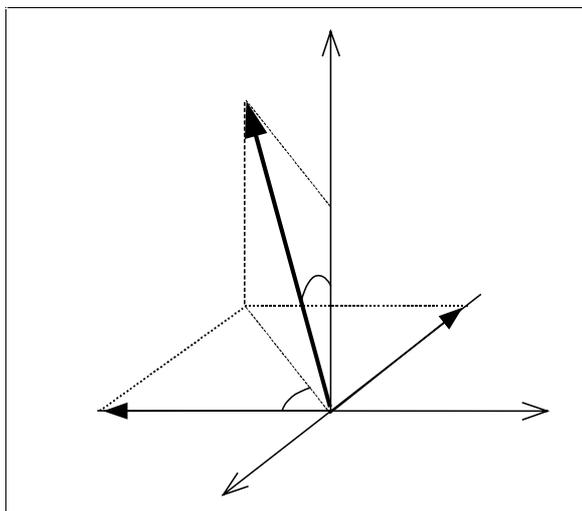


$$X_p.H_p \times \sin(B) \times \cos(C)$$

$$Y_p.H_p \times \sin(B) \times \sin(C)$$

$$Z_p.H_p \times \cos(B)$$

£ Rotary axes A and C (with the Z-axis representing the tool axis)

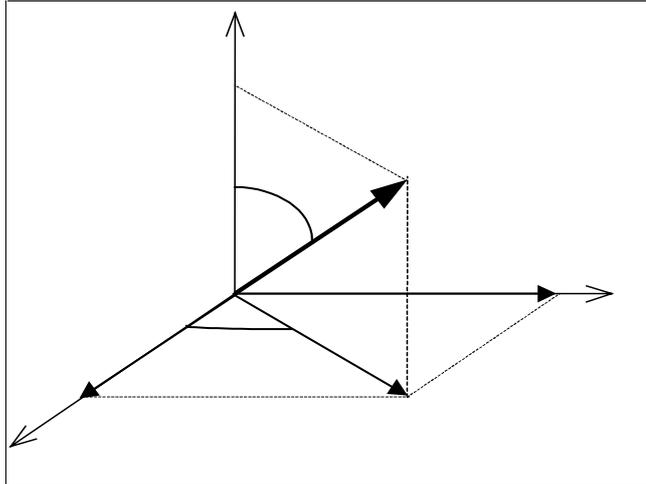


$$X_p.H_p \times \sin(A) \times \sin(C)$$

$$Y_p.H_p \times \sin(A) \times \cos(C)$$

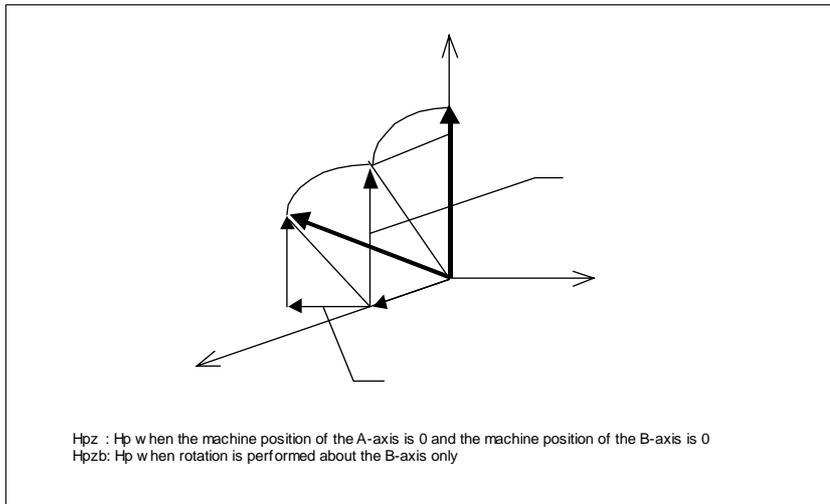
$$Z_p.H_p \times \cos(A)$$

ε Rotary axes A and B (with the X-axis representing the tool axis)



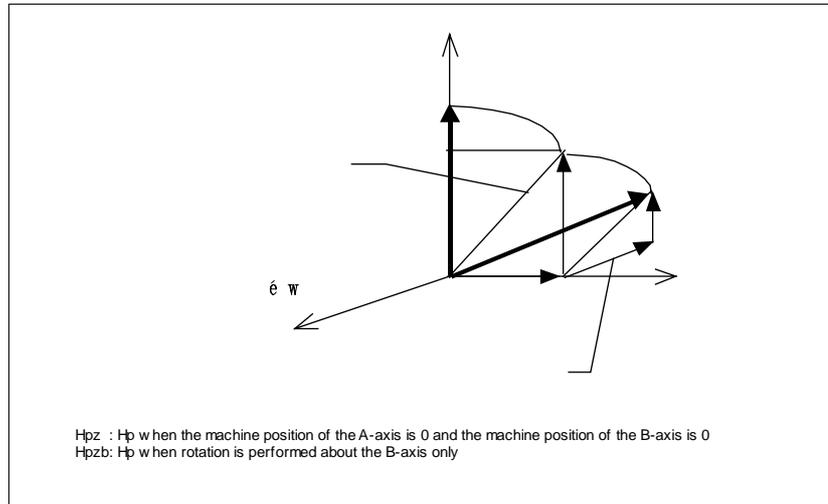
$$\begin{aligned}
 X_p &= H_p \times \cos(B) \\
 Y_p &= H_p \times \sin(B) \times \sin(A) \\
 Z_p &= H_p \times \sin(B) \times \cos(A)
 \end{aligned}$$

-Rotary axes A and B (with the A-axis being the master and the Z-axis representing the tool axis)



$$\begin{aligned}
 X_p &= H_p \times \sin(B) \\
 Y_p &= H_p \times \cos(B) \times \sin(A) \\
 Z_p &= H_p \times \cos(B) \times \cos(A)
 \end{aligned}$$

-Rotation axes A and B (with the B-axis being the master axis and the Z-axis representing the tool axis)



$$X_p = H_p \times \cos(A) \times \sin(B)$$

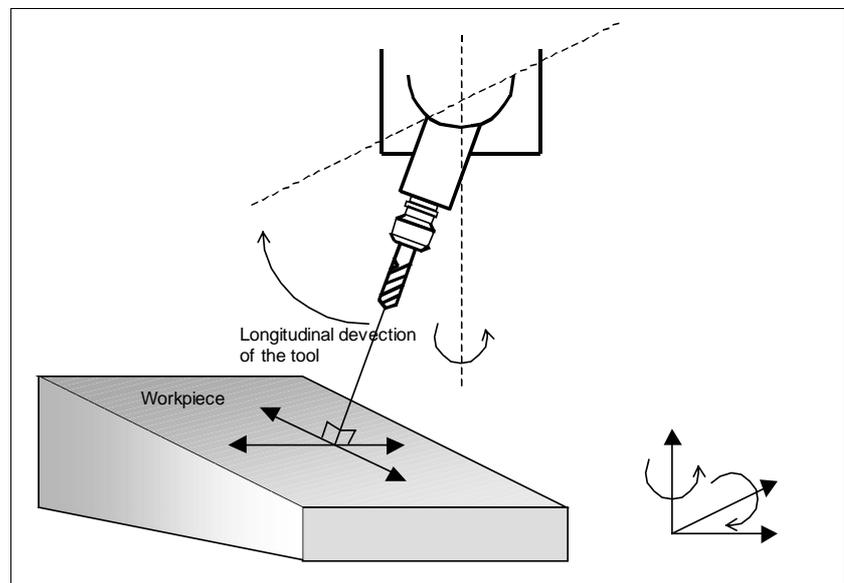
$$Y_p = H_p \times \sin(A)$$

$$Z_p = H_p \times \cos(A) \times \cos(B)$$

3.9.2 Handle feed in the transverse direction of the tool

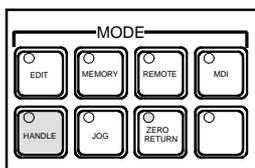
Select the mode of handle feed in the transverse direction of the tool and rotate the handle of the manual pulse generator. The tool rotates about the rotation axis, causing the slanted tool to move in the specified direction perpendicular to the longitudinal direction of the tool by the specified distance.

If the mode is selected in the manual handle interrupt mode in automatic operation, the transverse movement by the manual pulse generator is superimposed on the movement by automatic operation.



Tool Axis Direction Handle Feed

Procedure



- 1 Select the HANDLE switch from the mode selection switches.Procedure
- 2 Select the tool axis direction handle feed switch.
- 3 Select the tool axis direction handle feed mode axis as the handle feed axis for the first manual pulse coder (parameter No. 7121).
- 4 When the handle is turned, the tool moves in the tool axis direction by the corresponding distance.

When both the tool axis direction handle feed mode and tool axis right-angle direction handle feed mode are selected, neither mode is set, but the ordinary handle mode is set.

The procedure above is only an example. Refer to the relevant manual published by the machine tool builder for other possible operations.

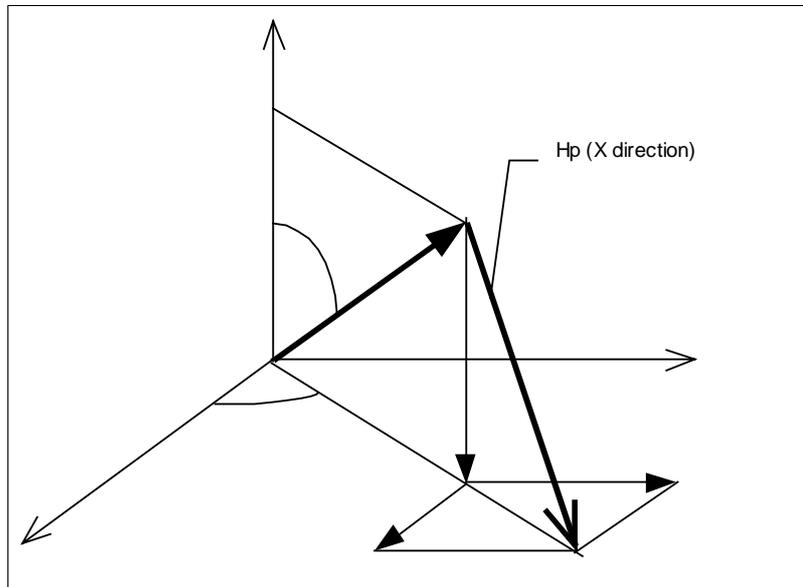
Explanation

£ **Pulses distributed to each axis**

When handle feed/interruption in the transverse direction of the tool is performed, pulses are distributed to the axes according to the machine configuration as explained below.

£ **Rotation axes B and C (with the Z-axis representing the tool axis)**

- When handle feed is performed in the transverse direction of the tool (in the X-axis direction)

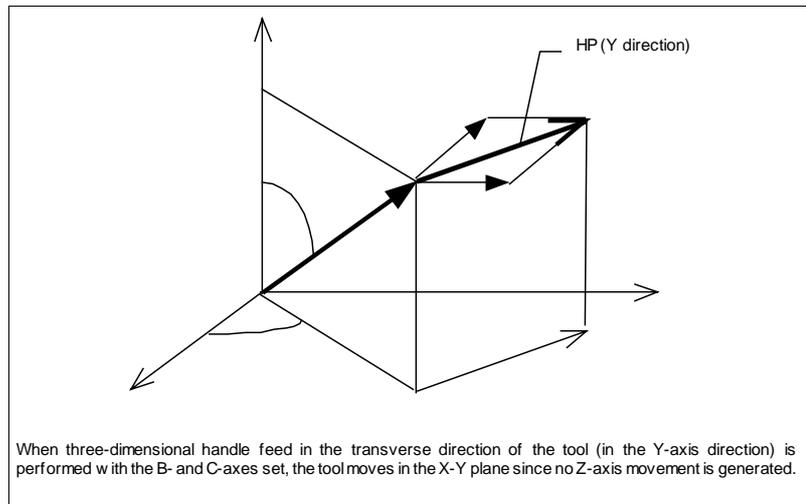


$$Xp.Hp \times \cos(B) \times \cos(C)$$

$$Yp.Hp \times \cos(B) \times \sin(C)$$

$$Zp.Hp \times \sin(B)$$

- When handle feed is performed in the transverse direction of the tool (in the Y-axis direction)



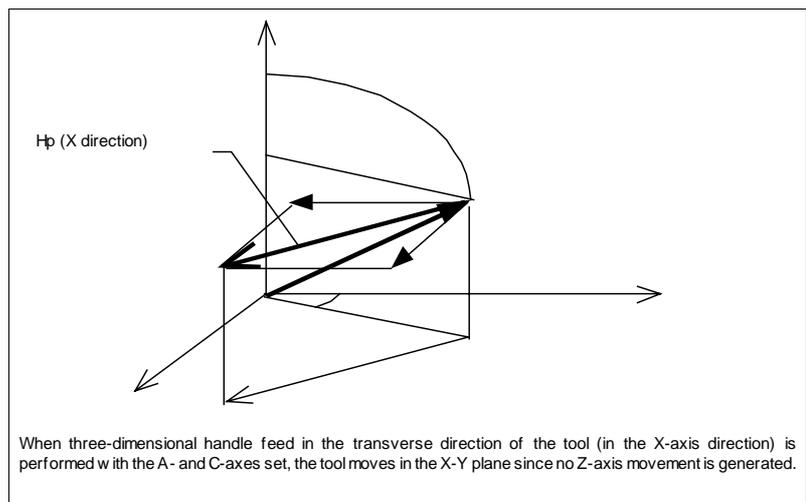
$$X_p..H_p \times \sin(C)$$

$$Y_p.H_p \times \cos(C)$$

$$Z_p.0$$

£ Rotation axes A and C (with the Z-axis representing the tool axis)

- When handle feed is performed in the transverse direction of the tool (in the X-axis direction)

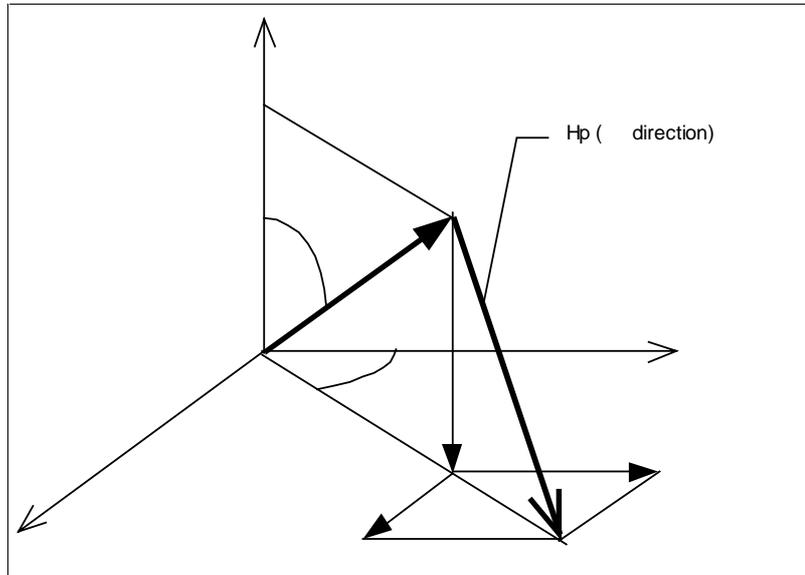


$$X_p.H_p \times \cos(C)$$

$$Y_p.H_p \times \sin(C)$$

$$Z_p.0$$

- When handle feed is performed in the transverse direction of the tool (in the Y-axis direction)



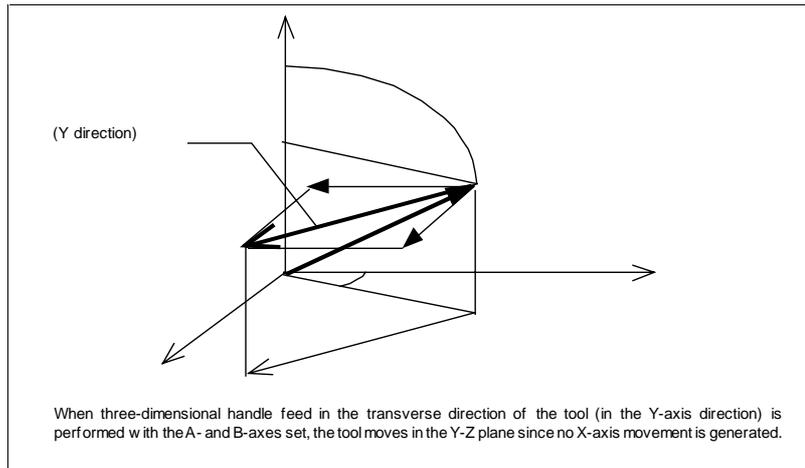
$$X_p = H_p \times \cos(A) \times \sin(C)$$

$$Y_p = H_p \times \cos(A) \times \cos(C)$$

$$Z_p = H_p \times \sin(A)$$

ε Rotation axes A and B (with the X-axis representing the tool axis)

- When handle feed is performed in the transverse direction of the tool (in the Y-axis direction)

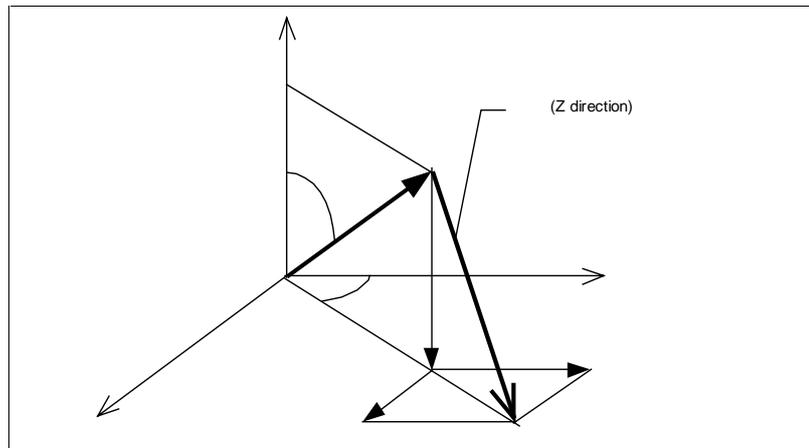


$$X_p = 0$$

$$Y_p = H_p \times \cos(A)$$

$$Z_p = H_p \times \sin(A)$$

- When handle feed is performed in the transverse direction of the tool (in the Z-axis direction)



$$X_p \cdot H_p \times \sin(B)$$

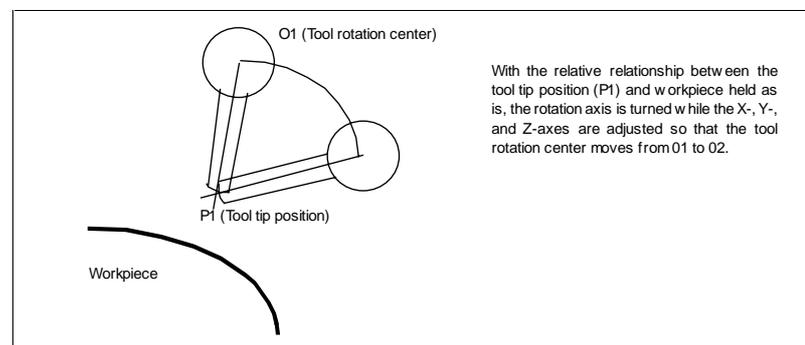
$$Y_p \cdot H_p \times \cos(B) \times \sin(A)$$

$$Z_p \cdot H_p \times \cos(B) \times \cos(A)$$

3.9.3 Rotational handle feed around the center of the tool tip

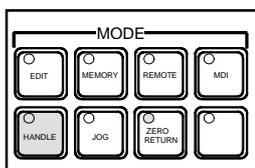
Select the mode of rotational handle feed around the center of the tool tip and rotate the handle of the manual pulse generator. When the rotation axis (A-, B-, or C-axis) is moved, the X-, Y-, and Z-axes are adjusted so that the movement of the rotation axis does not shift the tool tip.

If this mode is selected in the manual handle interrupt mode in automatic operation, rotation around the center of the tool tip by the manual pulse generator is superimposed on the movement by automatic operation.



Rotational Handle Feed Around Tool Tip

Procedure



- 1 Select the HANDLE switch from the mode selection switches.
- 2 Select the rotational handle feed around tool tip switch.
- 3 Select the rotational handle feed around tool tip mode axis as the handle feed axis for the first manual pulse generator (parameter Nos. 7554, 7555).
- 4 When the handle is turned, the tool rotates around the tool tip by the corresponding distance.

When the tool direction handle feed mode and tool axis right-angle direction handle feed mode are selected, neither mode is set, but the ordinary handle mode is set.

The procedure above is only an example. Refer to the relevant manual published by the machine tool builder for other possible operations.

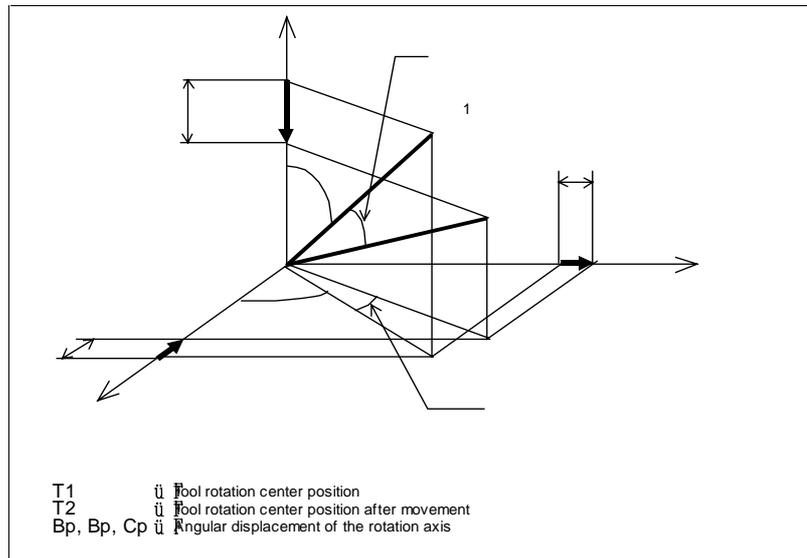
Explanation

£ Pulse distribution to basic axes

In the case of rotation around tool tip, linear axis movement (X_p , Y_p , Z_p) calculated by the equation below is output simultaneously with

rotation of the rotary axis. As a result, the tool tip position hardly moves. (The tool tip may move when the rotary axis is rotated at high speed.) The pulse distribution to basic axes by rotational handle feed/interruption around tool tip is as follows according to your machine tool confirmation:

£ Rotation axes B and C (with the Z-axis representing the tool axis)



Movement amount of linear axis

$$Xp.Lx[\sin(B.Bp) \times \cos(C.Cp) \cdot \sin(B) \times \cos(C)]$$

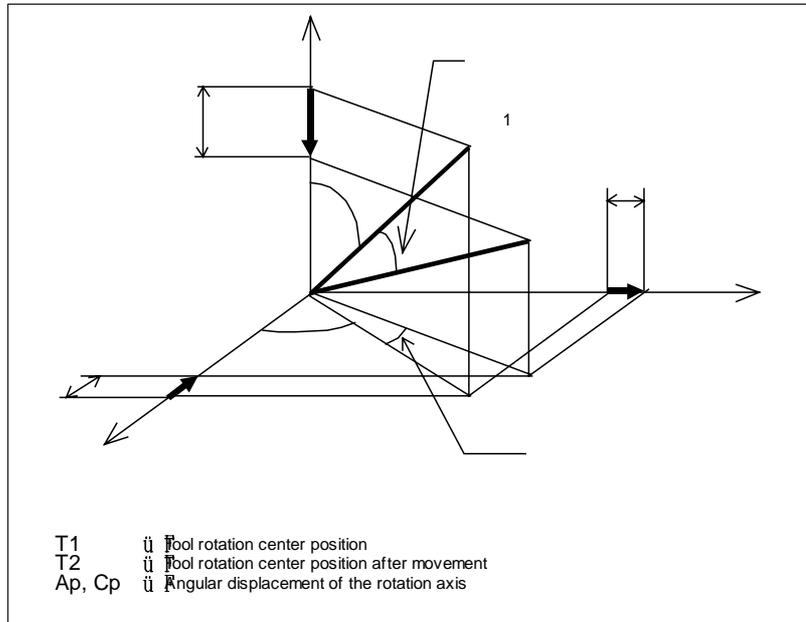
$$Yp.Lx[\sin(B.Bp) \times \sin(C.Cp) \cdot \sin(B) \times \sin(C)]$$

$$Zp.Lx[\cos(B.Bp) \cdot \cos(B)]$$

Angular displacement of the rotation axis

$$Bp.Hp \text{ or } Cp.Hp$$

£ Rotation axes A and C (with the Z-axis representing the tool axis)



Movement amount of linear axis

$$Xp.Lx[\sin(A.Ap) \times \sin(C.Cp) \cdot \sin(A) \times \sin(C)]$$

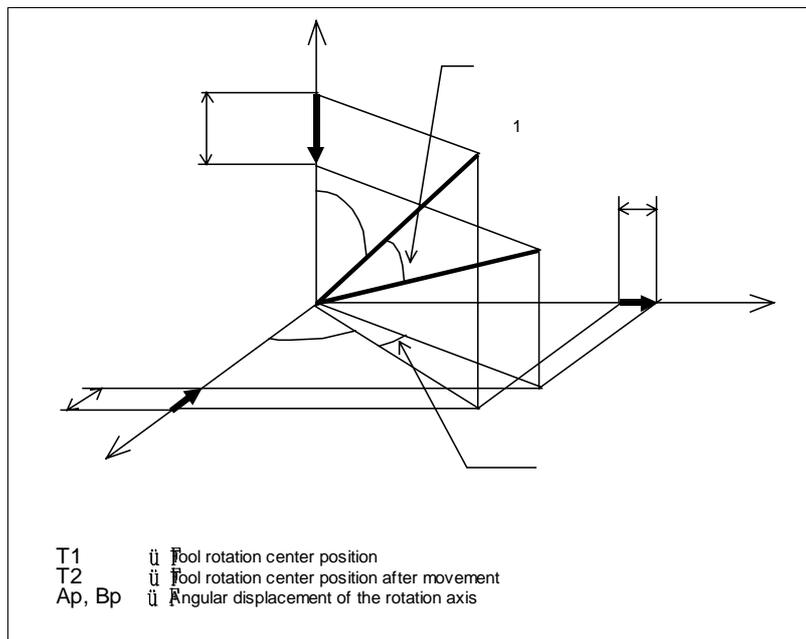
$$Yp.Lx[\sin(A.Ap) \times \cos(C.Cp) \cdot \sin(A) \times \cos(C)]$$

$$Zp.Lx[\cos(A.Ap) \cdot \cos(A)]$$

Angular displacement of the rotation axis

$$Ap.Hp \text{ or } Cp.Hp$$

£ Rotation axes A and B (with the X-axis representing the tool axis)



Movement amount of linear axis

$$X_p.L \times [\cos(B.B_p) \cdot \cos(B)]$$

$$Y_p.L \times [\sin(B.B_p) \times \sin(A.A_p) \cdot \sin(B) \times \sin(A)]$$

$$Z_p.L \times [\sin(B.B_p) \times \cos(A.A_p) \cdot \sin(B) \times \cos(A)]$$

Angular displacement of the rotation axis

$A_p.H_p$ or $B_p.H_p$

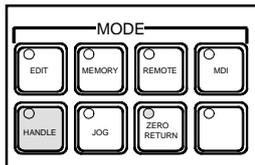
3.10 CHANGING TOOL LENGTH COMPENSATION IN THE LONGITUDINAL DIRECTION OF THE TOOL

Select the mode of changing the tool length compensation in the longitudinal direction of tool when both the following modes are selected: Mode of tool length compensation in the longitudinal direction of the tool (G43.1) and mode of manual handle interrupt (MOVL and H are on). Then rotate the handle of the manual pulse generator. The tool length compensation is changed in the longitudinal direction of the tool.

The change is cleared to zero when a reset or the G49 command is executed or when the offset number is changed.

Changing the Tool Length Compensation Along the Tool Axis

Procedure

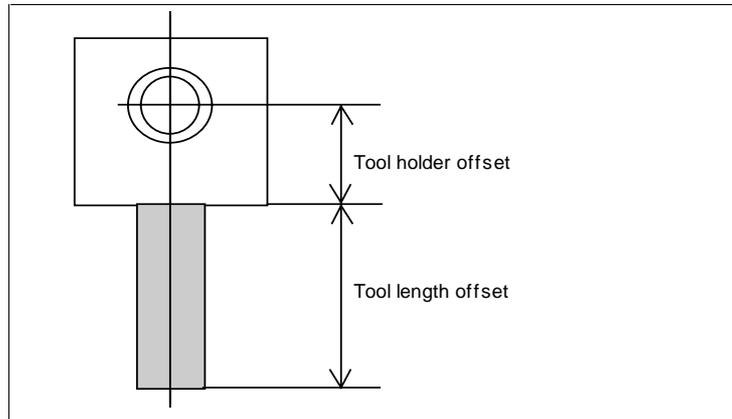


- 1 Make sure that the tool length compensation along the tool axis mode (G43.1) mode is entered and that manual handle interruption is enabled.
- 2 Select the tool length compensation along the tool axis change mode switch.
- 3 Select the axis whose tool length compensation along the tool axis is to be changed as the handle feed axis for the first manual pulse generator (parameter Nos. 7556).
- 4 When the handle is turned, the tool length compensation along the tool axis is changed by the corresponding distance.

The procedure above is only an example. Refer to the relevant manual published by the machine tool builder for other possible operations.

3.11 Tool Holder Offset

The length unique to the machine tool from the rotation center of a rotary axis (A and B axes, A and C axes, B and C axes) around which the tool axis rotates to the tool mounting position is called the "tool holder offset." This tool holder offset can be set in parameter No. 7548 in addition to the tool length offset.



By setting bit 6 (DTH) of parameter No. 7540 to 1, the tool tip coordinates can be displayed with the tool holder offset added.

3.12

Explanation

- Rotary axis origin compensation

When a rotary axis deviates slightly due to thermal displacement, the deviation of the origin of the rotary axis can be set in parameter No. 7518 to perform origin compensation. The three-dimensional handle feed is calculated with the angular displacement resulting from compensating the angular displacement of the rotary axis with the origin compensation set in the appropriate parameter.

- Rotary axis offset

The offset for the angular displacement of a rotary axis can be set in parameter No. 7517 to perform compensation. The three-dimensional handle feed is calculated with the angular displacement resulting from compensating the angular displacement of the rotary axis with the offset set in the appropriate parameter.

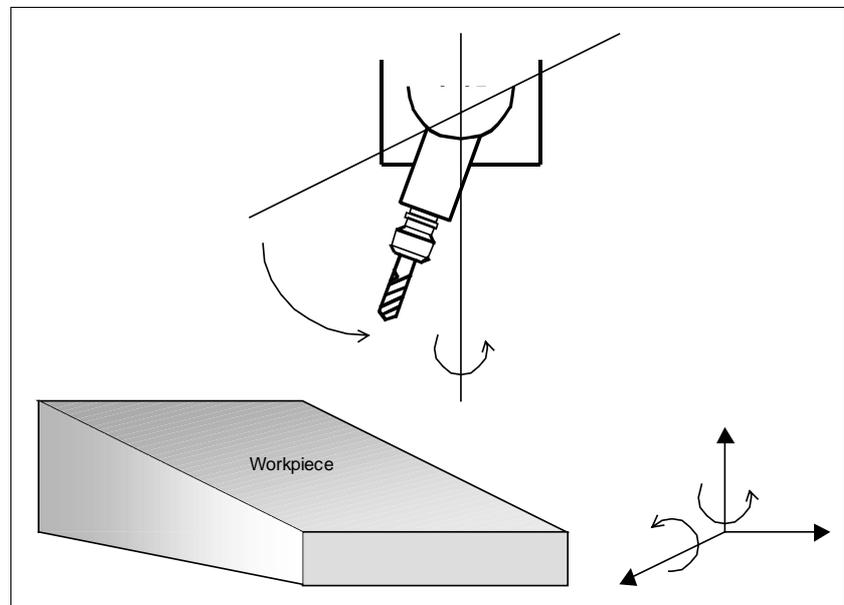
The angular displacement of a rotary axis that is used in the calculation of the three-dimensional handle feed/interrupt when rotary axis origin compensation and compensation with a rotary axis offset are performed is as follows:

Angular displacement of the rotary axis = (angle of the rotary axis in the machine coordinate system) - [parameter No. 7517 + parameter No. 7518].No.7518.]

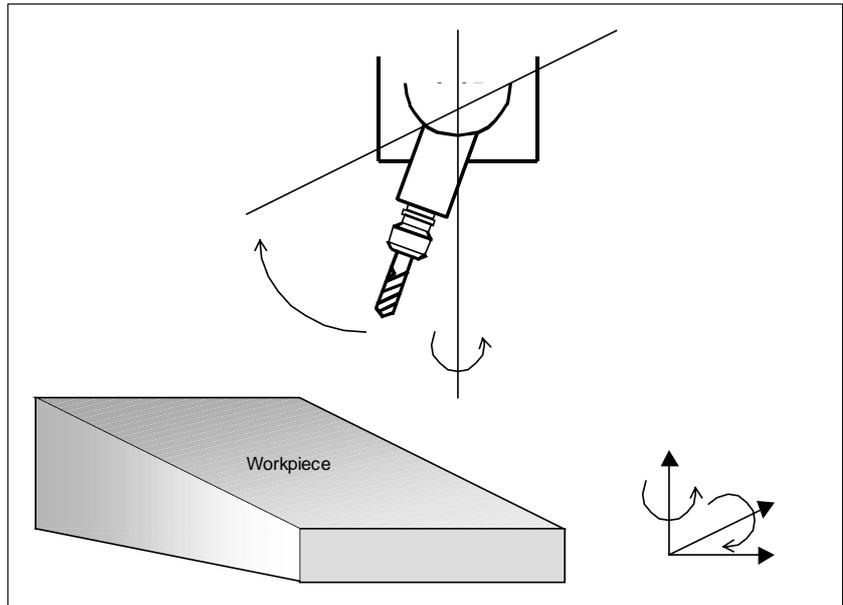
3.13 PRECAUTIONS DURING 3-DIMENSIONAL HANDLE FEED

- Interruption by 3-dimensional handle feed/interruption function is effective when manual handle interruption is added on at the same time.
- When performing interruption by the 3-dimensional handle, rotary axis commands must not be in execution during automatic operation.
- The following axis configurations are supported on the machine tool as described in the "Tool direction handle feed/Interruption" item.

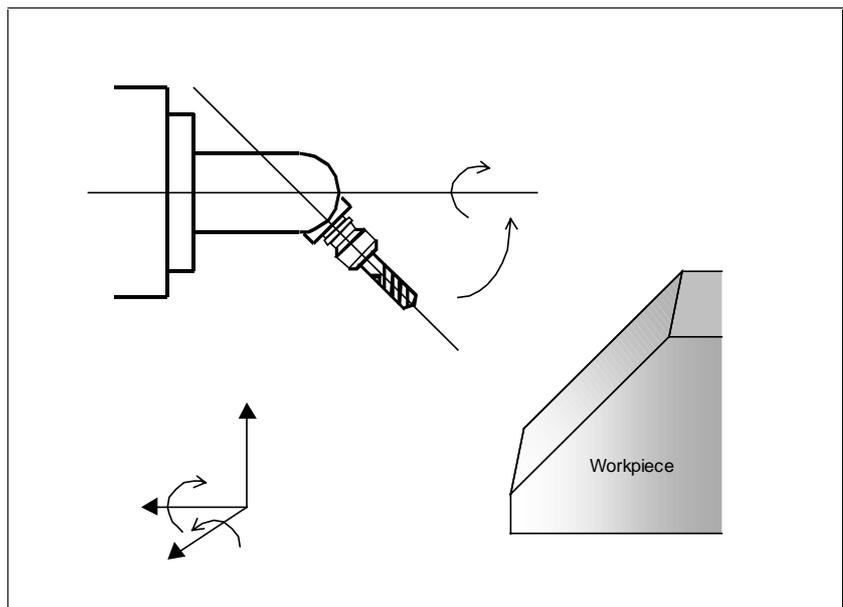
(1) Tool direction Z axis, rotary axes A and C axes

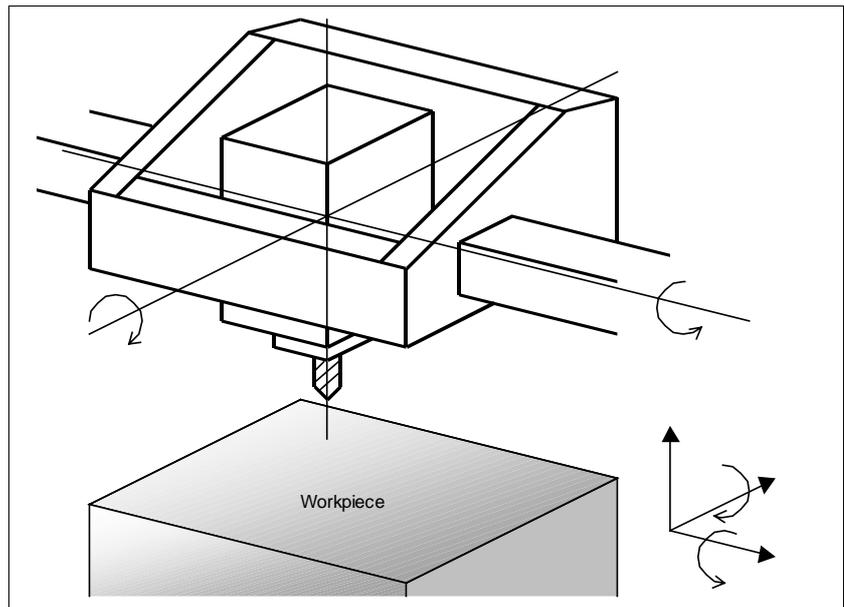
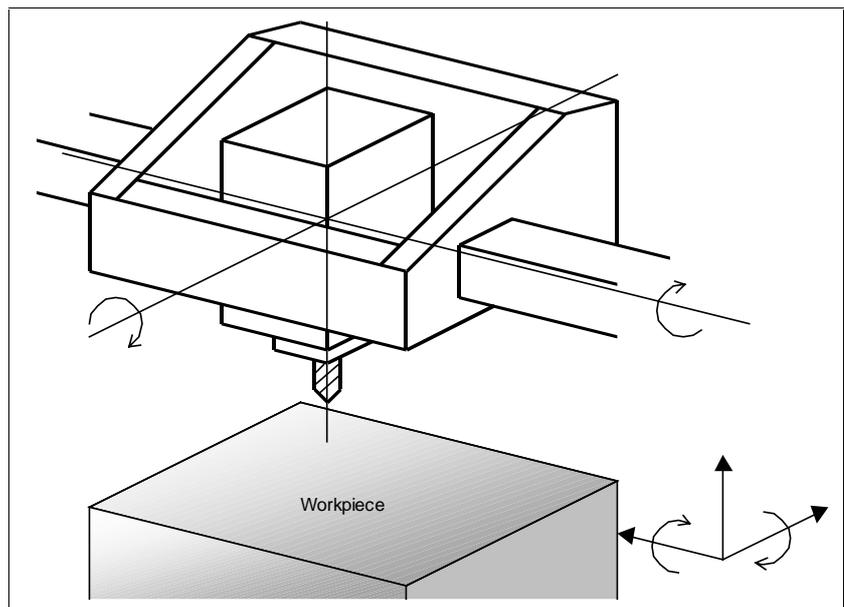


(2) Tool direction Z axis, rotary axes B and C axes



(3) Tool direction X axis, rotary axes A and B axes



(4) Tool direction Z axis, rotary axes A and B axes (A-axis master)**(5) Tool direction Z axis, rotary axes A and B axes (B-axis master)**

- . Only the first manual pulse generator can be used in the 3-dimensional handle feed/interruption mode.
- . Tool feed directions by tool direction handle feed /interruption and by tool normal direction handle feed /interruption are determined when the 3-dimensional handle feed/interruption mode is selected. So, operation of rotary axes by which the tool direction is determined must be completed before you select the 3-

dimensional handle feed/interruption mode. Even if the rotary axes are moved after the 3-dimensional handle feed /interruption mode is entered, and the tool direction is changed, operation along the changed tool direction is not performed.

- . Do not change parameters relating to the 3-dimensional handle feed/interruption in the 3-dimensional handle feed/interruption mode.
- . 3-dimensional handle interruption is not possible in the high-precision contour control mode.

3.14 DISPLAYING THE COORDINATES OF THE TOOL TIP

The absolute coordinates of the tool tip and the actual speed are displayed when handle feed in the longitudinal or transverse direction of the tool or rotational handle feed around the center of the tool tip is executed. Even in the manual interrupt mode, the coordinate and the speed are displayed in the same manner.

The tool tip coordinates to be displayed are calculated based on the current position on each axis and the distance from the tool rotation center to the tool tip which is set in parameter No. 7557.

Explanation

£ Expressions for calculating the tool tip coordinates to be displayed

The tool tip coordinates to be displayed are obtained from the current position on each axis in the machine configuration by using expressions shown below.

X_d, Y_d, Z_d : Tool tip position

X, Y, Z, A, B, C : Current position on each axis from the origin of the workpiece coordinate system

L : Distance from the tool rotation center to the tool tip, set in parameter No. 7557

(1) When the rotation axes are the B- and C-axes, and the tool axis is the Z-axis

$$X_d = X \cdot L \cdot \cos(C) \cdot \sin(B)$$

$$Y_d = Y \cdot L \cdot \sin(C) \cdot \sin(B)$$

$$Z_d = Z \cdot L \cdot \cos(B)$$

(2) When the rotation axes are the A- and C-axes, and the tool axis is the Z-axis

$$X_d = X \cdot L \cdot \sin(C) \cdot \sin(A)$$

$$Y_d = Y \cdot L \cdot \cos(C) \cdot \sin(A)$$

$$Z_d = Z \cdot L \cdot \cos(A)$$

(3) When the rotation axes are the A- and B-axes, and the tool axis is the X-axis

$$X_d = X \cdot L \cdot \cos(B)$$

$$Y_d = Y \cdot L \cdot \sin(A) \cdot \sin(B)$$

$$Z_d = Z \cdot L \cdot \cos(A) \cdot \sin(B)$$

(4) When the rotation axes are the A- and B-axes, the tool axis is the Z-axis, and the master axis is the A-axis

$$X_d = X \cdot L \cdot \sin(B)$$

$$Y_d = Y \cdot L \cdot \sin(A) \cdot \cos(B)$$

$$Z_d = Z \cdot L \cdot \cos(A) \cdot \cos(B)$$

- (5) When the rotation axes are the A- and B-axes, the tool axis is the Z-axis, and the master axis is the B-axis

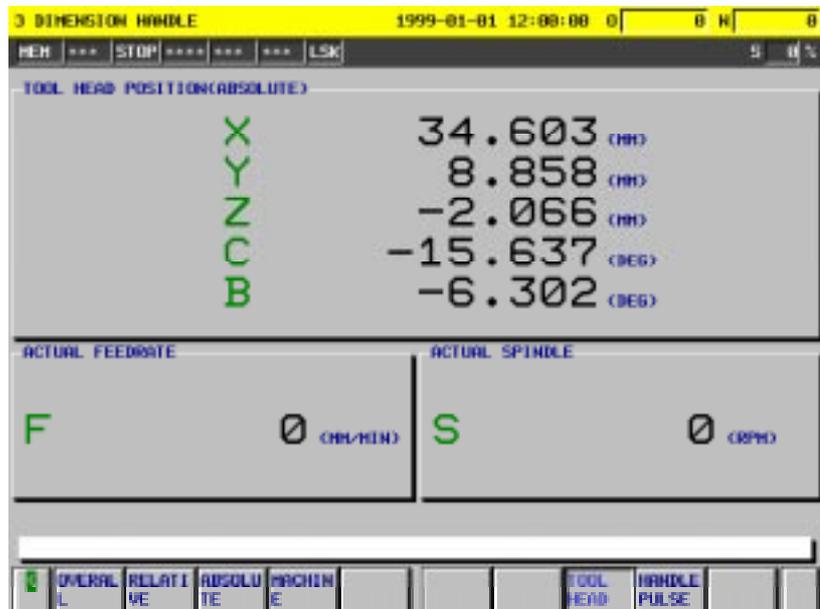
$$X_d = X.L \times \sin(B) \times \cos(A)$$

$$Y_d = Y.L \times \sin(A)$$

$$Z_d = Z.L \times \cos(B) \times \cos(A)$$

Tool tip coordinates display screen

Follow the procedure described in Section III.x.x, "Displaying Tool Tip Coordinate Display Screen" to display the tool tip coordinate display screen shown below.



Display data

£ Tool Head Position

The axis addresses of the machine axes used for three-dimensional handle feed and the current position of the tool tip are displayed.

£ Actual feedrate

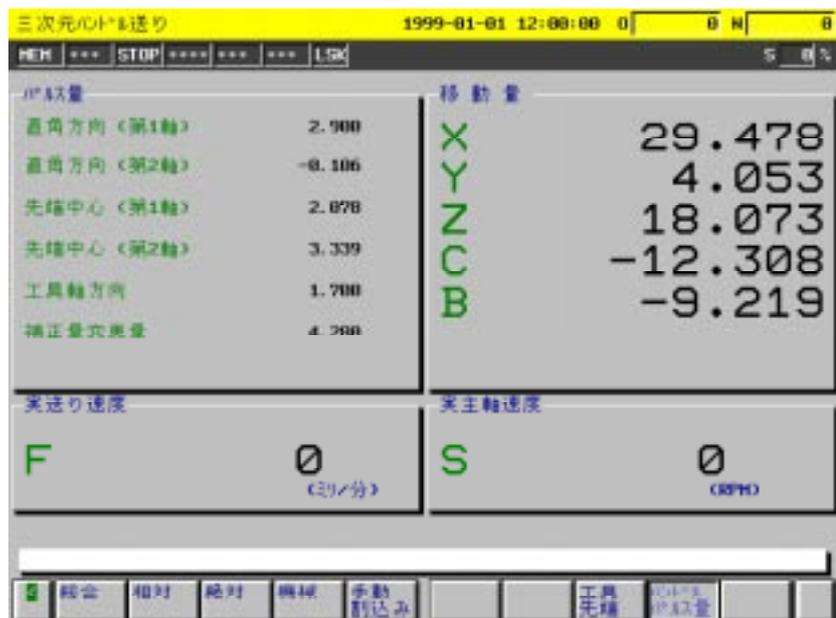
The actual feedrate of the tool tip is displayed.

3.15 Displaying Pulse Values and Amount of Movement by Manual Interrupt

The amount of a handle interrupt is displayed for each of the functions including handle feed in the longitudinal direction of the tool, handle feed in the transverse direction of the tool, rotational handle feed around the center of the tool tip, and change of tool length compensation in the longitudinal direction of the tool. In addition, the total amount of movement when each handle feed function is performed is displayed.

Screen displaying pulse values/amount of movement by manual interrupt

Follow the procedure described in Section III.x.x, "Displaying the Screen Displaying Pulse Values/Amount of Movement by Manual Interrupt" to display the screen displaying pulse values/amount of movement by manual interrupt, shown below.



Display data

£ Pulse value

The amount of handle pulse interrupt is displayed for each three-dimensional handle feed/interrupt function. For Tool Direction and Change offset, one handle pulse equals the least input increment on

the tool axis. For the other functions, one handle pulse equals the least input increment on a target axis for tool movement.

£ **RIGHT DIR**

The amount of handle pulse interrupt in the mode of handle feed/interrupt in the transverse direction of the tool is indicated in units of the least input increment of a target axis for movement.

£ **Tip center**

The amount of handle pulse interrupt in the mode of rotational handle feed/interrupt around the center of the tool tip is indicated in units of the least input increment of the target axis for movement.

£ **Tool Direction**

The amount of handle pulse interrupt in the mode of handle feed/interrupt in the longitudinal direction of the tool is indicated in units of the least input increment of the tool axis.

£ **Changed offset**

The amount of handle pulse interrupt in the mode of change of tool length compensation in the longitudinal direction of the tool is indicated in units of the least input increment of the tool axis.

£ **Motion value**

The addresses of the machine axes used for three-dimensional handle feed and the total amount of movement for each axis by three-dimensional handle feed are indicated.

£ **Actual feedrate**

The actual feedrate at the tool tip is indicated.

Operation

The displayed pulse values can be cleared using soft keys.

- 1 On the screen displaying pulse values/amount of movement by manual interrupt, press the operation menu key (located at the right end).



- 2 Select the soft key of the function for which pulse values are to be cleared.



- 3 To clear the pulse values of the specified function, press the [CLEAR PULSE] soft key. To cancel the operation, press [CANCEL].



3.16 MANUAL INTERRUPTION FUNCTION FOR THREE-DIMENSIONAL COORDINATE CONVERSION

When the handle of the manual pulse generator is rotated in the three-dimensional coordinate conversion mode, this function adds the travel distance specified by the manual pulse generator to the travel distance during automatic operation.

This addition is performed for the new (converted) three-dimensional coordinate component (programmed coordinate system) in the direction of the axis selected for manual feed using the manual pulse generator.

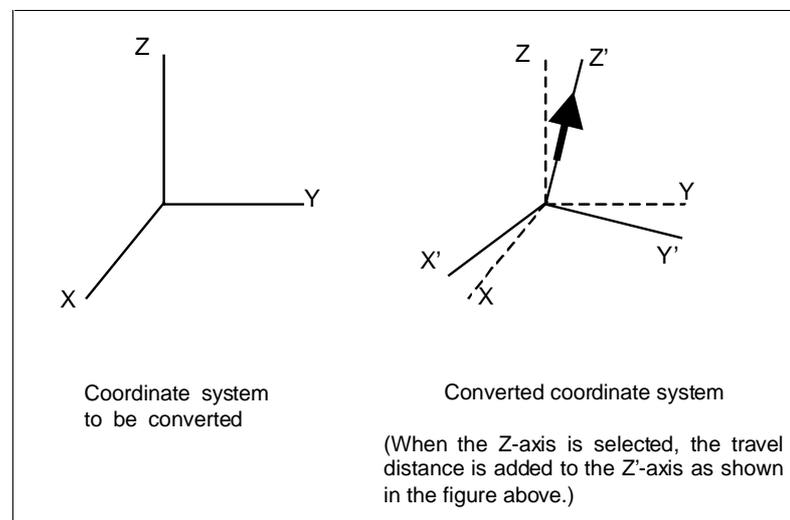


Fig.3.16 (e) For the three basic axes

Explanation

£ Interruption

This function performs interruption only when all the conditions in 1 to 8 are met.

- 1 During simultaneous manual and automatic operation mode (including manual interruption using the manual pulse generator)
- 2 During three-dimensional coordinate conversion (i.e. while blocks G68 to G69 are being executed.)
- 3 During cutting mode (i.e. while blocks from a G code for cutting, such as G01, G02, or G03, to G00 are being executed.)
- 4 When parking along the feed axis is not enabled (i.e. when PKn for the feed axis is "0".)
- 5 When the machine moves along the three axes selected for three-dimensional coordinate conversion

- 6 When the axis for which the function performs interruption is a master axis under parallel axis control (in this case, no slave axes are converted).
- 7 When feed in the tool axis direction or the direction perpendicular to the tool axis or rotation around the tool tip center is not selected for the manual operation or the manual interruption using the manual pulse generator when the ALNGH, RGHTH, and RNDH input signals are all "0".
- 8 While the playback function is not being executed

The result of adding the feedrate in the three-dimensional coordinate conversion mode is not permitted to exceed the maximum cutting speed for each axis.

£ Manual interruption and the coordinate system

When this function is effective, the current position in the workpiece coordinate system does not depend on the travel distance specified during manual interruption using the manual pulse generator. The current position is therefore not updated even when the manual pulse generator is rotated. Each current position in the machine coordinate system and in the relative coordinate system depends on the travel distance specified during manual interruption. As shown in Fig. 1, the travel distance is added along the Z'-axis. The travel distance is displayed using the X, Y, and Z coordinates of the coordinate system to be converted. The travel distance specified by this function can be monitored on the manual overlap screen. The bit 0 (DMK(bit of parameter 2208 specifies whether the travel distance is displayed on the manual overlap screen using the coordinate system before (X, Y, Z) or after (X', Y', Z') three-dimensional coordinate conversion.

4

AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.

This chapter explains the following types of automatic operation:

.MEMORY OPERATION

Operation by executing a program registered in CNC memory

.MDI OPERATION

Operation by executing a program entered from the MDI panel

.DNC OPERATION

Operation while reading a program from an input/output device

.PROGRAM SELECTION

Method for selecting a program to be executed in automatic operation

.EXECUTION OF AUTOMATIC OPERATION

Buffering of the next block during automatic operation

.STOP/TERMINATION OF AUTOMATIC OPERATION

Method for stopping and terminating automatic operation

.RESTART OF AUTOMATIC OPERATION

Restart of automatic operation of a program from an intermediate point

.MANUAL INTERRUPT DURING AUTOMATIC OPERATION

Function for superimposing handle feed on movement by automatic operation

.OVERRIDE PLAYBACK

Function allowing the use of stored override values for operation at program restart

4.1 DNC OPERATION

In the DNC operation mode, automatic operation is carried out while a program is read from an external I/O device via the interface.

When a floppy format external I/O device (Handy File, Floppy Cassette, FA Card) is connected, it is possible to select the program to be read, and specify the sequence and frequency of execution for automatic operation.

The following procedure is given as an example. For details of actual operation, refer to the manual published by the machine tool builder.

DNC OPERATION

£ Mode selection

Press the DNC mode selection switch.

£ Program selection

Select the program to be executed from the programs currently stored on the external I/O device.

For details on how to select the program, refer to X.X Selection of Execution Programs.

£ Starting DNC operation

When you press the Cycle Start button on the machine operator's panel, DNC operation is started and the cycle start lamp lights. When DNC operation is completed, the cycle start lamp goes out.

£ Halting/ending DNC operation

There are two ways of halting/ending DNC operation, by programming the halt command or by key operation on the machine operator's panel.

For details on the commands and operations for halting/ending DNC operation, refer to X.X Halting/Ending Automatic Operation.

4.2 MEMORY OPERATION

In the memory operation mode, automatic operation is performed according to a program stored in memory in advance.

When the Cycle Start button on the machine operator's panel is pressed after the program to execute is selected, memory operation is started, and the cycle start lamp lights.

If the feed hold switch on the machine operator's panel is pressed during memory operation, machine operation is paused. To restart machine operation, press the Cycle Start button again.

When the [RESET] key on the MDI panel is pressed, automatic operation stops, and the reset state is entered.

The following procedure is given as an example. For details of actual operation, refer to the manual published by the machine tool builder.

Procedure for Memory Operation

£ Mode selection

Press the MEMORY mode selection switch.

£ Program selection

Select the program to be executed from the programs currently stored in memory.

For details on how to select the program, refer to X.X Selection of Execution Programs.

£ Starting DNC operation

When you press the Cycle Start button on the machine operator's panel, memory operation is started and the cycle start lamp lights.

When memory operation is completed, the cycle start lamp goes out.

£ Halting/ending memory operation

There are two ways of halting/ending memory operation, by programming the halt command or by key operation on the machine operator's panel.

For details on the commands and operations for halting / ending memory operation, refer to X.X Halting / Ending Automatic Operation.

4.3 MDI OPERATION

In the MDI mode, a program up to 256 characters can be prepared in the same format as normal programs from the MDI panel and executed from the MDI panel. MDI operation is used for simple test operations.

The following procedure is given as an example. For details of actual operation, refer to the manual published by the machine tool builder.

Procedure for MDI Operation

£ Mode selection

Press the MDI mode selection switch.

£ Preparing programs

Prepare programs in the program screen. Follow the procedure for displaying the program screen to display the program screen.

Prepare a program to be executed in the same format as normal programs. The same operations as in program editing can be used to prepare the program.

For details on program editing, refer to X. Program Editing.

M99 specified in the last block can return control to the beginning of the program after operation ends. M30 does not return control to the beginning of the program.

£ Starting MDI operation

Set the cursor on the top of the program. (Start from an intermediate point is also possible.)

Press the Cycle Start button on the machine operator's panel to start MDI operation.

When the program end (M02, M30) or EOR(%) is executed, the prepared program is automatically erased and operation ends.

When M99 is executed, control returns to the top of the prepared program.

£ Halting/ending MDI operation

There are two ways of halting/ending MDI operation, by programming the halt command or by key operation on the machine operator's panel.

For details on the commands and operations for halting/ ending MDI operation, refer to X.X Halting/ Ending Automatic Operation.

£ Restart

After the editing operation during the stop of MDI operation was done, operation starts from the current cursor position. Limitations

Program for MDI operation

£ Program erasure

A program created in MDI mode is erased in the following cases:

- When M02, M30, or ER(%) is executed in MDI operation
- When memory operation is performed
- When editing is performed in EDIT mode
- When background editing is performed
- When reset key  is pressed

£ Program registration

Programs created in MDI mode cannot be registered.

£ Subprogram nesting

Calls to subprograms (M98) can be specified in a program created in the MDI mode. This means that a program registered in memory can be called and executed during MDI operation.

4.4 SELECTION OF EXECUTION PROGRAMS

This section describes how to select a program to be executed (execution program) from among multiple programs and how to rewind the program to the top of the program.

4.4.1 Program Number Search

Search for the execution program from among currently stored programs by its program number.

Program number search is performed in the program screen.

£ Mode selection

Select the MEMORY mode.

£ Number search

Method 1

- (1) Key in address [O].
- (2) Key in a program number to be searched.
- (3) Press soft key [FRWRD SEARCH] or [BKWRD SEARCH].

Method 2

- (1) Press soft key [FRWRD SEARCH] or [BKWRD SEARCH].
- (2) Key in address [O].
- (3) Key in a program number to be searched.
- (4) Press soft key [EXEC].

Method 3

- (1) Press soft key [FRWRD SEARCH] or [BKWRD SEARCH].
- (2) Press soft key [(PROG#)]. "0" is automatically keyed in.
- (3) Key in a program number to be searched.
- (4) Press soft key [EXEC].

Method 4 To select the next program:

- (1) Press soft key [FRWRD SEARCH].
- (2) Press soft key [NEXT PROG].

Method 5 To select the previous program:

- (1) Press soft key [BKWRD SEARCH].
- (2) Press soft key [NEXT PROG].

NOTE

- 1 If a program having the specified number is not found, you are informed of this on the program screen.
- 2 Program number search is enabled only in the memory operation mode. Program number search is not possible in the DNC operation mode.

4.4.2 Search by sequence number

A selected program is searched for a block by sequence number. This function is used when a program is to be started or restarted from a block having a specified sequence number. Search by sequence number is performed on the program screen.

£ Mode selection

For memory operation, select MEM mode. For DNC operation, select DNC mode.

£ Search

Method 1

- (1) Press the  address key.
- (2) Key in the sequence number of a block you want to search for.
- (3) Press the [FRWRD SEARCH] or [BKWRD SEARCH] soft key.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Press soft key [FRWRD SEARCH] or [BKWRD SEARCH].
- (2) Press the  address key.
- (3) Key in the sequence number of a block you want to search for.
- (4) Press soft key [EXEC].

Method 3

- (1) Press soft key [FRWRD SEARCH] or [BKWRD SEARCH].
- (2) Press the [(SEQ#)] soft key. "N" is entered automatically.
- (3) Key in the sequence number of a block you want to search for.
- (4) Press soft key [EXEC].

£ Alarm

If the specified sequence number is not found, the following alarm is generated:

Number	Message	Contents
PS0060	CANNOT FIND SEQUENCE NUMBER	The specified sequence number was not found in the sequence number search. The sequence number of the jump destination specified in GOTO-- and M99P-- was not found.

4.4.3 Rewind

Rewind searches the top of the execution program.
This is used for starting the program from the top of the program.
Rewind is performed in the program screen

£ Mode selection

If memory operation is in operation, select the MEMORY mode.

£ Rewind

Press the soft key [REWIND].

4.5 EXECUTING AUTOMATIC OPERATION

While executing a block, the CNC usually calculates the next block to convert it into data in the executable form. This is called buffering.

After automatic operation is started, the following are executed:

- 1 A one-block command is read from the specified program.
- 2 The block command is decoded.
- 3 The command execution is started.
- 4 The command in the next block is read.
- 5 Buffering is executed. That is, the command is decoded to allow immediate execution.
- 6 Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.
- 7 Hereafter, automatic operation can be executed by repeating the steps (4) to (6).

4.6 STOP/TERMINATION OF AUTOMATIC OPERATION

Automatic operation can be stopped/terminated by specifying a stop/termination instruction at a desired position in a program. Automatic operation can also be stopped by using a switch on the operator's panel.

Stop method

£ Program stop (M00)

When M00 is specified in a block, automatic operation is stopped after that block is executed. As with single block stop, this stop method maintains all the modal information recorded up to that point in time. Automatic operation is restarted by pressing the cycle start button. The operation may differ depending on the machine tool builder. Refer to the manual provided by the machine tool builder.

£ Optional stop (M01)

As with M00, M01 specified in a block stops automatic operation after that block is executed. In this case, however, automatic operation is stopped only when the optional stop switch on the machine operator's panel is set to ON. The operation may differ depending on the machine tool builder. Refer to the manual provided by the machine tool builder.

£ Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, automatic operation ends, and the reset state is entered. On some machine tools, M30 returns control to the top of the program. For details of actual operation, refer to the manual published by the machine tool builder.

£ Feed hold

When Feed Hold button on the operator's panel is pressed during automatic operation, the tool decelerates to a stop at a time. When pressed, the feed hold lamp on the machine operator's panel lights and the cycle start lamp turns off.

At this time,

Feeding stops if the tool is moving.

Dwell execution stops, if executing.

M, S, T or B operation continues up to the end of the block.

£ Reset

Automatic operation can be stopped and the system can be

made to the reset state by using . .  key or external reset

signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

£ Single-block stop according to the comparison between sequence numbers

When the program contains a block having the same sequence number as the preset sequence number while the program is being executed, the block is executed, then the system enters the single-block stop state.

Setting a sequence number

In setting parameter No. 7681, specify the sequence number of the block you want to stop. For how to set the parameter, see Section x.x.x, "Setting parameter display and setting." When a block having the parameter-set sequence number has been stopped, or when a reset is performed, this parameter is set to 0.

When this parameter is set to 0, this stop function is disabled. Even when there is a block with N0, the block is not stopped.

Conditions for single-lock stop

- 1 When the specified sequence number agrees with the sequence number assigned to the block (such as macro statement, M98, and M99) containing an instruction processed only in the CNC control unit, the system does not enter the single-block stop state.

Example

```
N1 #1 = 1 ;
N2 IF [#1 EQ 1] GOTO8;
N3 GOTO9;
N4 M98P1000L3;
N5 M99;
£
£
```

If the sequence numbers match in case of the block shown in the example above, the system does not enter the single-block stop state.

- 2 When the program contains multiple blocks whose sequence numbers are the same, the block whose sequence number agrees with the specified one first in the execution order is executed, then the system enters the single-block stop state.

- 3 Stopping at the block for which the number of times the block is executed is specified
When the sequence number of this block agrees with the preset sequence number, the block (such as L command of the canned cycle) is executed the specified number of times, then the system enters the single-block stop state.

Example

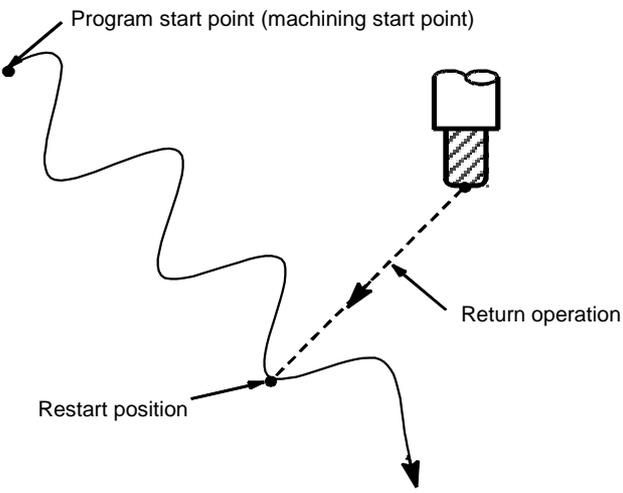
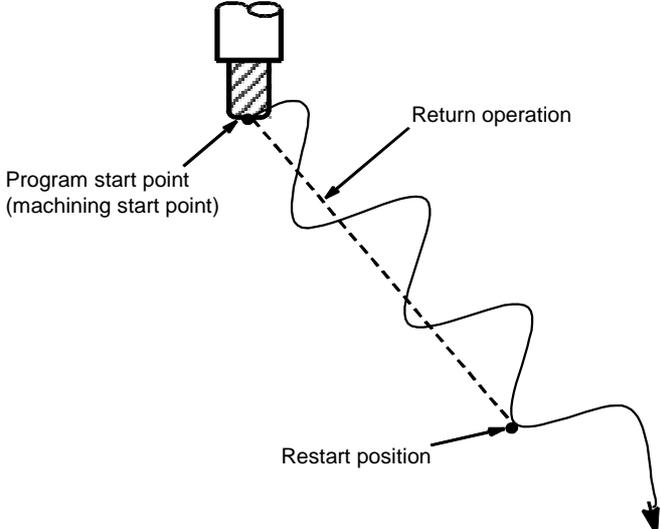
```
N1 G91;  
N2 G81X1000Y1000R-5000Z-2000F100;  
N3 X1000Y1000L3;  
    £  
    £  
    G80;
```

In this example, the cycle of block N3 is executed three times, then the system enters the single-block stop state.

4.7 PROGRAM RESTART

This function specifies Sequence No. of a block (number of blocks between the top of the program and the block to be restarted) to be restarted when a tool is broken down or when it is desired to restart machining operation after a day off, and restarts the machining operation from that block. It can also be used as a high-speed program check function.

There are two restart methods: the P-type method and Q-type method.

P TYPE	Operation can be restarted anywhere. This restart method is used when operation is stopped because of a broken tool.
 <p>The diagram illustrates the P-type restart method. It shows a tool starting at a 'Program start point (machining start point)' and following a wavy path. At a 'Restart position', the tool is broken. A dashed line labeled 'Return operation' shows the tool moving back to the 'Program start point (machining start point)'. From there, the tool resumes its path from the 'Restart position'.</p>	
Q TYPE	Before operation can be restarted, the machine must be moved to the programmed start point (machining start point)
 <p>The diagram illustrates the Q-type restart method. It shows a tool starting at a 'Program start point (machining start point)' and following a wavy path. At a 'Restart position', the tool is broken. A dashed line labeled 'Return operation' shows the tool moving back to the 'Program start point (machining start point)'. From there, the tool resumes its path from the 'Restart position'.</p>	

Procedure for Program Restart by Specifying a Sequence Number

Procedure 1

[P TYPE]

- 1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

- 1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
- 2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
- 3 If necessary, modify the offset amount.

Procedure 2

[P TYPE]

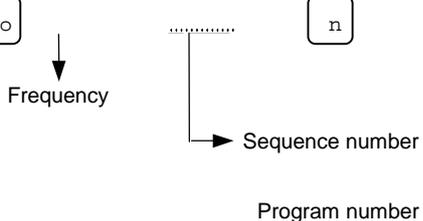
- 1 Turn the program restart switch on the machine operator's panel ON.
For details of the program restart switch, refer to the manual provided by the machine tool builder.



- 2 Press ...  key to display the desired program.
- 3 Find the program head.
- 4 Press the [PROGRAM RESTART] soft key then the [(P_SEQ#)] soft key, enter the sequence number of a block you want to restart, then press the [EXEC] soft key. (Go to step 3.)



If the same sequence number appears more than once, the location of the target block must be specified. Specify a frequency and a sequence number. Specify a frequency then a sequence number. Separate them with a comma ",".

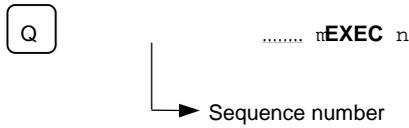


To search only a sequence number in a specific program (e.g. when a subprogram is called several times during a search for a sequence number in a specific subprogram), first specify from which sequence number program execution is to be restarted, then specify sequence number delimited by "," and specify "O" follows by the program number delimited by ",".

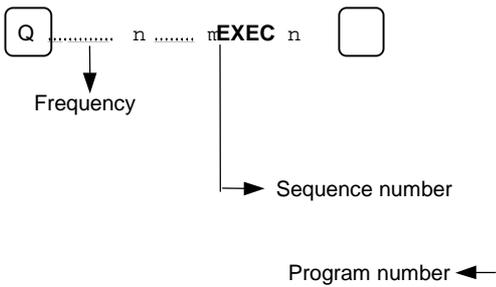
However, note that when the frequency is set to "1" or the program number need not be specified, the number of repeats and program number can be omitted.

[Q TYPE]

- 1 Turn the program restart switch on the machine operator's panel ON.
- 2 Press ..  . key to display the desired program.
- 3 Find the program head.
- 4 Press the [PROGRAM RESTART] soft key then the [(Q_SEQ#)] soft key, enter the sequence number of a block you want to restart, then press the [EXEC] soft key. (Go to step 3.)



If the same sequence number appears more than once, the location of the target block must be specified. Specify a frequency and a sequence number. Specify a frequency then a sequence number. Separate them with a comma ",".



To search only a sequence number in a specific program (e.g. when a subprogram is called several times during a search for a sequence number in a specific subprogram), first specify from which sequence number program execution is to be restarted, then specify sequence number delimited by "," and specify "O" followed by the program number delimited by ",".

However, note that when the frequency is set to "1" or the program number need not be specified, the number of repeats and program number can be omitted.

Procedure 3**[COMMON TO P TYPE/Q TYPE]**

- 1 The sequence number is searched for, and the program restart screen is displayed.

Fig4.7 (a)

DESTINATION shows the position where machining is to restart.
 DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart.

BC: The block number

M: 35 most recently specified M codes However, note that only 2-digit M codes are displayed. If M codes comprising three or more digits are displayed, M codes are displayed only up to as far as possible.

T: Two most recently specified T codes

S: Most recently specified S code

B: (second auxiliary function address) Most recently specified second auxiliary (in this case, a B code)

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

- 2 Turn the program restart switch OFF.
- 3 Check the screen for M, S, T and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T and B functions. After execution, restore the previous mode.
 These codes are not displayed on the program restart screen.
- 4 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether or not there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If there is this possibility, move the tool manually to a position from where the tool can move to the machining restart position without hitting any objects.
- 5 Press the Cycle Start button. The moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter setting No. 7110. Machining is then restarted.

Procedure for Program Restart by Specifying a Block Number

Procedure 1

[P TYPE]

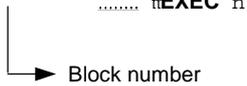
- 1 Retract the tool and replace it with a new one. When necessary, change the offset. (Go to step 2.)

[Q TYPE]

- 1 When power is turned ON or emergency stop is released, perform all necessary operations at that time, including the reference position return.
- 2 Move the machine manually to the program starting point (machining start point), and keep the modal data and coordinate system in the same conditions as at the machining start.
- 3 If necessary, modify the offset amount.

Procedure 2

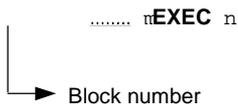
[P TYPE]



- 1 Turn the program restart switch on the machine operator's panel ON.
- 2 Press ...  key to display the desired program.
- 3 Find the program head. Press function ... key. 
- 4 Press the [PROGRAM RESTART] soft key then [(P_BLK#)] soft key, enter the block number of the block you want to restart, then press the [EXEC] soft key. (Go to step 3.)

[Q TYPE]

QB



- 1 Turn the program restart switch on the machine operator's panel ON.
- 2 Press ...  key to display the desired program.
- 3 Find the program head. Press function ...  key.
- 4 Press the [PROGRAM RESTART] soft key then [(Q_BLK#)] soft key, enter the block number of the block you want to restart, then press the [EXEC] soft key. (Go to step 3.)

Procedure 3

[COMMON TO P TYPE/Q TYPE]

- 1 The Block number is searched for, and the program restart screen is displayed.

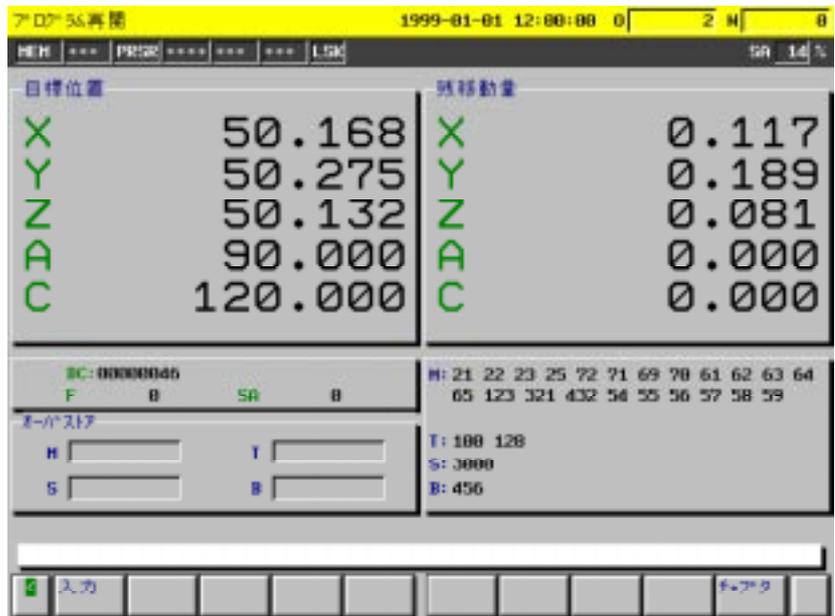


Fig.4.7 (b)

DESTINATION shows the position where machining is to restart.
 DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart.

BC:The block number

M: 35 most recently specified M codes However, note that only 2-digit M codes are displayed. If M codes comprising three or more digits are displayed, M codes are displayed only up to as far as possible.

- T: Two most recently specified T codes
- S: Most recently specified S code
- B: (second auxiliary function address) Most recently specified second auxiliary (in this case, a B code)

Codes are displayed in the order in which they are specified. All codes are cleared by a program restart command or cycle start in the reset state.

- 2 Turn the program restart switch OFF.
- 3 Check the screen for M, S, T and B codes to be executed. If they are found, enter the MDI mode, then execute the M, S, T and B functions. After execution, restore the previous mode.
These codes are not displayed on the program restart screen.
- 4 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether or not there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If there is this possibility, move the tool manually to a position from where the tool can move to the machining restart position without hitting any objects.
- 5 Press the Cycle Start button. The moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter setting No. 7110. Machining is then restarted.

Explanation

£ Block number

When the CNC is stopped, the number of executed blocks is displayed on the program screen or program restart screen. The operator can specify the number of the block from which the program is to be restarted, by referencing the number displayed on the CRT. The displayed number indicates the number of the block that was executed most recently. For example, to restart the program from the block at which execution stopped, specify the displayed number, plus one.

The number of blocks is counted from the start of machining, assuming one NC line of a CNC program to be one block.

< Example 1 >

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G01 X100. F100 ;	3
G03 X01 -50. F50 ;	4
M30 ;	5

< Example 2 >

CNC Program	Number of blocks
O 0001 ;	1
G90 G92 X0 Y0 Z0 ;	2
G90 G00 Z100. ;	3
G81 X100. Y0. Z120. R-80. F50. ;	4
#1=#1+1 ;	4
#2=#2+1 ;	4
#3=#3+1 ;	4
G00 X0 Z0 ;	5
M30 ;	6

Macro statements are not counted as blocks.

£ Storing / clearing the block number

The block number is held in memory while no power is supplied. The number can be cleared by cycle start in the reset state.

£ Block number when a program is halted or stopped

The program screen usually displays the number of the block currently being executed. When the execution of a block is completed, the CNC is reset, or the program is executed in single-block stop mode, the program screen displays the number of the program that was executed most recently.

When a CNC program is halted or stopped by feed hold, reset, or single-block stop, the following block numbers are displayed:

Feed hold : Block being executed

Reset : Block executed most recently

Single-block stop : Block executed most recently

For example, when the CNC is reset during the execution of block 10, the displayed block number changes from 10 to 9.

£ MDI intervention

When MDI intervention is performed while the program is stopped by single-block stop, the CNC commands used for intervention are not counted as a block.

£ Status display

During the time from the specification of program restart until return along the last axis is completed, "PRSR" appears on the status line of the screen.

Limitation

£ P-type restart

Under any of the following conditions, P-type restart cannot be performed:

- When automatic operation has not been performed since the power was turned on
- When automatic operation has not been performed since an emergency stop was released
- When automatic operation has not been performed since the coordinate system was changed or shifted (change in an external offset from the workpiece reference point)

For the block for which the coordinate system was set or changed last before machining interruption, as well as for subsequent blocks, return can be made correctly by P-type program restart.

£ Restart block

The block to be restarted need not be the block which was interrupted; operation can restart with any block. When P-type restart is performed, the restart block must use the same coordinate system as when operation was interrupted.

£ During search

Search operation must be performed under the same conditions including input signal and offset conditions as those used for machining. Otherwise, return to the correct machining restart position cannot be made. Note that even when the single-block switch is set to ON, search operation continues.

£ Single block

When single block operation is ON during movement to the restart position, operation stops every time the tool completes movement along an axis. When operation is stopped in the single block mode, MDI intervention cannot be performed.

£ Manual intervention

During movement to the restart position, manual operation can be performed for an axis along which a return operation has not yet been completed. In this case, return to the restart position can be performed by the subsequent return operation. When manual

operation is performed for an axis after the tool completes a return operation along that axis, however, return operation is not performed again along that axis.

£ **MDI**

After a search is completed, axis movement by MDI cannot be specified before axes are moved.

£ **Reset**

Never reset during the time from the start of a search at restart until machining is restarted. Otherwise, restart must be performed again from the first step.

£ **Feed hold**

When a feed hold has been applied during a search, repeat the restart operation from the beginning.

£ **Manual absolute**

Regardless of whether machining has started or not, manual operation must be performed when the manual absolute mode is on.

£ **Reference position return**

If no absolute-position detector (absolute pulse coder) is provided, be sure to perform reference position return after turning on the power and before performing restart.

£ **Program restart switch**

When the program restart switch is ON, pressing the Cycle Start button does not start operation.

£ **Macro statement, macro call and subprogram call blocks**

Macro statement, macro call and subprogram call blocks cannot be searched even if they are prefixed with a sequence number. In this case, try searching the previous block.

Alarm

Number	Message	Contents
SR0592	END OF RECORD	<p>The EOR (End of Record) code is specified in the middle of a block.</p> <p>This alarm is also generated when the percentage at the end of the NC program is read.</p> <p>This alarm is also generated when the specified block is not found by the program restart function.</p>
SR0600	PARAMETER OF RESTART ERROR	<p>An illegal value is set to parameter No. 7110 that specifies the order in which axes move when machining is restarted in the dry run.</p> <p>The setting range is 1 to the number of controlled axes.</p>

CAUTION

As a rule, the tool cannot be returned to a correct position under the following conditions.

Special care must be taken in the following cases since none of them cause an alarm:

£ Manual operation is performed when the manual absolute mode is OFF.

£ When operation is performed with the machine locked and Z-axis ignored.

£ When the mirror image function is used. For the block in which the mirror image function was turned on or off most recently, as well as for subsequent blocks, P-type return can be performed. In this case, the mirror image signal must be held in the same state as that observed at the time of interruption.

£ When no coordinate system is set at the beginning of a program in which major commands are in incremental mode.

£ When manual operation is performed in the course of axis movement for returning operation.

£ When the program restart is commanded for a block between the block for skip cutting and subsequent absolute command block.

£ The machine lock state is released after the command for restarting the program is issued in that state.

£ When a coordinate system is set, changed, or shifted after search operation terminates, the tool cannot be returned to the correct position.

CAUTION

Notes on restarting the program for which macro variables are used:

£ Common variables

The previously specified values are used as the common variables when restarting the program. The variables are not automatically preset. The necessary common variables are therefore initialized to the values used for the previous automatic operation immediately before the program is restarted.

£ DI/DO

While the program is restarted, DI data can be read from the corresponding system variables, but DO data cannot be output.

£ Clock

While the program is restarted, the clock time can be checked with the system variable, but the time cannot be preset.

£ Tool offset and offset from the workpiece reference point

While the program is being restarted, offsets can be read from the corresponding system variables, but offsets cannot be changed except for programs of type Q.

4.8 OUTPUT OF PROGRAM RESTART M, S, T AND B CODES

The following operations are possible after searching the program execution restart block.

- 1 Before moving to the machining restart position
 - (1) The final M, S, T and B codes can be automatically output to the PMC. If the final S code is specified to the same block containing an G92, that S code is output as the maximum spindle speed. Other S codes are output as the programmed spindle speed. However, note that only the final programmed S code is displayed in the program restart screen regardless of whether it is specified in the same block as G92.
 - (2) All sampled M codes and the final S, T and B codes can be automatically output to the PMC during searching of the program execution restart block. However, note that only up to 35 M codes can be sampled. If the number of M codes to be sampled exceeds 35, the latest 35 M codes are output to the PMC.

Whether (1) or (2) is to be performed is switched by bit 6 (BOA) parameter No. 7620.

- 2 Until arrival at the machining restart position
You can specify M, S, T and B codes in the program restart screen from the MDI panel in the MEMORY or TAPE modes.

Output of final M, S, T and B codes

If the Cycle Start button is pressed after the program execution restart block is searched when bit 3 (MOP) of parameter No. 7620 is set to "1", the final M, S, T and B codes are automatically output to the PMC before the tool moves to the machining restart position.

If the Cycle Start button is pressed again after the final M, S, T and B codes are output when the machine tool is in a single block halt state, the tool moves to the machining restart position.

Output of all M codes and final S, T and B codes

If the Cycle Start button is pressed after the program execution restart block is searched when bit 6 (MOA) of parameter No. 7620 is set to "1", all sampled M codes and final S, T and B codes are automatically to the PMC before the tool moves to the machining restart position.

<Example>

When codes M10, M11, M12, M13, M14, T0101, S1000 and B10 have been sampled, the program is executed in the following format before the tool moves to the machining restart position.

```
M10 T0101 S1000 B10 ;  
M11 ;  
M12 ;  
M13 ;  
M14 ;
```

Outputting the M, S, B codes on the program restart screen

When the MOP bit (bit 3 of parameter 7620) = 1, enter M, S, and B codes from the MDI in the MEM or TAPE mode until the tool completes moving to the machining restart position after searching for the block to be restarted with the program restart function. The program restart function then outputs these codes to the PMC.

Operation

- 1 Search for the block to be restarted with the program restart function. The program restart screen is displayed. When bit 3 (MOP) of parameter No. 7620 is set to 1, the [OVERSTORE] and [CLEAR] soft keys appear.

Fig.4.8 (c)

- 2 When the [OVERSTORE] soft key is pressed before the tool reaches the machining restart position, overstore mode is set, allowing the input of the displayed M, S, T, and B codes in the overstore field on the screen.

To set the overstore mode while the tool is moving to the machining restart position, temporarily stop the restart operation with the feed hold function, then press the [OVERSTORE] soft key. Enter the M, S, T, and B codes to be output in the overstore field from the MDI.

Example To enter M10, S1000, and B20 in the overstore field

- 1 Enter M10 from the MDI.
- 2 Press the INPUT key.

S, T, and B codes can also be entered by performing steps 1 and 2.

Fig. 4.8 (d)

- 3 After the codes have been entered in the overstore field, press the cycle start button. The codes in the overstore field are output to the PMC, then they are cleared from the field.

- 4 To clear the M, S, T, B codes entered in the overstore field, press the [CLEAR] soft key. All the entered codes are cleared.
- 5 To cancel the overstore mode, press the [OVERSTORE] soft key or reset key in the overstore mode.
After the OVERSTORE soft key or reset key is pressed.
- 6 To continue the restart operation, cancel the overstore mode and press the cycle start button.

CAUTION

- 1 The M, S, T, and B codes specified in overstore mode are not displayed on the program restart screen.
- 2 When the operation mode is changed to a mode other than the MEM or TAPE mode in the overstore mode, the overstore mode is not canceled, but no code can be entered in the overstore field.

4.9 Block Restart

The block restart function makes it possible to restart automatic operation at the start point or another point of the block in which automatic operation was interrupted due to an error such as tool breakage.

Restart block procedure

Be sure to perform the following before you restart execution of a block. The "block return switch" in the following description refers to the switch on the machine operator's panel. For details of actual operation, refer to the manual published by the machine tool builder.

Procedure 1

- 1 Error occurs.
- 2 Press the Feed Hold button or change the operation mode to halt automatic operation.
- 3 Manually retract the tool, and carry out the necessary operation.
- 4 If necessary, reset the tool offset value. (Go to procedure 2.)

NOTE

A block restart cannot be performed if the Reset button or Emergency Stop button has been pressed to enter the reset state.

Procedure 2

[When the block return switch is ON]

- 1 Turn the block return switch ON in the jog feed mode. (The letters "RSTR" blink on the status line on screen.)
- 2 Move the tool towards the start point of the block that was interrupted by jog feed. The tool automatically halts when it arrives at the start point. (If the machine tool is in the cutter compensation mode, a vector is made at the start point at right angles to the tool path programmed in the interrupted block, and the tool returns to the tip of that vector. (Go to procedure 3.)

[When the block return switch is OFF]

- 1 Manually move the tool as close as possible to the point where automatic operation was interrupted. (Make sure that the tool does not hit a workpiece or other objects.) (Go to procedure 3.)

Procedure 3

- 1 Return the mode selection switch to the mode that was active before automatic operation was interrupted. The block return switch turns ON if it was OFF. (The letters "RSTR" may be displayed or blink on the status line of the screen depending on the machining conditions.)
- 2 Press the Cycle Start button.
- 3 The cycle start lamp lights, and automatic operation is restarted. Then, the block return switch is turned OFF.

Explanation

£ Manual operation with the block return switch turned on

Move the tool in the continuous manual feed mode toward the start point of the interrupted block while the block return switch is on. The tool automatically stops at the start point. When the start point is reached for all axes, the message RSTR disappears from the bottom of the screen.

If the block return switch is turned on in the continuous manual feed (JOG) mode, the distance between the current position of the tool and the start point of the interrupted block is calculated. The start point of the interrupted block refers to the point calculated as follows:

Absolute coordinate of the programmed start point

+

Previously compensated tool value

+

Newly calculated tool length compensation value

=

Newly calculated cutter compensation value perpendicular to the interrupted block at the start point

If the tool offset value is changed, new compensation values (vectors) for the tool length compensation (G43 and G44) for the fixed compensation axis (bit 4 (LXY) of parameter No. 6000 is set to 0) and the cutter compensation (G41 and G42) are calculated based on the new offset value. However, the compensation values for the tool length compensation for an arbitrary compensation axis (bit 4 (LXY) of parameter No. 6000 is set to 1) and the tool offset (G45 to G48) remain unchanged.

The calculated distance to the start point is displayed by pressing the [BLOCK RESTR] soft key on the position display screen. Normally,

when the block return switch is turned on in jog feed (JOG) mode, this screen (called the restart display) appears automatically. On the restart display screen, the following values are indicated.

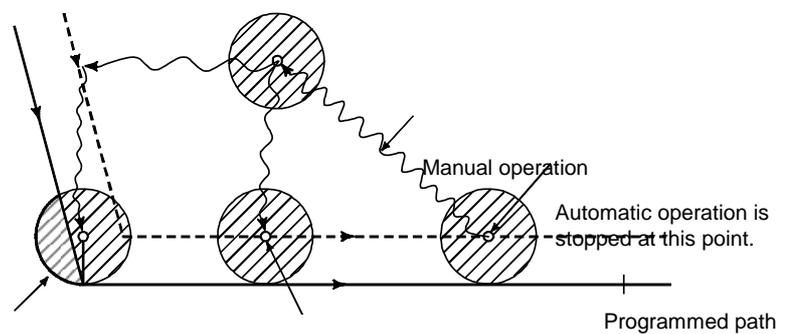
Fig. 4.9 (e)

- (i) (DESTINATION): Absolute coordinates of the tool center at the start point of the interrupted block
- (ii) (DISTANCE TO GO): Distance from the current position (ABSOLUTE) of the tool to the start point (DESTINATION) of the interrupted block
- (iii) Status of RSTR display

Various conditions (mode, cutter compensation, etc.)		RSTR is displayed continuously.
Continuous manual feed (JOG) mode		RSTR blinks.
Automatic operation (MEMORY, TAPE, or MDI)	When the outer corner is being compensated in the cutter compensation mode.	RSTR blinks.
	Other than the above	RSTR is displayed continuously.
Reset state		RSTR does not appear regardless of what the current mode is.

CAUTION

- 1 When returning the tool to the start point of the block by a manual operation using the block restart function, the tool is returned to the tip of the vector vertical to the block at its start point. Therefore, if the start point of the interrupted block is inside the corner, the workpiece may be cut when the tool returns to the start point.



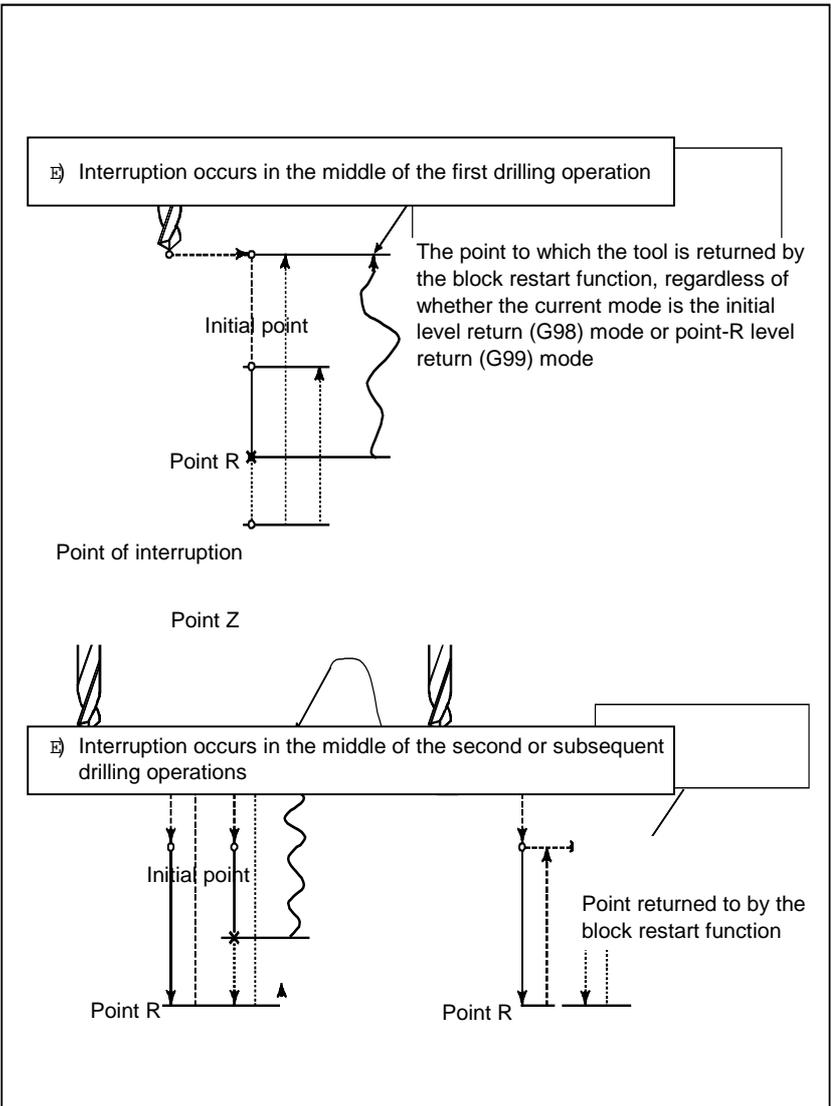
When the tool moves to the start point, it may cut the workpiece

The tool should be returned manually to an arbitrary point in the block.

In this case, return the the tool to an appropriate position while watching it, rather than to the start point.

CAUTION

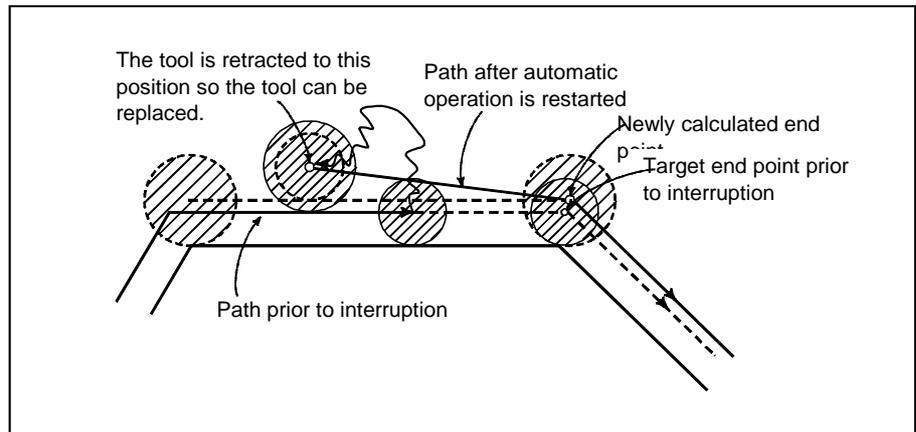
- 2 When the tool movement is interrupted during a canned drilling cycle, after which block restart is performed, the tool does not return to the point at which tool movement was interrupted as shown in the figures below. In either case, care must be exercised since no alarm is issued.



On the second or subsequent drilling cycle, the tool returns to either of the above two positions according to the G98 or G99 mode.

£ Restart of automatic operation with the block return switch turned on

If the cycle start button is pressed while the block return switch is on, the tool length compensation value and the cutter compensation value



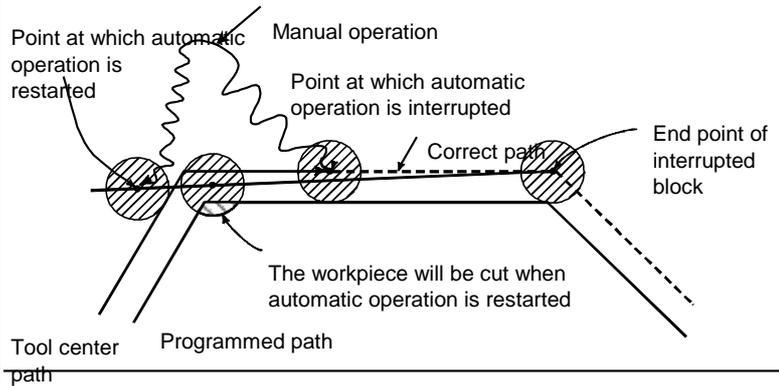
are recalculated, and automatic operation is restarted toward the compensated end point from the current position.

Fig. 4.9 (f)

The tool moves toward the recalculated end point from its current position regardless of whether it has been returned to the start point of or an arbitrary point in the interrupted block prior to pushing the cycle start button.

CAUTION

The tool may cut the workpiece when it is being moved toward the end point of the interrupted block. In this case, an alarm is displayed and the the tool is inhibited from moving to the end point. This function is referred to as the block restart interference check.



If the tool will cut the workpiece when automatic operation is restarted, press the cancel key to reset the alarm. Move the tool to an appropriate position while observing the movement of the tool. Then, perform the block restart operation again from step 1 of procedure 3.

£ Block restart interference check

The block restart interference check is made based on the value (d: permissible block restart interference check value) set in parameter No. 7651. The interference check issues an alarm unless the distance between the normal path and the block restart point is within d.

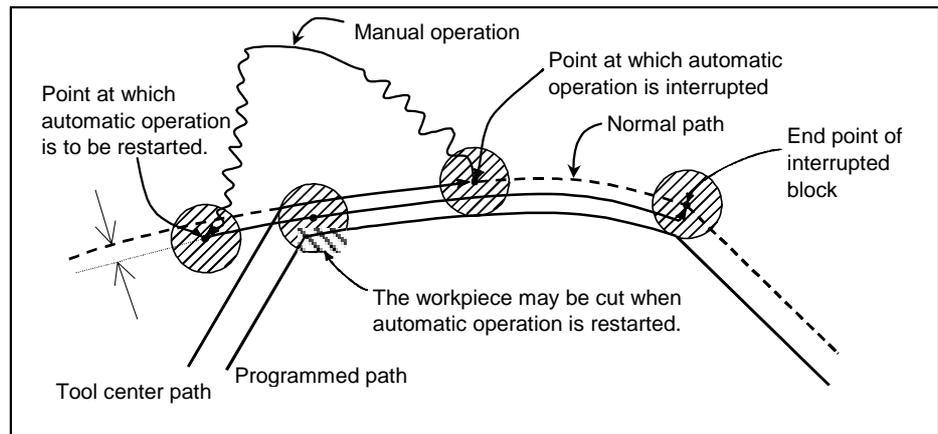


Fig. 4.9 (g)

The block restart interference check function can be enabled and disabled using bit 7 of parameter No. 7605. However, the interference check is not performed in the following cases even if it has been enabled:

- When the tool is moved manually to the start point of the interrupted block while the block return switch is on.
- When the interrupted automatic operation is restarted with the cutter compensation mode canceled.
- When no vector is available for cutter compensation at the start point of the block in which the tool movement was interrupted. (Start up of cutter compensation)

Limitation

£ Automatic reference position return

The command for automatic reference point return (G28 and G30) in the cutter compensation mode does not return the tool to the correct position even when the restart operation is performed within the block.

£ Dwell

The restart operation cannot be performed in the block for which a dwell (G04) has been specified.

£ Setting a compensation value

The compensation value must be set prior to turning on the block return switch.

£ In three-dimensional cutter compensation mode

In the three-dimensional cutter compensation mode with zero value, suppose the tool movement is interrupted, the compensation value is changed, and the operation is restarted. In this case, the vector for the cutter compensation is not recalculated.

Note that if the compensation values are changed excessively, the accuracy for vector calculation may deteriorate.

£ Cycle start ignored

When the block return switch is on, if the message RSTR does not appear on the CRT screen, the cycle start command is ignored.

£ When the block return switch status is changed

Operating the block return switch during continuous manual feed stops the tool.

£ Block containing no move command

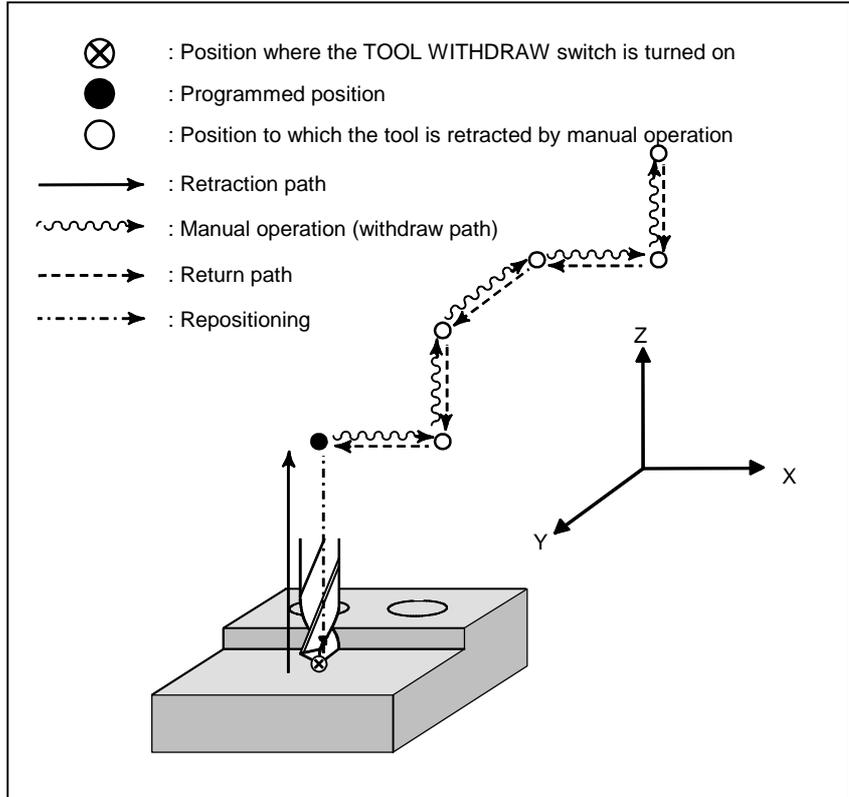
Operation must not be restarted in a block without a move command.

£ Restart after single-block stop

After a single-block is stopped, operation is restarted at the block after the block that has already been executed by the single-block function.

4.10 TOOL WITHDRAWAL AND RETURN

The tool can be withdrawn from a workpiece in order to replace the tool when it is damaged during machining, or merely to check the status of machining. The tool can then be advanced again to restart

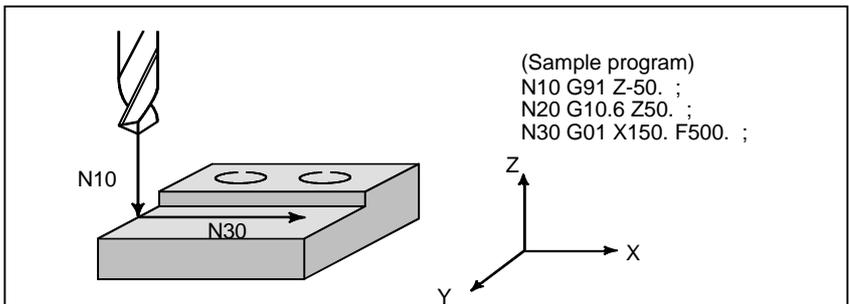


machining efficiently.

Procedure for tool withdrawal and return

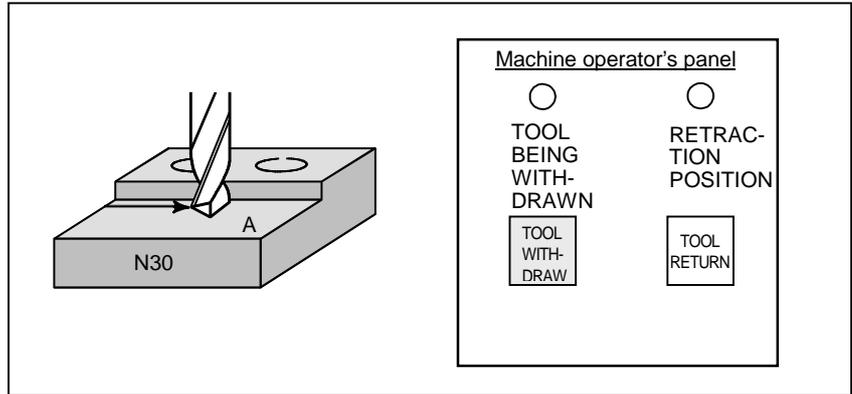
Procedure1 Programming

Specify a retraction axis and distance in command G10.6IP_ beforehand. In the sample program below, the N20 block specifies that the Z-axis is the retraction axis and the retraction distance is to be 50 mm.

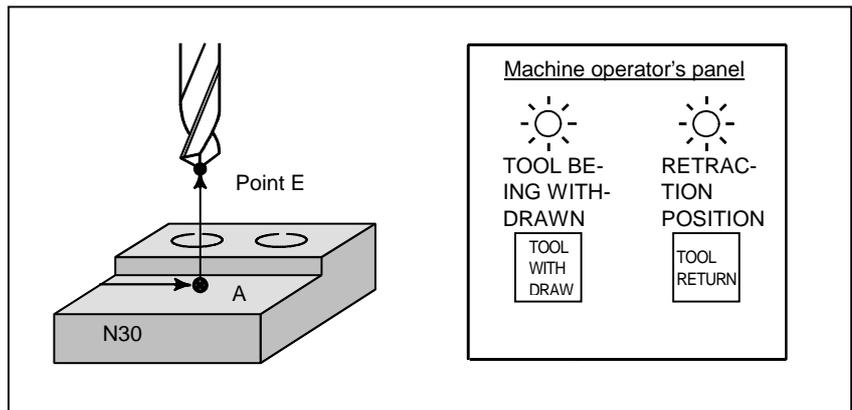


Procedure2 Retract

Suppose that the TOOL WITHDRAW switch on the machine operator's panel is turned on when the tool is positioned at point A during execution of the N30 block.



Next, the tool withdrawal mode is set and the TOOL BEING WITHDRAWN LED goes on. At this time, automatic operation is temporarily halted. The tool is then retracted by the programmed distance. If point A is the end point of the block, retraction is performed after automatic operation is stopped. Retraction is based on linear interpolation. The dry run feedrate is used for retraction. Upon completion of retraction, the RETRACT POSITION LED on the operator's panel goes on.



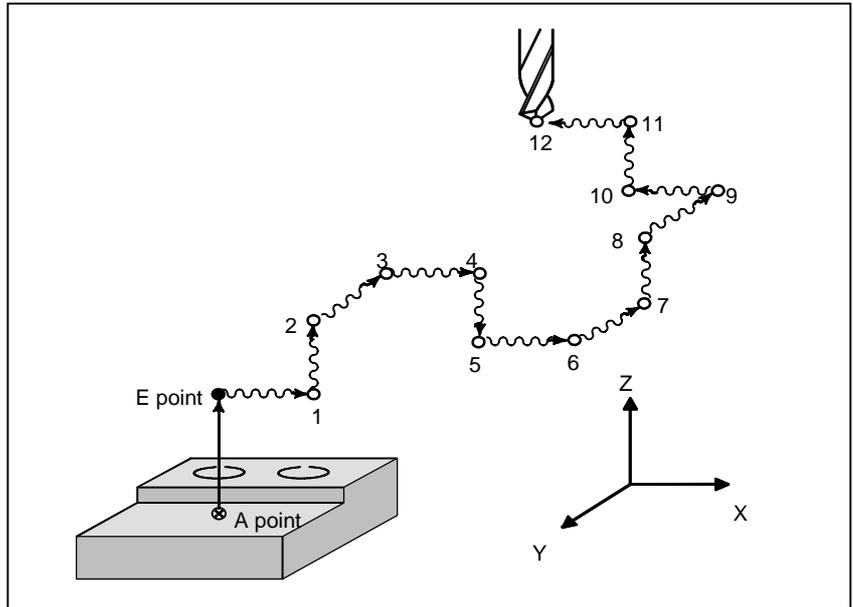
During retraction, the screen displays "PTRR" and "STRT".



- "PTRR" blinks in the field for indicating states such as the program editing status.
- "STRT" is displayed in the automatic operation status field.
- "MTN" is displayed in the field for indicating status such as movement along an axis.

Procedure3 Withdrawal

Set the manual operation mode, then withdraw the tool. For manual operation, either jog feed or handle feed is possible.

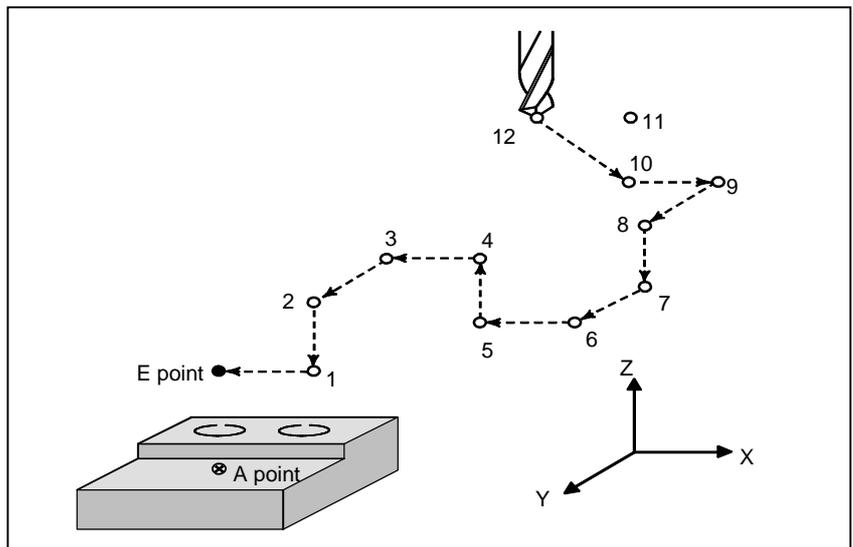


Procedure4 Return

After withdrawing the tool and any additional operation such as replacing the tool, move the tool back to the previous retraction position.

To return the tool to the retraction position, return the mode to automatic operation mode, then turn the TOOL RETURN switch on the operator's panel on then off again. The tool returns to the retraction position at the dry run feedrate, regardless of whether the dry run switch is on or off.

When the tool has returned to the retraction position, the RETRACTION POSITION LED comes on.



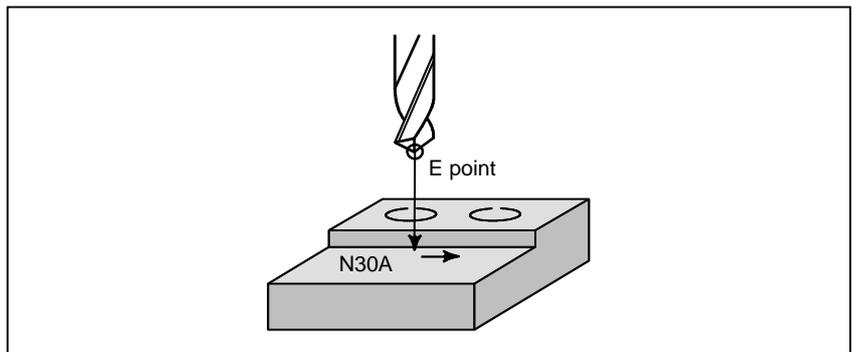
During return operation, the CRT screen displays “PTRR” and “MSTR”.



- “PTRR” blinks in the field for indicating states such as program editing status.
- “MSTR” is displayed in the automatic operation status field.
- “MTN” is displayed in the field for indicating states such as movement along an axis.

Procedure 5 Repositioning

While the tool is at the retraction position (point E in the figure below) and the RETRACTION POSITION LED is on, press the cycle start switch. The tool is then repositioned at the point where retraction was started (i.e. where the TOOL WITHDRAW switch was turned on). The tool is repositioned by linear interpolation at the dry run speed. Upon completion of repositioning, the tool withdraw mode is cancelled,

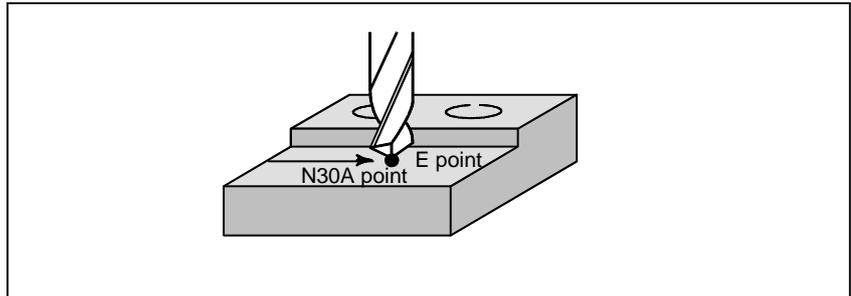


and the TOOL BEING WITHDRAWN LED goes off and restart N30.

Explanation 1 Retraction

£ When no retraction distance is specified

If no retraction distance or direction required for retraction are specified, retraction is not performed when the TOOL WITHDRAW switch on the operator's panel is turned on. Instead, the block being executed in automatic operation is interrupted (automatic operation is held or stopped). In this state, the tool can be withdrawn and



returned.

£ Retraction from the automatic operation hold or stop state

When the single block switch is turned on during automatic operation, or the TOOL WITHDRAW switch is turned on after the automatic operation hold or stop state is set by feed hold: Retraction is performed, then the automatic operation hold or stop state is set again.

£ Stopping retraction

During retraction, feed hold operation is ignored. However, reset operation is enabled (retraction is stopped at reset). When an alarm is issued during retraction, the retraction is stopped immediately.

£ Repositioning immediately after retraction

After retraction is completed, tool repositioning can be started without performing the withdraw and return operations.

Explanation 2 Withdrawal

£ Axis selection

To move the tool along an axis, select the corresponding axis selection signal. Never specify axis selection signals for two or more axes at a time.

£ Path memorization

When the tool is moved in manual operation along an axis, the control unit

memorizes up to ten paths of movements. If the tool is stopped after being moved along a selected axis and is then moved along another selected axis, the position where this switch takes place is memorized. After ten paths have been memorized, the control unit does not memorize any additional switching points.

£ Reset

Upon reset, memorized position data is lost and the tool withdraw mode is cancelled.

NOTE

Simultaneous automatic/manual operations are impossible in the tool retraction mode. If they are selected, manual operation such as jog feed, incremental feed, or handle feed is not performed in the tool retraction mode.

In the tool retraction mode, if an attempt is made to move the tool along two or more axes at one time using a manual numeric value command, alarm (PS0183) is issued.

Explanation 3 Return

£ Return path

When there are more than ten return paths, the tool first moves to the tenth position, then to the ninth position, then to the eighth position, and so forth until the retraction position is reached.

£ Single block

The single block switch is enabled during return operation. If the single block switch is turned off, continuous return operation is performed. If the single block switch is turned off, the tool stops at each memorized position. In this case, return operation can be resumed by turning the TOOL RETURN switch on then off again.

£ Interruption of return operation

When an alarm is issued during return operation, return operation stops.

£ Feed hold

The feed hold function is enabled during return operation.

Explanation 4 Repositioning

£ Feed hold

The feed hold function is disabled during repositioning.

£ Operation after completion of repositioning

The operation after completion of repositioning depends on the automatic operation state present when the TOOL WITHDRAW switch is turned on.

1. When automatic operation is being started
After completion of repositioning, the interrupted execution of the block is resumed.
2. When automatic operation is held or stopped
After completion of repositioning, the tool stops once at the repositioned point, then the original automatic operation hold or stop state is set. When the cycle start switch is pressed, automatic operation is resumed.

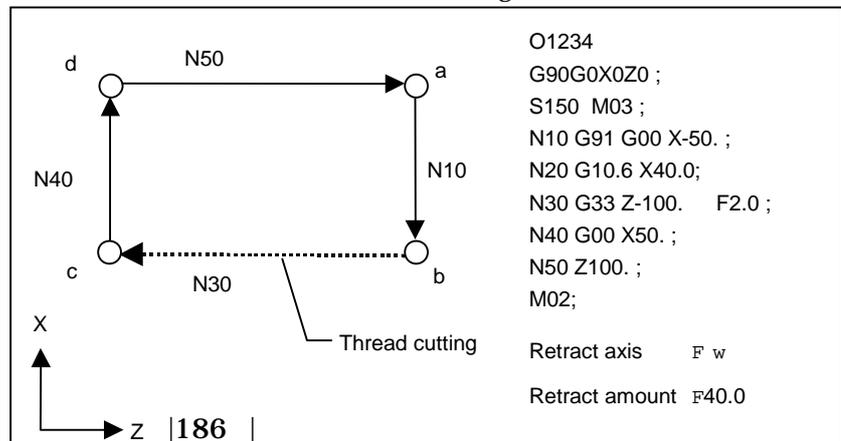
Explanation 5 Tool retract and return after thread cutting

£ Difference with regular command

1. Retract moves the tool off the workpiece between the retract-specified axis and the thread cutting axis.
2. After the tool is retracted, a single block not containing the thread cutting command is executed and program execution is halted.
3. When the thread cutting long axis is specified as the retract axis, retract is not performed even by pressing the tool retract switch. After the block not containing the thread cutting command is executed, alarm PS0215 is generated and tool movement halts.
4. During repositioning, the tool is returned to the program execution position in the first block not containing the thread cutting command.

£ Operation procedure

1. Specify the axis along which the tool is retracted and the retract amount by specifying "G10.6IP- -;" in the program.
2. Turn the tool retract switch ON during execution of blocks to



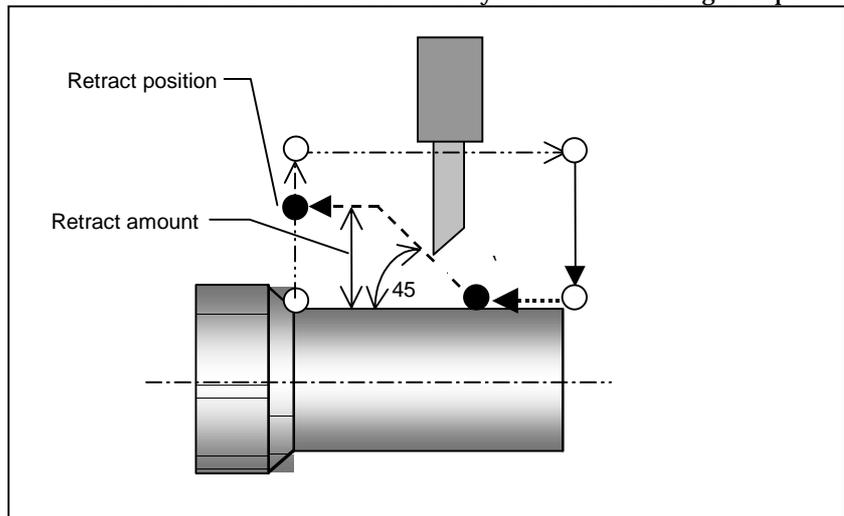
which plus thread cutting is programmed.

3. This sets the control to the tool retract mode, and retract is executed. Tool retract moves the tool off the workpiece at an angle of 45° by the retract amount between the retract axis and the thread cutting long axis.

Retract operation differs as follows according to the relationship between the remaining thread cutting amount and the retract amount when the tool retract switch is turned ON.

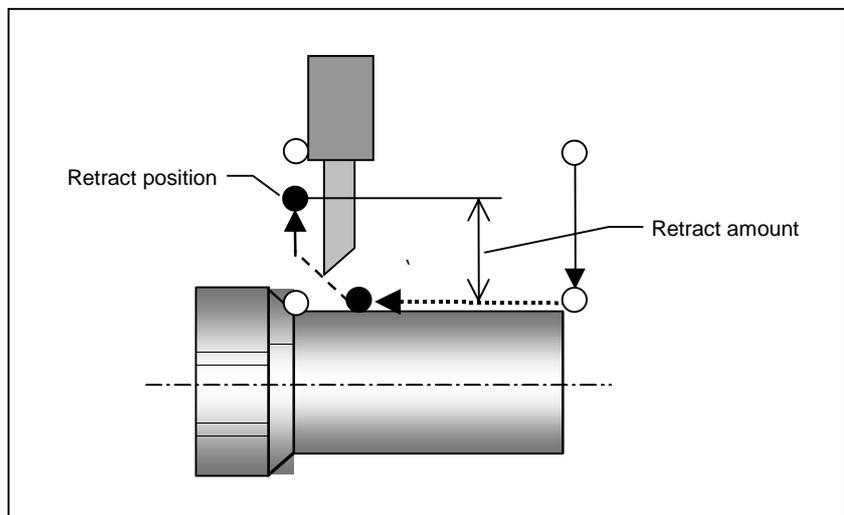
- (1) When the remaining thread cutting amount => retract amount

When the tool does not travel beyond thread cutting end point



(c), where the tool moves off the workpiece at an angle of 45° by the retract amount, the tool moves to the thread cutting end point after being moved off the workpiece.

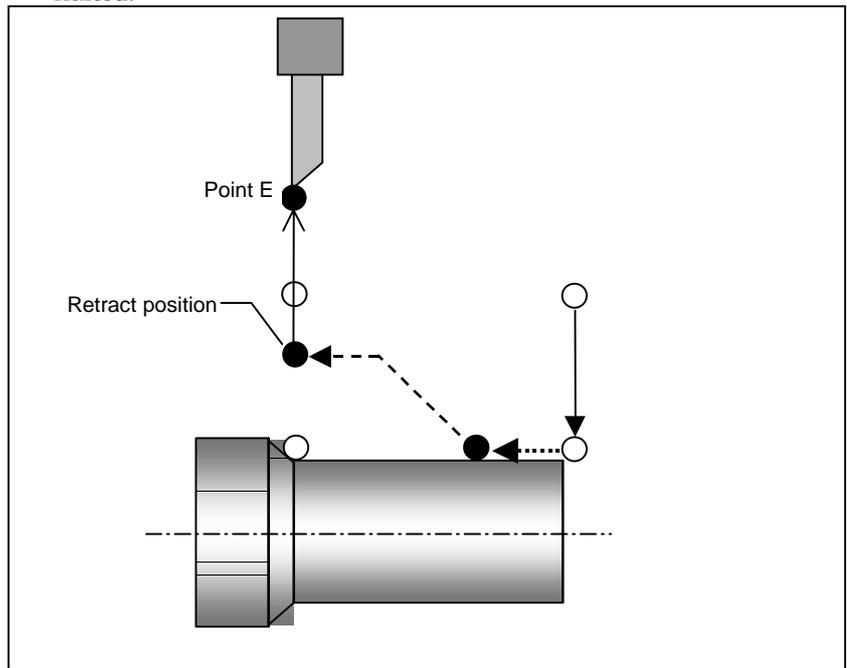
- (2) When the remaining thread cutting amount < retract amount



When the tool travels beyond thread cutting end point (c), where the tool is moved off the workpiece at an angle of 45° by

the retract amount, the retract axis moves to the retract position after arriving at the thread cutting end point.

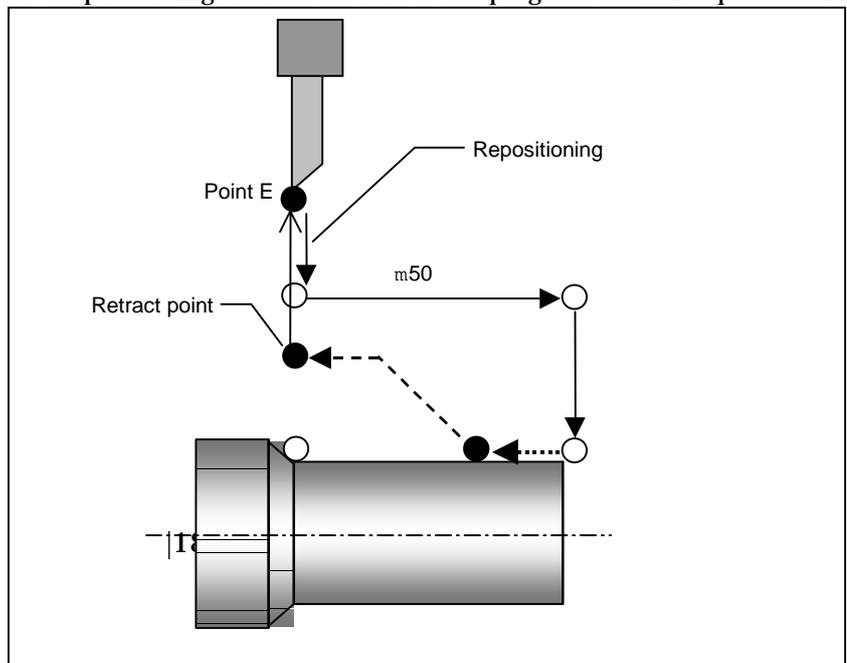
4. After the tool has been retracted, the next block not containing the thread cutting command is executed and program execution is halted.



In this example, as the first block not containing the thread cutting command is incremental X50.0, the tool moves to point E and halts.

When the thread cutting long axis is specified as the retract axis, alarm PS0215 is generated and tool movement halts after the block not containing the thread cutting command is executed without retract being executed.

5. Repositioning returns the tool to the program execution position in

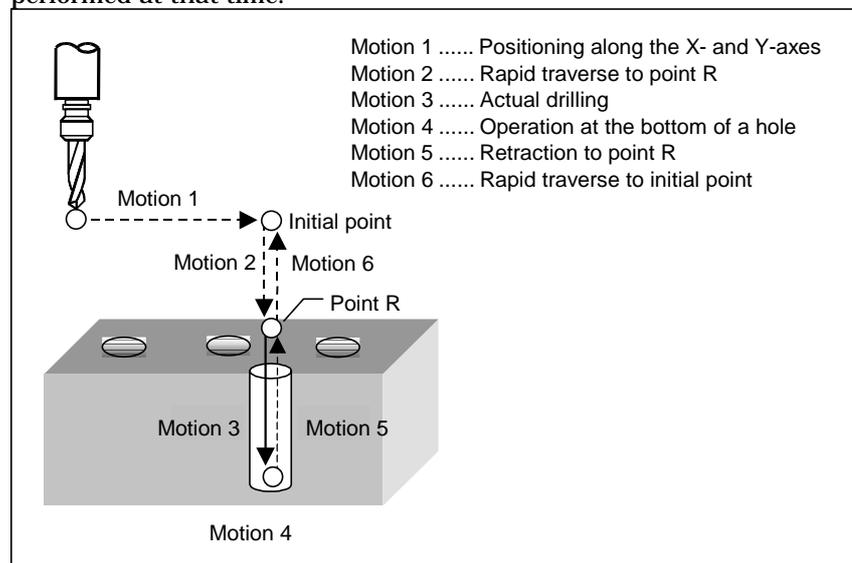


the first block not containing the thread cutting command.
In this example, the positioning point is d. Automatic operation after repositioning is executed from the block containing the N50 command.

Explanation 6 Procedure for retraction and recovery in a canned drilling cycle

£ Retract

When the TOOL RETRACT switch is turned on in a canned drilling cycle, how the tool is retracted depends on the operation being performed at that time.



- (i) If the switch is turned on during motion 1, the tool is moved retracted by the distance specified by G10.6, as in ordinary retraction.
- (ii) If the switch is turned on during motion 2, the motion is suspended and the tool moves to the initial point and stops there.
- (iii) If the switch is turned on during motion 3, the motion is suspended, cycle motions 4, 5, and 6 are executed from that position. After motion 6, the tool stops at the initial point.
- (iv) If the switch is turned on during motion 4, 5, or 6, the motion is continued until the tool reaches the initial point.

During motions 2 to 6, turning on the TOOL RETRACT switch does not retract the tool as specified by G10.6. However, when the switch is turned on and the tool reaches the initial point, the machine enters the tool retraction mode.

When the canned cycle is in its second or subsequent operation, if the switch is turned on during motions 2 to 6, the retraction position differs according to G98 or G99.

- G98 (initial level return): The tool moves to the initial level.

- G99 (point-R level return): The tool moves to point R.

£ Repositioning

Repositioning of the canned cycle is performed by applying a cycle start at the retract position.

1. Repositioning when the tool retract switch is turned ON in procedure 1:

Automatic operation is restarted after repositioning is completed in the same way as regular repositioning.

2. Repositioning when the tool retract switch is turned ON in procedure v2:

The program is re-executed from step 2 of the canned cycle.

3. Repositioning when the tool retract switch is turned ON in procedure 3:

The program is re-executed from step 2 of the canned cycle.

4. Repositioning when the tool retract switch is turned ON in procedure 4:

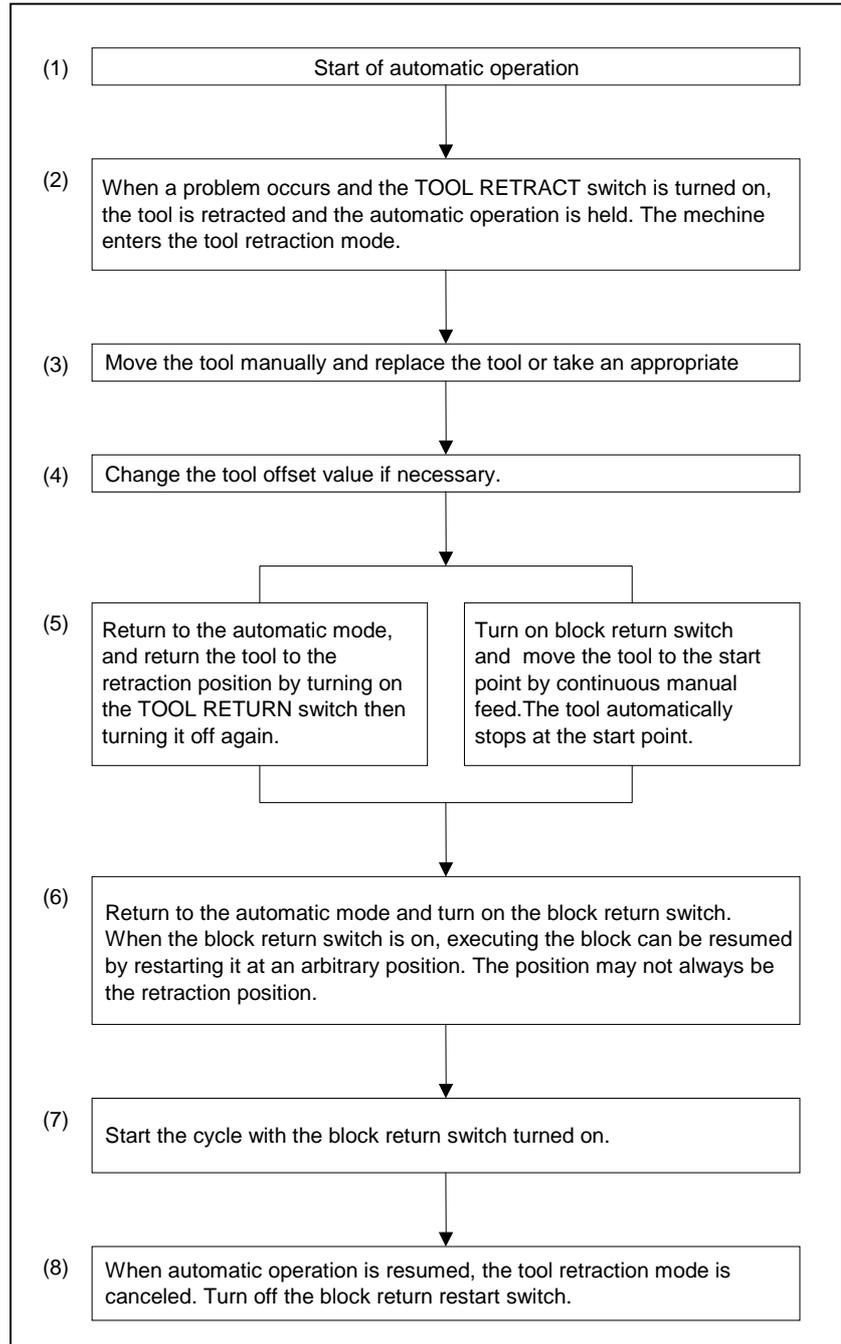
The program is re-executed from step 2 of the canned cycle at the same hole position again.

Explanation 7

Combining the block restart function and the retraction and recovery function

The procedure for combining the tool retraction and recovery and block restart functions is shown following. See Item 5.6.2 for the block restart function.

If the tool breaks, for example, after automatic operation is suspended, the tool is replaced, and the offset value is changed, the



operation is resumed from the start point of or an intermediate point in the interrupted block.

Limitation

1. After the retraction position is specified with G10.6 using an absolute command, any changes in the origin, preset, and workpiece origin offset (or external workpiece origin offset) will not be reflected in the retraction position. If, therefore, these operations are performed or the workpiece origin offset (or external workpiece origin offset) is changed, the retraction position must be re-specified with G10.6.
2. When retracting the tool by manual operation in the tool retraction mode, do not use the machine lock, mirror image, or scaling function.

WARNING

The retraction axis specified by G10.6 and the distance the tool is retracted need to be changed in the necessary block according to the figure to be machined. If correct retraction is not specified, the tool or the workpiece may be damaged.

Alarm

Number	Message	Contents
PS0183	TOO MANY SIMULTANEOUS AXES	A move command was specified for more axes than can be controlled by simultaneous axis control. Either add on the simultaneous axis control extension option, or divide the number of programmed move axes into two blocks.
PS0215	ILLEGAL COMMAND IN G10.6	The retract command was specified in the long axis for threading when retract was started by the threading block.

4.11 RETRACE

The retrace function enables the tool to retrace the path of movement so far. It also returns the tool along the same path once again. When the tool returns to the point where it started retracing its path, it continues machining as directed by program.

The retrace function can be triggered by the trace switch on the machine operator's panel. It can also be automatically triggered (without the operator's intervention) when a damaged tool is detected. Refer to the appropriate manual issued by the machine builder to check what conditions cause the tool to retrace its path and to return to the machining position.

Procedure

£ Forward movement □ Reverse movement

To move the tool in the forward direction, turn off the REVERSE switch on the operator's panel, then press the cycle start switch. If the REVERSE switch on the operator's panel is on, the tool moves in the reverse direction or completes reverse movement.

Three methods are available for moving the tool in the reverse direction along the programmed path.

- 1) When the tool is moving in the forward direction, turn on the REVERSE switch on the operator's panel during block execution.
- 2) When the tool is moving in the forward direction, turn on the REVERSE switch on the operator's panel after a single block stop.
- 3) When the tool is moving in the forward direction, turn on the REVERSE switch on the operator's panel after a feed hold stop.

In the case of 1) above, the tool starts reverse movement after completion of the block currently being executed (after execution up to the position of a single block stop). Turning on the REVERSE switch on the operator's panel does not immediately start reverse movement.

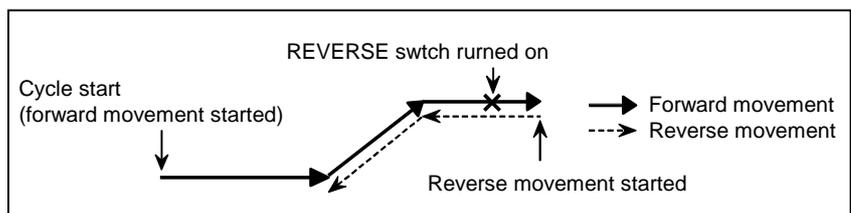


Fig. 4.11 (h)

In the case of 2) above, the tool starts reverse movement at the position of a single block stop when the cycle start switch is pressed.

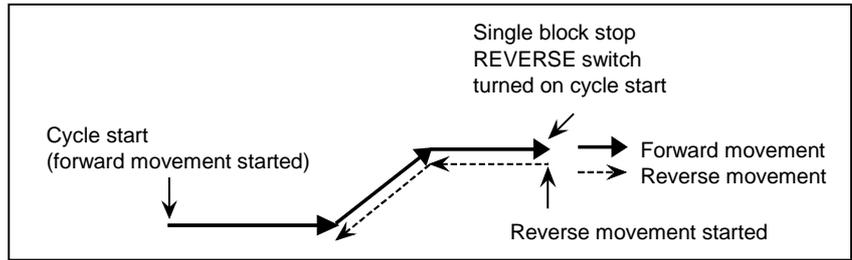


Fig. 4.11 (i)

In the case of 3) above, the tool starts reverse movement at the position of a feed hold stop when the cycle start switch is pressed.

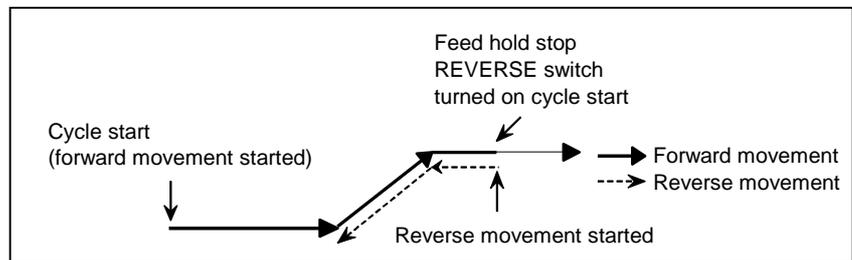


Fig. 4.11 (j)

£ Reverse movement □ Forward return movement

Three methods are available for moving the tool in the forward direction again along the retraced path.

- 1) When the tool is moving in the reverse direction, turn off the REVERSE switch on the operator’s panel during block execution.
- 2) When the tool is moving in the reverse direction, turn off the REVERSE switch on the operator’s panel after a single block stop.
- 3) When the tool is moving in the reverse direction, turn off the REVERSE switch on the operator’s panel after a feed hold stop.

In the case of 1) above, the tool starts forward return movement after completion

of the block currently being executed (after execution up to the position of a single block stop). Turning off the REVERSE switch on the operator’s panel does not immediately start forward return movement.

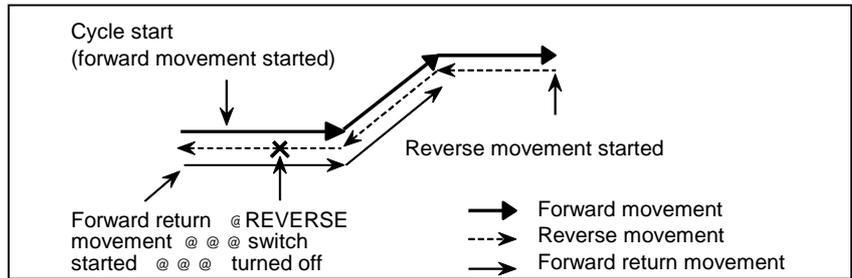
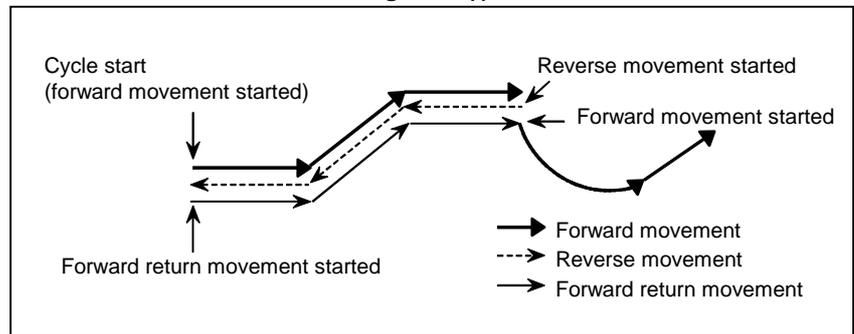


Fig. 4.11 (k)

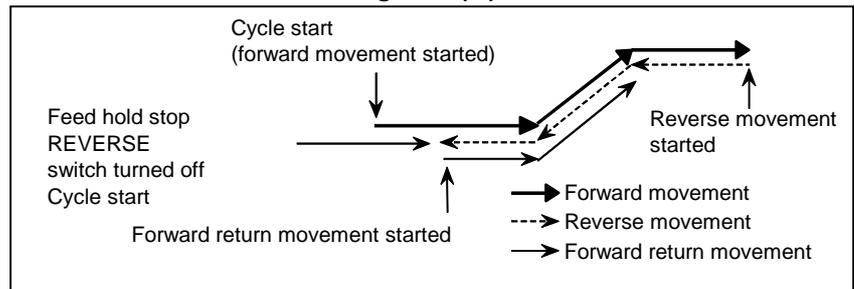
In the case of 2) above, the tool starts forward return movement at the position of a single block stop when the cycle start switch is pressed.

Fig. 4.11 (l)



In the case of 3) above, the tool starts forward return movement at the position of a feed hold stop when the cycle start switch is pressed.

Fig. 4.11 (m)

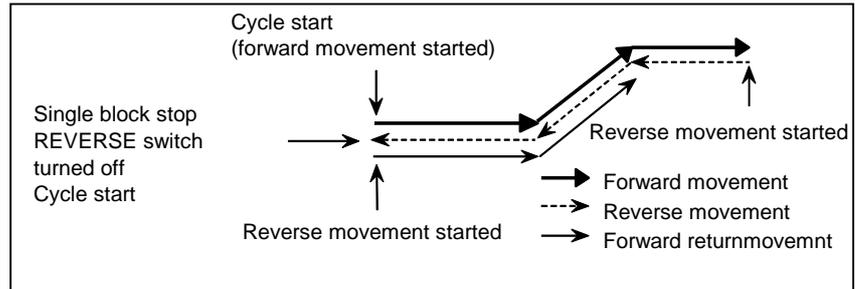


£ Reverse movement □ Reverse movement completion □ Forward return movement

When there are no more blocks for which to perform reverse movement (when the tool has moved back to the initial forward movement block or the tool has not yet started forward movement), the reverse movement completion state is entered and operation stops. Even when the cycle start switch is pressed with the REVERSE switch on the operator's panel turned on, no operation is performed (the reverse movement completion state remains unchanged). When the cycle start switch is pressed after turning off the REVERSE switch on

the operator's panel, the tool starts forward return movement or forward movement.

Fig. 4.11 (n)



£ Forward return movement □ Forward movement

When the tool completes a forward return movement up to the block where reverse movement was started, the tool automatically resumes forward movement. Programmed commands are read and program execution is continued. No particular operation is required to resume forward movement.

When tool movement switches from forward return movement to forward movement, the display of RTRY (Re-TRY) in the lower-right corner of the CRT screen disappears.

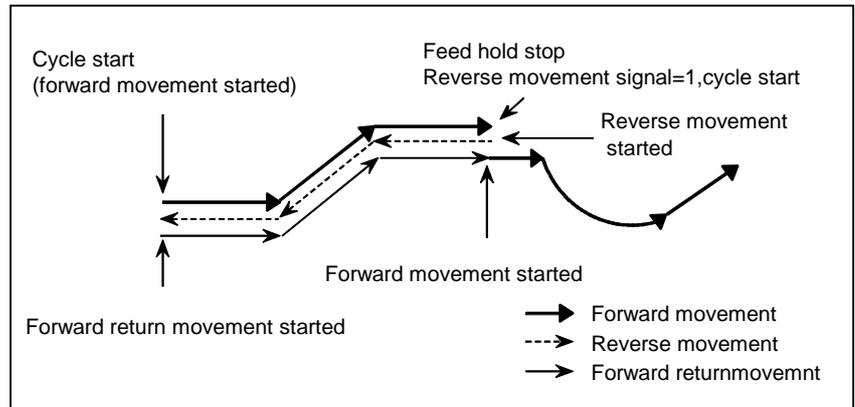


Fig. 4.11 (o)

If the tool moves in the reverse direction after a feed hold stop, the tool stops forward return movement at the position of the feed hold stop, then resumes forward movement. If the tool moves in the reverse direction after a single block stop, the tool also stops forward return movement at the position of the single block stop.

Explanation

£ Forward reverse movement

In automatic operation, a program is usually executed in the order that commands are specified. This mode of execution is referred to as forward movement. The retrace function can execute in reverse, program blocks that have already been executed. This mode of execution is referred to as reverse movement. In reverse movement, the tool can retrace the tool path followed by forward movement. A program can be executed in the reverse direction only for those blocks that have already been executed in the forward direction. The tool can perform reverse movement one block at a time when the single block mode is set.

£ Status display

During reverse movement, the letters "RVRS" blink on screen. During forward return movement, the letters "TRRY" blink on screen to inform the operator that forward return movement is in operation. The "RTRY" display continues until the tool returns to the block where reverse movement was started to start regular operation, in other words, to the position where forward return movement is enabled.

£ Number of blocks permissible on reverse movement

Reverse movement is possible up to about 200 blocks. This limit may be reduced according to the specified program.

£ Reset

Upon reset (when the RESET key on the MDI panel is pressed, the external reset signal is applied, or the reset and rewind signal is applied), the memorized reverse movement blocks are cleared.

£ Feedrate

A feedrate for reverse movement can be specified using parameter (No. 1494). When this parameter is set to 0, the feedrate used for forward movement is used. For forward return movement, the feedrate for forward movement is always used.

In reverse movement and forward return movement, the feedrate override function, rapid traverse override function, and dry run function are enabled.

£ Start of reverse movement and forward return movement after completion of block execution

Reverse movement or forward return movement can be started at blocks containing a rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03), dwell (G04), skip function (G31) or other miscellaneous function command in the automatic operation

mode (memory operation, tape operation, MDI operation). Reverse movement or forward return movement is not possible immediately after the reverse movement signal has been switched. Reverse movement or forward return movement is started when execution of the block is completed, in other words, after completion of tool movement, dwell or miscellaneous functions.

£ **Start of reverse movement and forward return movement after feed hold halt**

If the reverse movement signal is switched to restart program execution after a feed hold is executed during execution of rapid traverse (G00), linear interpolation (G01), circular interpolation (G02, G03) and skip function (G31), reverse movement or forward return movement can be started immediately from that point. However, note that this is not possible during execution of dwell (G04) or miscellaneous functions.

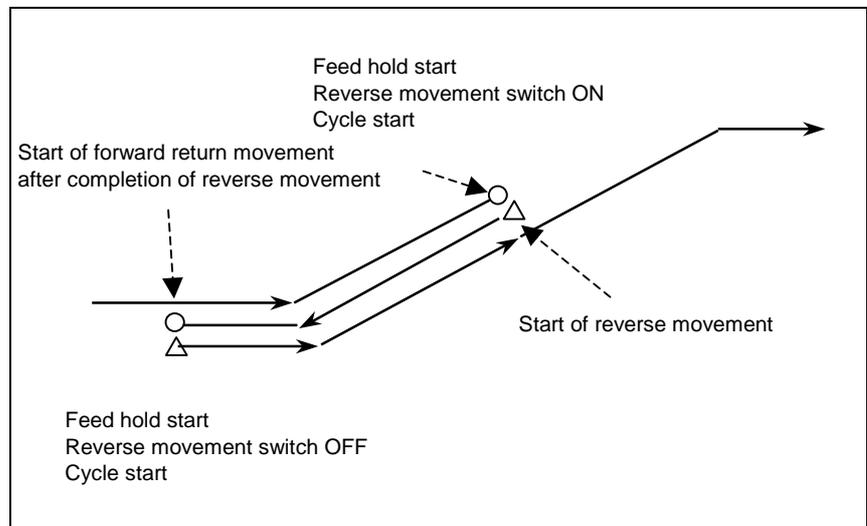


Fig. 4.11 (p)

When reverse movement is started after a feed hold halt, the position from the start point of the current block up to the position where feed hold was halted is taken as a single block. For this reason, the single block is halted if the tool returns to the position where reverse movement is started when forward return movement is performed with the single block switch ON.

£ **Start of reverse movement and forward return movement after single block halt**

Reverse movement or forward return movement can be restarted after a single block halt by switching the reverse movement signal and restarting program execution.

£ **Dwell (G04)**

The dwell command (G04) is executed in reverse movement and forward return movement in the same way as during regular operation.

Limitation

£ Blocks that do not allow reverse movement

Reverse movement cannot be performed for the blocks listed below. When forward movement is stopped by feed hold during the execution of one of these commands or modes, after which an attempt is made to perform reverse movement at that point, reverse movement terminates immediately. When one of the following commands is encountered during reverse movement, the reverse movement terminates at that point, and "RVED" is displayed.

- . Inch/metric conversion
- . Functions related to reference position return
- . Remote buffer
- . Circular thread cutting .
- . Polar coordinate interpolation
- . Cylindrical interpolation
- . Exponential interpolation
- . Involute interpolation
- . Spline interpolation
- . NURBS interpolation
- . Chopping function
- . Rigid tapping cycle
- . Cs contour control
- . Three-dimensional coordinate system conversion

£ Manual intervention

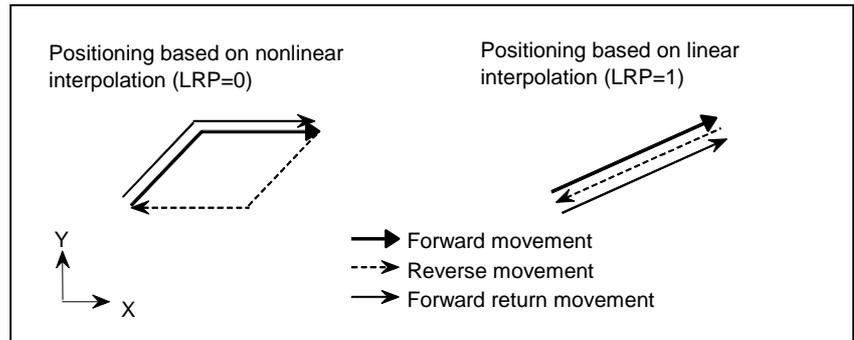
To perform reverse movement after operation is stopped by feed hold or single-block stop, after which manual intervention is made, return the tool to the stop position, then turn on the reverse signal. Movement made by manual intervention is ignored in reverse movement and return movement.

£ Positioning (G00)

When the tool is positioned based on nonlinear interpolation by setting bit 4 (LRP) of parameter No. 1400 to 0, the path of the tool for reverse movement does not match the path for forward movement. The path for forward return movement is the same as the path for forward movement. When the tool is positioned based on linear interpolation by setting bit 4 (LRP) of parameter No. 1400 to 1, the

path of the tool for reverse movement matches the path for forward movement.

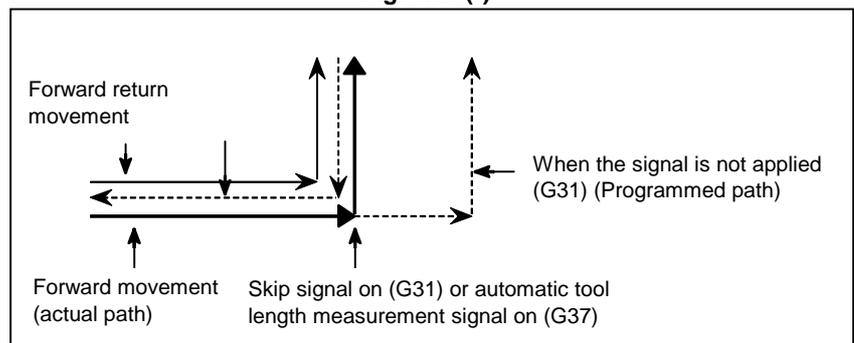
Fig. 4.11 (q)



£ Skip function (G31), automatic tool length measurement (G37)

In reverse movement and forward return movement, the skip signal and automatic tool length measurement signal are ignored. In reverse movement and forward return movement, the tool moves along the path actually followed in forward movement.

Fig. 4.11 (r)



£ Single block halt position

Blocks created internally by the control unit contain a single block when reverse movement is performed.

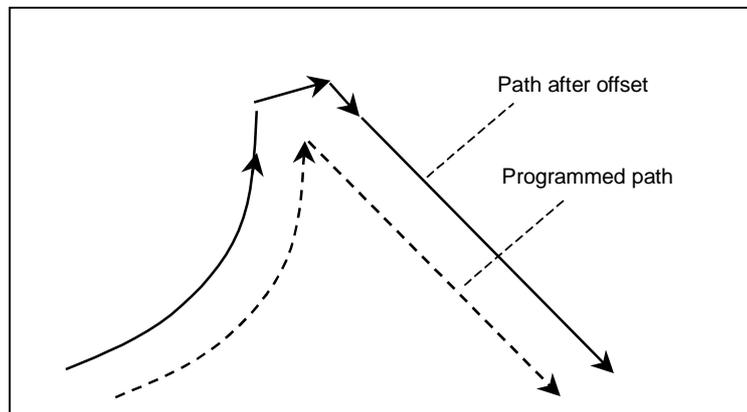


Fig. 4.11 (s) Tool path after cutter compensation

In the above example, axis movement commands for five blocks are actually created in program execution even if two blocks are programmed.

In this case, the single block halt position may differ during reverse movement and forward return movement.

£ Programmable data setting (G10)

A tool compensation value, parameter, pitch error data, workpiece zero point offset value, and tool life management setting specified or modified using the programmable data setting code (G10) are ignored in reverse movement and forward return movement.

£ Coordinate system setting (G92)

The current position of the coordinate system setting (G92) during reverse movement may differ from the current position during forward return movement. Actual machine coordinates, however, do not differ.

£ Mirror image

In blocks where the mirror image function is specified by programmable mirror image codes G50.1 and G50.2, reverse movement is performed along a tool path that is the mirror image of the programmed tool path.

Blocks where the mirror image function is specified by setting data or machine signals are stored to memory without mirror image being applied. Mirror image specified by setting data or machine signals is effective during both reserve movement and forward return movement. For this reason, the mirror image ON/OFF state specified by setting data or machine signals must be maintained at the same state as forward movement also during reserve movement and forward return movement.

£ Changing the tool offset value

Even if the cutter radius or length offset values are changed during reverse movement and forward return movement, the changes to the tool offset values are not effective until reverse movement and regular operation is returned to. The tool then moves by the tool offset value that was active when reserve movement of that block was first performed.

£ Interrupt-type custom macro

1. Never initiate an interrupt during reverse movement.

2. Never perform reverse movement for an interrupted block and the program that has issued the interrupt.

£ Tool life management

The tool life is not counted by tool life management during reverse movement and forward return movement.

£ Constant surface speed control on/off (G96,G97)

If the on/off mode of constant surface speed control is specified in reverse movement, the specified mode is used in subsequent reverse movement. In other words, when a block specifying G96Sxxxx; appears in reverse movement, constant surface speed control is on for subsequent reverse movement. When a block specifying G97Sxxxx; appears in reverse movement, constant surface speed control is off for subsequent reverse movement. Note that the on/off mode of constant surface speed control in forward movement is reversed in reverse movement.

£ Clamping maximum spindle speed (G92Sxxxx)

If the command for clamping maximum spindle speed is specified in reverse movement, the specified clamp is applied to subsequent reverse movement. In other words, when G92Sxxxx appears in reverse movement, the spindle speed is clamped at Sxxxx. Note, however, that the spindle speed is clamped only when the G96 mode is set.

£ Auxiliary function

The M, S, and T functions, and secondary auxiliary functions (B functions) are output directly in reverse movement and forward return movement. When an M, S, or T function, or secondary auxiliary function (B function) is specified in a block containing a move command, the function and the move command are output at the same time in forward movement, reverse movement, and forward return movement. This means that the position where an M, S, or T function, or secondary auxiliary function (B function) is output differs in forward movement, reverse movement, and forward return movement.

£ Custom macro operation

All custom macro operations are ignored in reverse movement and forward return movement.

£ Tool withdrawal and return function

The tool cannot move along the path retraction or repositioning performed using the tool withdrawal and return function. All retraction and repositioning operations are ignored in reverse movement and forward return movement.

4.12 ACTIVE BLOCK CANCEL

Part programs can be halted, and the remaining amount of tool movement in the currently executing block can be canceled by pressing the active block cancel switch on the machine operator's panel. Though this halts automatic operation, modal information is still maintained. Operation is restarted from the block following the block where operation was canceled by cycle start.

Operation is as follows when the active block cancel switch on the machine operator's panel is pressed during automatic operation:

- 1 When the machine tool is moving, tool feed decelerates and comes to a stop, and the remaining amount of tool movement is canceled.
- 2 Dwell is canceled during dwell execution.
- 3 Refer to the relevant manual published by the machine tool builder for operations during execution of miscellaneous functions, spindle functions, tool functions and second auxiliary functions.

If the Cycle Start button is pressed with machine operation halted, automatic operation is restarted from the block following the block where operation was canceled.

Refer to the relevant manual published by the machine tool builder for the active block cancel switch.

Explanation

£ Operation at restart of operation (G90)/(G91)

When operation is restarted, operation at restart of operation is determined by the command programmed to the block following the block where operation was canceled. If the following block is programmed using absolute commands, operation is restarted after the tool moves towards the end point of the following block from the position where it stopped. If incremental commands are programmed in the following block, operation is restarted in the incremental mode from the position where the tool stopped.

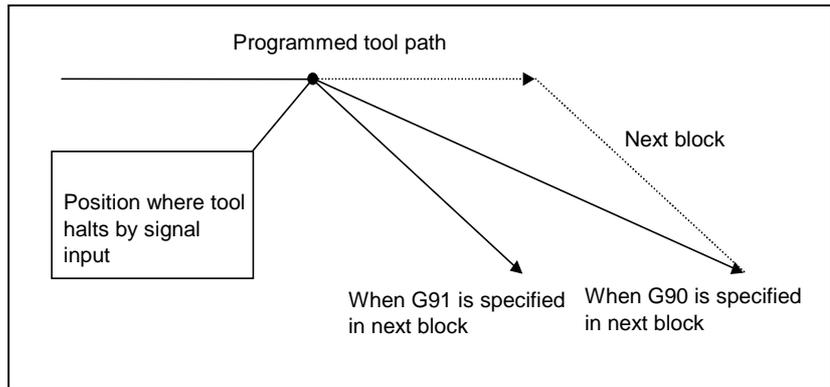


Fig. 4.12 (t) G90/G91

£ **Cutter compensation**

The tool path after cutter compensation is calculated from the block following the position where the tool stopped and from the subsequent block. The tool moves as follows when program execution is restarted after cutter compensation:

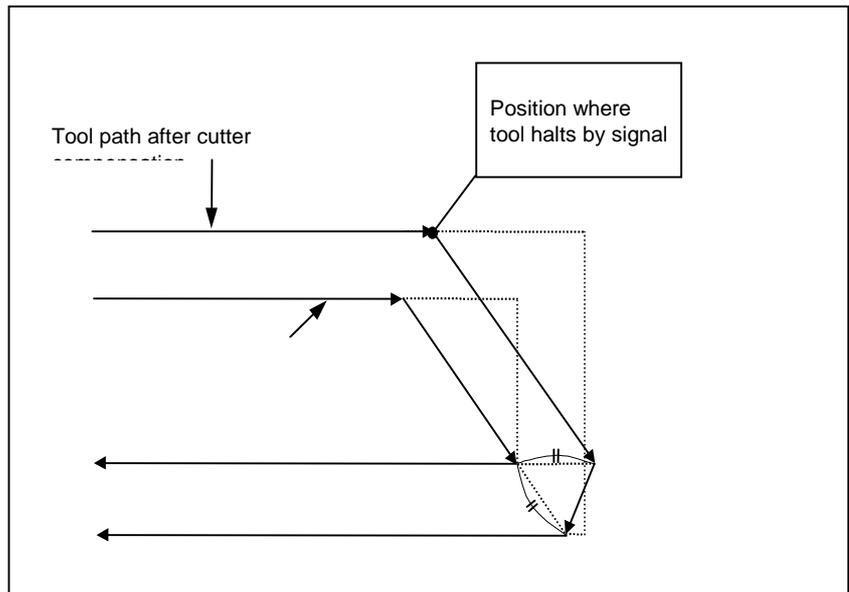


Fig. 4.12 (u) Cutter compensation

Canned cycle

When a block is canceled during a canned cycle, that canned cycle is canceled. Program execution is restarted from the next block.

4 Position where tool halts by signal input

```
N10 G84 G91 X_ Y_ Z_ R_ L3;  
N20 G00 G90 X_ Y_ Z_
```

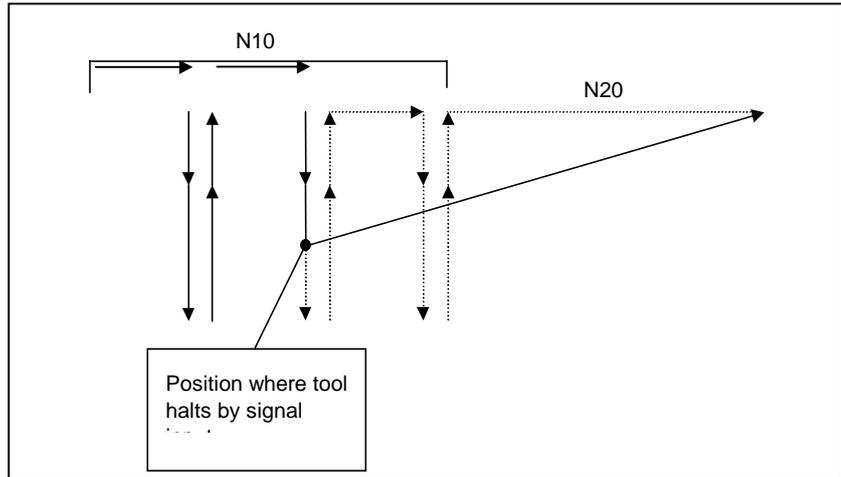


Fig. 4.12 (v) Fixed cycle

NOTE

Before restarting operation, make sure that the tool path does not interfere with the workpiece.

4.13 MANUAL HANDLE INTERRUPT

In automatic operation, this function allows a move command generated by the manual pulse generator to be superimposed on a programmed move command. As a result, the actual tool path deviates from the programmed path by the amount by which the operator turns the manual pulse generator. Therefore, this function can give the operator an additional margin in machining.

Explanations

£ Interrupt operation

- (1) A manual handle interrupt can be accepted in modes other than the positioning mode (such as G00).

Even in a block that does not move the tool or when automatic operation is in the stopped state, a manual handle interrupt can be accepted unless the positioning mode is effective.

NOTE

Ⓐ Even if the feedrate override signal is set to 0%, a manual handle interrupt is accepted.

Ⓚ In positioning mode, the tool moves at the rapid traverse rate.

Therefore, commands such as G28 and G29 can also be considered to be positioning mode.

- (2) For how to select a target axis for manual handle interrupt, refer to the manual provided by the machine tool builder.
- (3) The speed during manual handle interrupt is the sum of the speed in automatic operation and that of movement by manual handle interrupt. The speed, however, is controlled so as not to exceed the upper cutting feedrate on the axis.
- (4) The magnification used in the manual handle interrupt depends

Example

Suppose that the upper limit of the cutting feedrate on a given axis is 5 m/min. If the tool is moving along that axis in the positive direction at 2 m/min, the manual pulse generator can be turned to set up to 3 m/min. When the same movement is interrupted by turning the generator in the negative direction, up to 7 m/min is possible.

If the manual pulse generator is turned in beyond the limit, the excess pulses generated from the generator are discarded. Consequently, the reading of the pulse generator disagrees with the actual traveled distance made by the interrupt.

on the states of signals MP1, MP2, and MP4.

- (5) If the direction of movement is reversed as a result of a manual handle interrupt, backlash compensation is applied. In this case, pitch error compensation is applied to the position set after the interrupt.
- (6) For a manual handle interrupt, the application of acceleration/deceleration is based both on the cutting feed acceleration/deceleration (parameter No. 1622) and jog feed acceleration/deceleration (parameter No. 1624). When bit 0 (MNJ) of parameter No. 1604 is set to 1, however, the time constant for jog feed acceleration/deceleration is disabled, so that the application of acceleration/deceleration for a manual handle interrupt is based only on the time constant for cutting feed acceleration/deceleration.

£ Manual handle interrupt and coordinate systems

- (1) The workpiece and local coordinate systems are shifted by the magnitude of a manual handle interrupt. Therefore, when an absolute command is specified in a workpiece or local coordinate system, the end point is shifted by the magnitude of the interrupt. All workpiece and local coordinate systems are shifted by the same amount at the same time regardless of which coordinate system has been selected when the interrupt is performed.
- (2) A manual handle interrupt does not change the basic machine coordinate system. Therefore, the end points of an absolute command (G53) and automatic reference position return (G28) issued in the machine coordinate system are not affected by the manual handle interrupt.

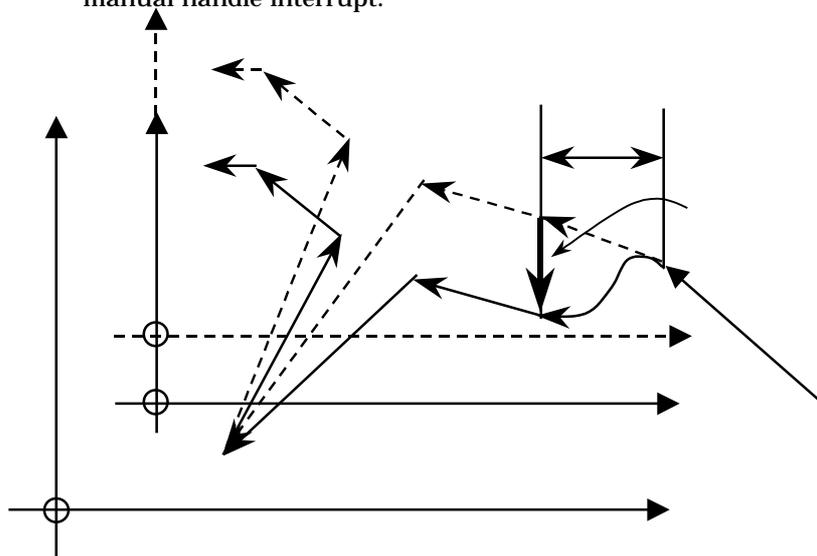


Fig. 4.13 (w) Fixed cycle

- (3) The magnitude of the manual handle interrupt can be cleared by presetting the workpiece coordinate system (G92.1).

£ Position display

- 1 The following table shows the relationship between movement by handle interruption and position display data:

Display	Relation
Absolute coordinate value	The absolute coordinate value does not change by handle interruption.
Relative coordinate value	The relative coordinate value changes by the handle interruption amount.
Machine coordinate value	The machine coordinate value changes by the handle interruption amount.

- 2 The amount of movement by manual handle interruption can be displayed on the manual interruption amount screen in the current position display.

If you press the chapter selection soft key after

pressing function key, the  amount of movement

by handle interruption is displayed.

Fig. 4.13 (x)

£ Relation with functions

The following table shows the relationship between handle interruption movement and the functions:

Signal	Relationship
Machine lock	The machine lock is effective. When the machine lock is ON, handle interruption is ignored.
Interlock	The interlock is effective. When the interlock is ON, the machine does not move.
Mirror image	Mirror image is ineffective. Even when the mirror image is ON, handle interruption in the plus direction moves the tool in the plus direction.

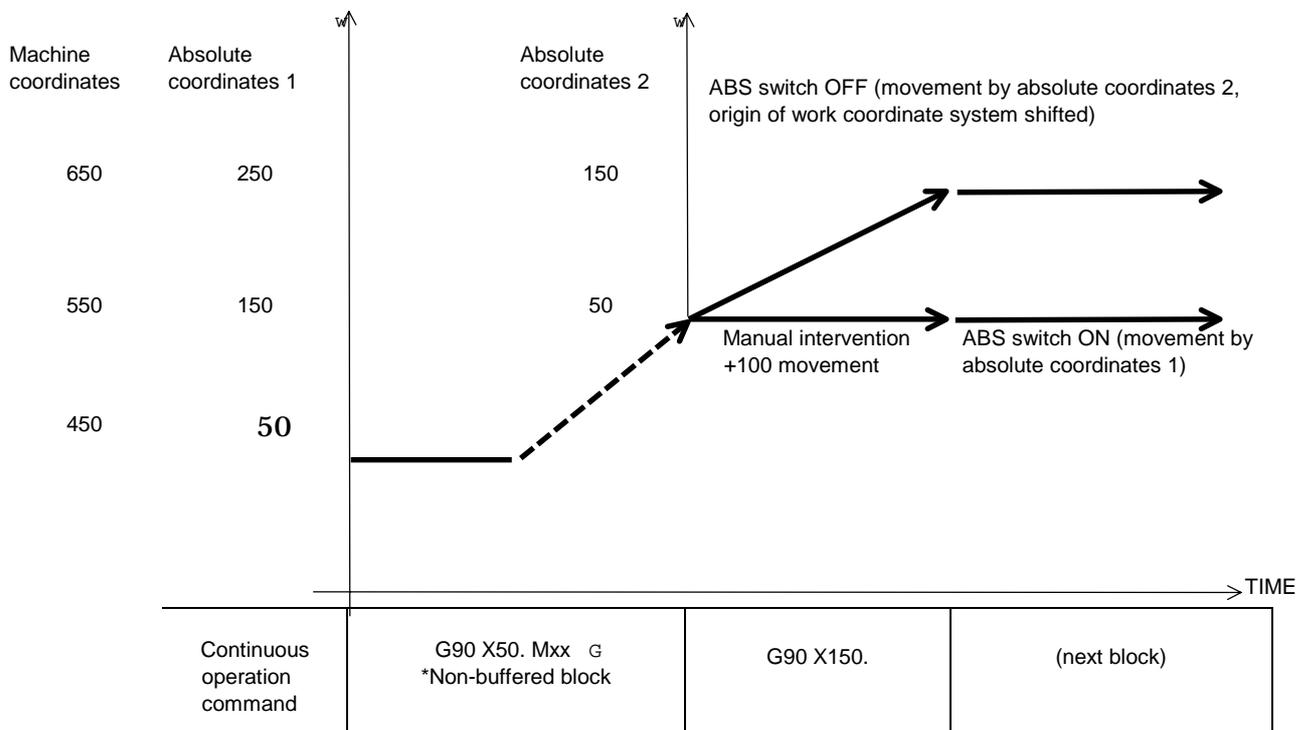
4.14 MANUAL INTERVENTION AMOUNT RETURN DURING AUTOMATIC OPERATION

Explanation

The manual intervention amount can be input during automatic operation without having to halt automatic operation. The manual intervention amounts for all axes are returned according to the manual absolute switch setting at completion of execution of the manual intervention amount return M code.

This reflects the manual intervention amounts during automatic operation in the movements programmed to the next command block to be read.

For details of the manual intervention amount return M code, refer to the manual published by the machine tool builder.



NOTE
The manual handle interrupt amount cannot be

4.15 SIMULTANEOUS AUTOMATIC AND MANUAL OPERATION

In the simultaneous automatic and manual mode, for example, cutting along an axis can be interrupted to change the depth of cut on that axis, or the next workpiece can be set up during automatic operation.

Explanation

£ Manual operation during simultaneous automatic and manual operation

There are two types of simultaneous automatic and manual operation.

(i) Interrupt type

An interrupt can be made for an axis for which the parking switch on the machine operator's panel is set to off. For the parking switch, refer to the manual provided by the machine tool builder. The interrupt type is enabled or disabled by the setting of the EMI bit (bit 6) of parameter No. 1000.

(ii) Setup type

For an axis whose park signal is on, movement by manual continuous feed or incremental feed can be performed during automatic operation. An interrupt can be made regardless of the automatic operation status.

For an axis whose park signal is on, automatic operation cannot be performed. If automatic operation is attempted for such an axis, an alarm occurs.

£ Switching between interrupt type and setup type

- 1 When the simultaneous automatic and manual operation mode is selected

The type of simultaneous automatic and manual operation is determined depending on the parking switch status when simultaneous automatic and manual operation mode is selected.

- 2 When the parking switch setting is changed

In automatic operation, the time at which the type of simultaneous automatic and manual operation is switched differs between single-block operation and continuous operation.

[In single-block operation]

The type of simultaneous automatic and manual operation is switched at the block subsequent to the block currently being executed.

[In continuous operation]

The block at which the type of simultaneous automatic and manual operation is switched varies depending on the execution

status of automatic operation. During continuous operation, do not change the parking switch setting.

[At reset]

The type of simultaneous automatic and manual operation is switched depending on the parking switch status at the time of reset.

£ Manual operation

Automatic operation during simultaneous automatic and manual operation is the same as ordinary automatic operation. Therefore, the following explains manual operation during simultaneous automatic and manual operation.

£ Simultaneous automatic and manual operation (in relation to automatic operation)

	Interrupt type	Setup type
In the positioning mode (*1)	Not possible	Possible
In the cutting mode (G01, G02, G03, etc.)	Possible	Possible
In the threading mode	Possible	Possible

As indicated in the above table, the setup type is not affected by automatic operation. For the interrupt type, interrupts are possible in modes other than the positioning mode.

NOTE

In the positioning mode, the tool is moved at the rapid traverse feedrate. Therefore, blocks containing G28 and G29 are also included.

£ Feedrate and acceleration/deceleration time constant

The following lists the traveled distance, feedrate, and acceleration/deceleration time constant:

	Interrupt type	Setup typ
Traveled distance	Same as normal manual operation	Same as normal manual operation
Feedrate	(*1)	Same as normal manual operation
Time constant	(*2)	Same as normal manual operation

NOTE

P(*1)The feedrate along the axis subject to an interrupt is the result of superimposition of the feedrate by the manual interrupt on the feedrate in automatic operation. If the resultant feedrate exceeds the maximum cutting feedrate for that axis, it is clamped at the maximum cutting feedrate.

QFor interrupt type, the application of acceleration / deceleration is based both on the time constant for cutting feed acceleration/deceleration (parameter No. 1622) and that for jog feed acceleration/deceleration (parameter No. 1624). When bit 0 of parameter No. 1604 is set to 1, however, the time constant for jog feed acceleration / deceleration is disabled, so that the application of acceleration / deceleration is based only on the time constant for cutting feed acceleration/deceleration.

£ Relationship with signals

The operation is related to signals as listed below:

	Interrupt type	Setup typ
Absolute on/off	Not affected (always off)	Affected
Interlock	Affected. If the signal is on, operation is not carried out.	
Machine lock	Affected. If the signal is on, operation is not carried out.	
Mirror image	Not affected. Even if the signal is on, a positivecommand moves the machine in the positive direction.	

£ Indication

The indication is as follows:

	Interrupt type	Setup typ
Absolute coordinates	Not updated	Updated
Relative coordinates	Updated	Updated
Machine coordinates	Updated	Updated

5

TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

- 5.1 Machine Lock and Auxiliary Function Lock
- 5.2 Feedrate Override
- 5.3 Rapid Traverse Override
- 5.4 Dry Run
- 5.5 Single Block

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the tool, use machine lock.

There are two types of machine lock: all-axis machine lock, which stops the movement along all axes, and specified-axis machine lock, which stops the movement along specified axes only. In addition, auxiliary function lock, which disables M, S, and T commands, is available for checking a program together with machine lock.

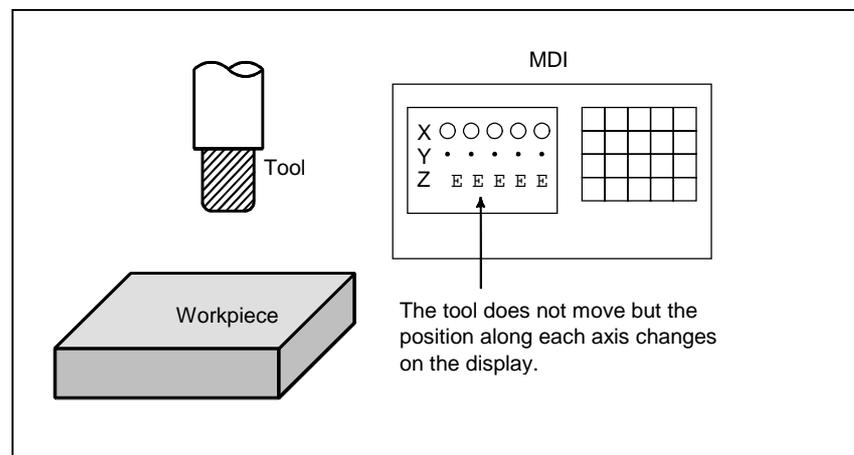


Fig.5.1 (a) Machine lock

Procedure for Machine Lock and Auxiliary Function Lock

£ Machine Lock

Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving. Some machines have a machine lock switch for each axis. On such machines, press the machine lock switches for the axes along which the tool is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.

CAUTION

The positional relationship between the workpiece coordinates and machine coordinates may differ before and after automatic operation using machine lock. In such a case, specify the workpiece coordinate system by using a coordinate setting command or by performing manual reference position return.

£ Auxiliary Function Lock

Press the auxiliary function lock switch on the operator's panel. M, S, T and B codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Limitation

£ M, S, T, B command by only machine lock

M, S, T and B commands are executed in the machine lock state.

£ Reference position return under Machine

When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

£ M codes not locked by auxiliary function lock

M00, M01, M02, M30, M98, and M99 commands are executed even in the auxiliary function lock state. M codes for calling a subprogram (parameters No. 6071 to 6079) and those for calling a custom macro (parameter No. 6080 to 6089) are also executed.

5.2 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

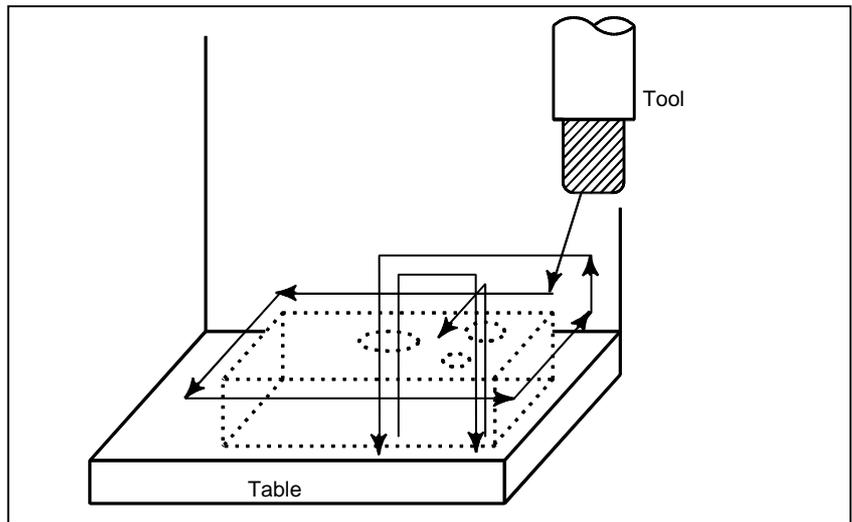


Fig.5.2 (b) Dry run

Procedure for Dry Run

Procedure

Press the dry run switch on the machine operator's panel during automatic operation.

The tool moves at the feedrate specified in a parameter. The rapid traverse switch can also be used for changing the feedrate.

Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

£ Dry run feedrate

SINGLE BLOCK	OPT STOP	BLOCK SKIP	MC LOCK
DRY RUN	MST LOCK	WORK LIGHT	

The dry run feedrate changes as shown in the table below according to the rapid traverse switch and parameters.

Rapid traverse button (RT signal)	Positioning type #4(LRP of parameter No.1400)	Programmed command	
		For rapid traverse	For cutting feed(*2)
RT=ON	LRP=0/1	Rapid traverse rate	The dry run feedrate ~the maximum manual feedrate override value
OFF	LRP=0	Jog feedrate (*1)	The dry run feedrate ~the manual feedrate override value
	LRP=1	Same as for cutting feed (*1)	

Rapid traverse rate.....Setting by parameter No.1420

Dry run feedrate.....Setting by parameter No.1410

Jog feedrate.....Setting by parameter No. 1423

Maximum manual feedrate override value

.....Setting by parameter No. 2016

Manual feedrate override value

.....Input signals (*JV0 to *JV15)

(*1) For rapid traverse, the indicated feedrates are valid only when RDR of parameter No.1400 is 1. If RDR is 0, the rapid traverse rate is used in the same way as when the rapid traverse button is set to ON

(*2) The feedrate is clamped at the maximum cutting feedrate. (parameter No.1422)

5.3 SINGLE BLOCK

Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.

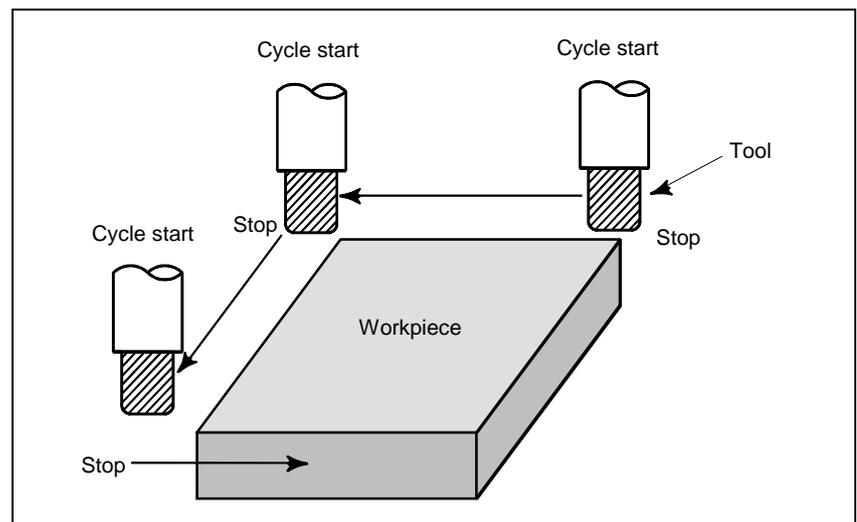


Fig.5.3 (a) Single block

Procedure for Single block

Procedure

- 1 Press the single block switch on the machine operator's panel. The execution of the program is stopped after the current block is executed.
- 2 Press the cycle start button to execute the next block. The tool stops after the block is executed. Refer to the appropriate manual provided by the machine tool builder for single block execution.

Explanation

£ Reference position return and single block

If G28 to G30 are issued, the single block function is effective at the intermediate point.

£ Single block during a canned cycle

In a canned cycle, the single block stop points are the end of 1, 2, and 6 shown below. When the single block stop is made after the point 1 or 2, the feed hold LED lights.

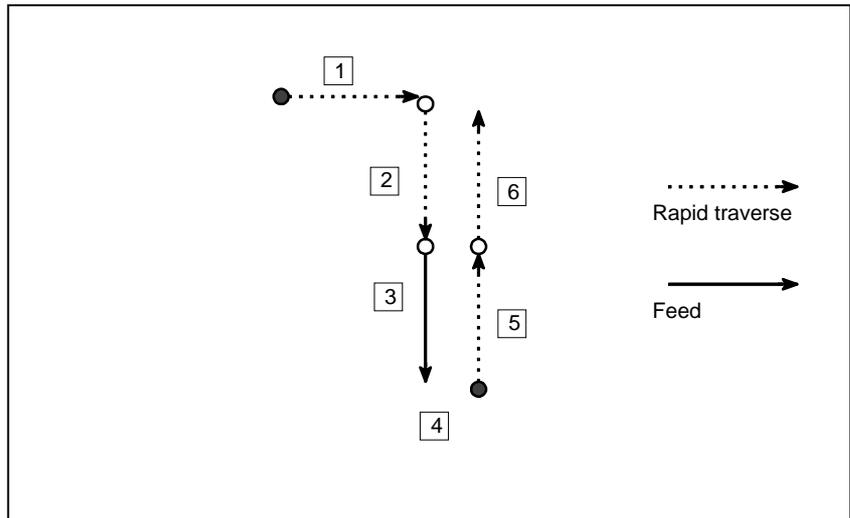


Fig.5.3 (b) Single block during canned cycle

£ Subprogram call and single block

Single block stop is not performed in a block containing M98P_; M99; or G65, G66 or G67.

However, single block stop is even performed in a block with M98P_ or M99 command,

if the block contains an address other than O, N, P, L.

6

SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the tool from exceeding the stroke ends, Overtravel check and Stroke check are available. This chapter describes emergency stop., overtravel check, and stroke check.

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

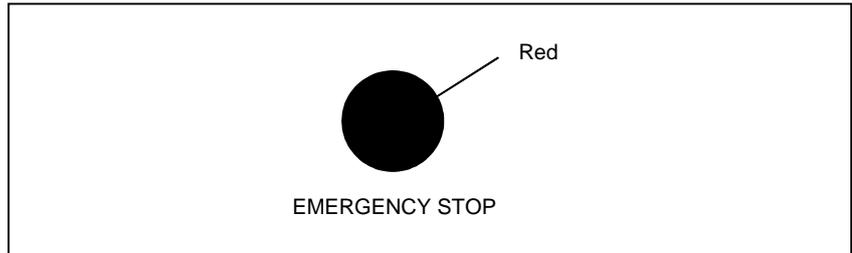


Fig.6.1 (a) Emergency stop

For how to use the emergency stop button, see the manual provided by the machine tool builder.

CAUTION

Causes of trouble must be removed before the button is released.

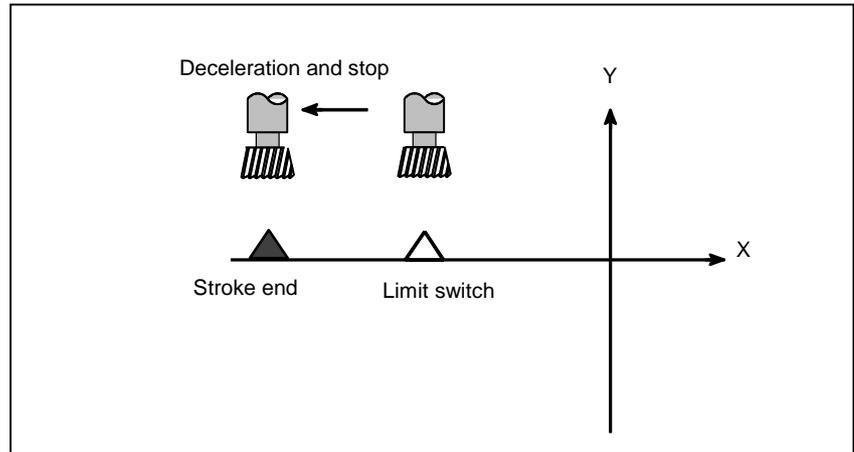
NOTE

- 1 EMERGENCY STOP interrupts the current to the motor.
- 2 The CNC enters the reset or alarm status. (Whether the CNC enters the reset or alarm status is specified by the bit0 ENR in parameter No. 2001.)

6.2 OVERTRAVEL

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

Fig.6.2 (a) Overtravel



Explanation

£ Overtravel during automatic operation

When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an overtravel alarm is displayed.

£ Overtravel during manual operation

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

£ Releasing overtravel

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

£ Alarm

Number	Message	Contents
OT0007	+ OVERTRAVEL (HARD)	<p>The stroke limit switch in the positive direction was triggered.</p> <p>This alarm is generated when the machine reaches the stroke end.</p> <p>When this alarm is not generated, feed of all axes is stopped during automatic operation.</p> <p>During manual operation, only the feed of the axis on which the alarm occurred is stopped.</p>
OT0008	- OVERTRAVEL (HARD)	<p>The stroke limit switch in the negative direction was triggered.</p> <p>This alarm is generated when the machine reaches the stroke end.</p> <p>When this alarm is not generated, feed of all axes is stopped during automatic operation.</p> <p>During manual operation, only the feed of the axis on which the alarm occurred is stopped.</p>

6.3 STROKE CHECK

Three areas which the tool cannot enter can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

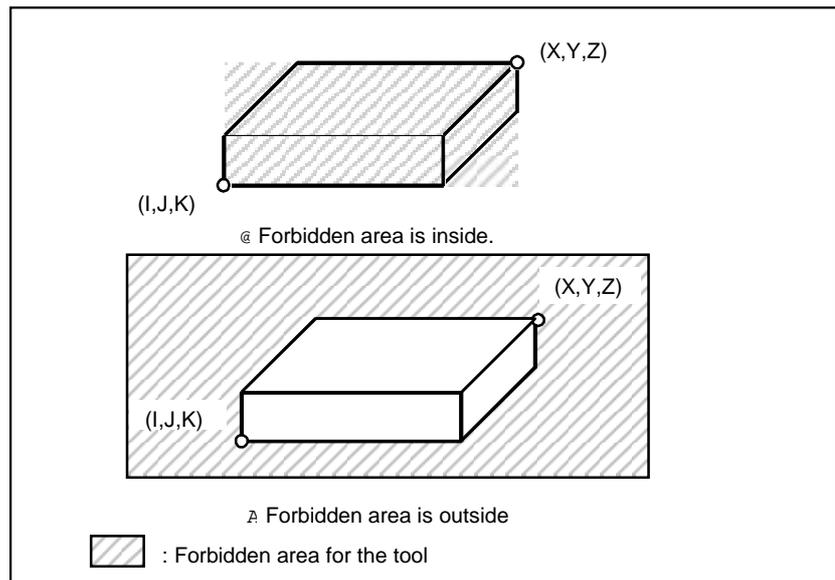


Fig.6.3 (a) Stroke check

When the tool exceeds a stored stroke limit, an alarm is displayed and the tool is decelerated and stopped.

When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

Explanation

£ Stored stroke check 1

Parameters (Nos. 5220, 5221 or Nos. 5260, 5261) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.

£ Stored stroke check 2 (G22, G23)

Parameters (Nos. 5222, 5223) or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter OUT (No. 5200#0) selects either inside or outside as the forbidden area.

In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area. Each of G22; and G23; should be commanded independently of another commands in a block.

The command below creates or changes the forbidden area:

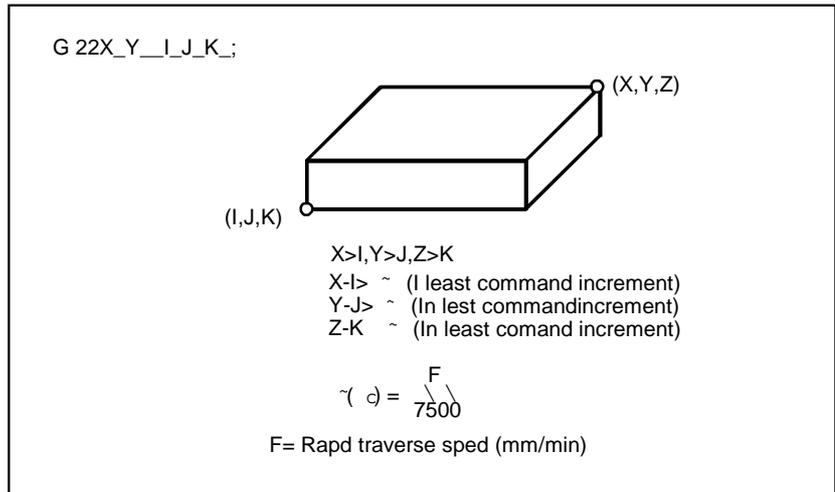


Fig.6.3 (b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the figure below must be set.

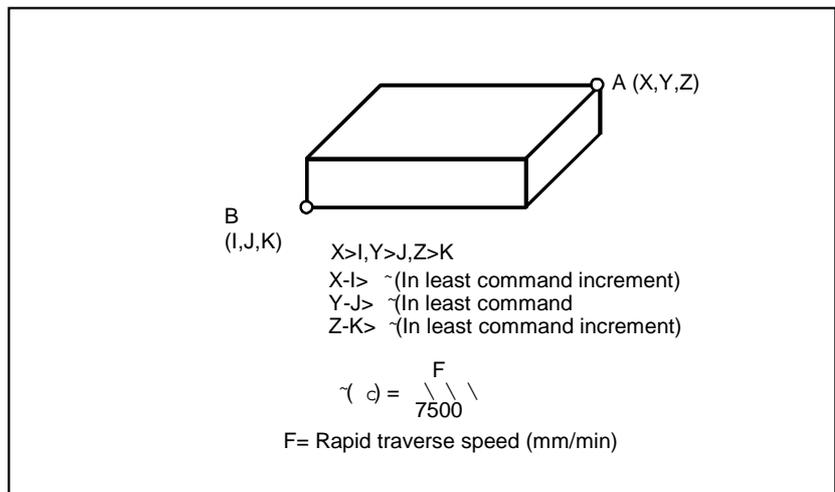


Figure 12.3 (c) Creating or changing the forbidden area using a parameters

In stored stroke check 2, even if you mistake the order of the coordinate value of the two points, a rectangular, with the two points being the apexes, will be set as the area.

When you set the forbidden area through parameters (Nos. 1322, 1323), the data should be specified by the distance from the machine coordinate system in the least command increment. (Output increment)

If it is set by a G22 command, specify the data by the distance from the machine coordinate system in the least input increment (Input increment.)

The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

£ Checkpoint for the forbidden area

Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.

If point A (The top of the tool) is checked in Fig. 6.3 (d) , the distance "a" should be set as the data for the stored stroke limit function. If point B (The tool chuck) is checked, the distance "b" must be set.

When checking the tool tip (like point A), and if the tool length varies for each tool, setting the forbidden area for the longest tool requires no re-setting and results in safe operation.

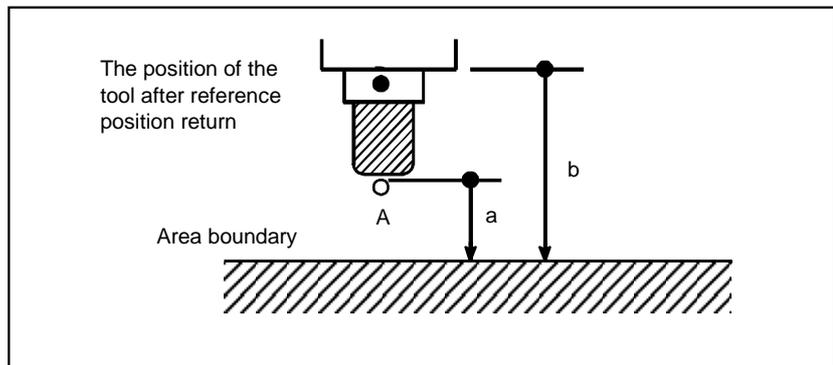


Fig.6.3 (c) Setting the forbidden area

£ Forbidden area over lapping

Area can be set in piles.

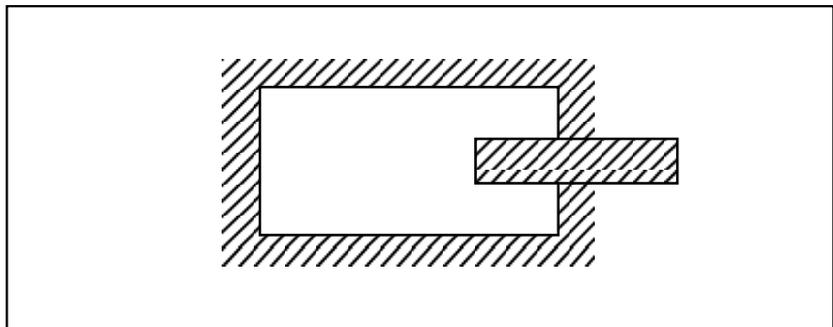


Fig.6.3 (d) Setting the forbidden area over lapping

Unnecessary limits should be set beyond the machine stroke.

£ Stop position with stored stroke limit

With stored stroke check 1, the tool stops a maximum of L mm before a set forbidden area. ($L_{mm} = F/7500$, where F denotes the rapid traverse rate (mm/min).) Therefore, the tool does not enter the forbidden area.

With stored stroke check 2, the tool overruns a maximum of L mm. The tool enters a set forbidden area by a maximum of L mm.

£ Effective time for a forbidden area

Each limit becomes valid when reference position return has been performed manually or by G28 after power-up.

After the power is turned on, if the reference position is in the forbidden

area of each limit, an alarm is generated immediately. (Only in G22 mode for stored stroke limit 2).

£ Releasing the alarms

If the tool enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward.

£ Change from G23 to G22 in a forbidden area

When G23 is switched to G22 in the forbidden area, the following results.

- (1) When the forbidden area is inside, an alarm is informed in the next move.
- (2) When the forbidden area is outside, an alarm is informed immediately.

If the tool cannot be moved out of the forbidden area, press the emergency stop button then release it, set G23 to retract the tool, correct the setting error, if any, then perform reference position return again.

NOTE

When the reference position return function is not provided for an axis, no forbidden area is set for that axis, so no forbidden-area alarm is generated for the axis.

NOTE

Ⓐ In setting a forbidden area, if the two points to be set are the same, the area is as follows:

- (1) When the forbidden area is stored stroke check 1, all areas are

forbidden areas.

(2) When the forbidden area is stored stroke check 2 or stored stroke check 3, all areas are movable areas.

Q Only the three basic axes can be specified in G22.

Alarm

Number	Message	Contents
OT0001	+ OVERTRAVEL (SOFT 1)	The tool entered the prohibited area of stored stroke check 1 during movement in the positive direction.
OT0002	- OVERTRAVEL (SOFT 1)	The tool entered the prohibited area of stored stroke check 1 during movement in the negative direction.
OT0003	+ OVERTRAVEL (SOFT 2)	The tool entered the prohibited area of stored stroke check 2 during movement in the positive direction.
OT0004	- OVERTRAVEL (SOFT 2)	The tool entered the prohibited area of stored stroke check 1 during movement in the negative direction.

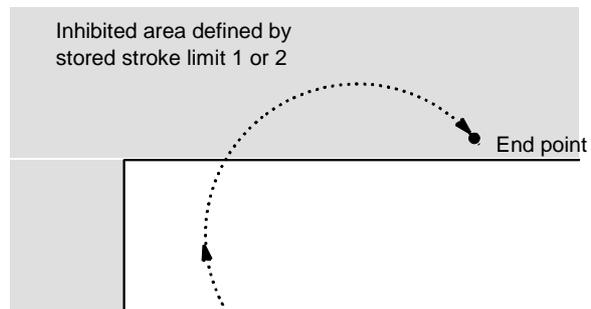
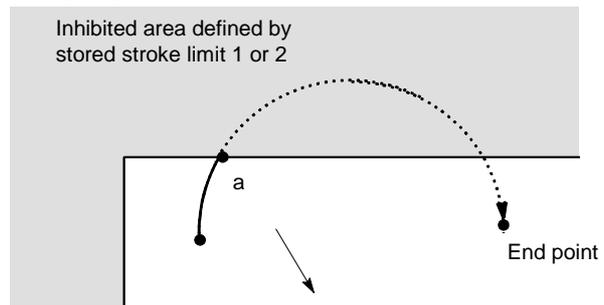
6.4 STROKE LIMIT CHECK PRIOR TO PERFORMING MOVEMENT

During automatic operation, before the movement specified by a given block is started, whether the tool enters the inhibited area defined by stored stroke limit 1 or 2 is checked by determining the position of the end point from the current position of the machine and a specified amount of travel. If the tool is found to enter the inhibited area defined by a stored stroke limit, the tool is stopped immediately upon the start of movement for that block, and an alarm is displayed.

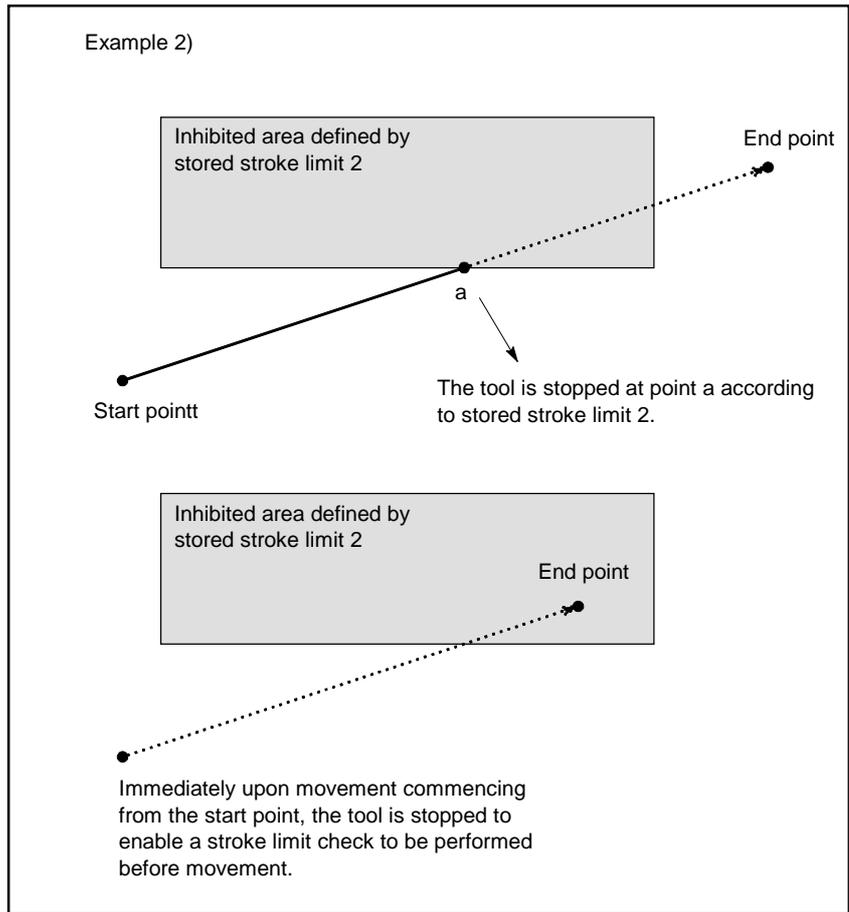
WARNING

Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a inhibited area is checked. In this case, the path followed by a move command is not checked. However, if the tool enters the inhibited area defined by stored stroke limit 1 or 2, an alarm is issued. (See the examples below.)

Example 1)



Immediately upon movement commencing from the start point, the tool is stopped to enable a stroke limit check to be performed before movement.



Limitation

£ **Manual numeric command**

For manual numeric commands, no stroke limit check is made before movement.

£ **Machine lock**

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

£ **G23**

When stored stroke limit 2 is disabled (G23 mode), no check is made to determine whether the tool enters the inhibited area defined by stored stroke limit 2.

£ **A block consisting of multiple operations**

If a block consisting of multiple operations (such as a canned cycle and Cutler Compensation) is executed, an alarm is issued at the start point of any operation whose end point falls within a inhibited area.

£ Polar coordinate interpolation mode

In polar coordinate interpolation mode, no check is made.

£ Three-dimensional coordinate conversion

In three-dimensional coordinate conversion mode, no check is made.

£ PMC axis control

No check is made for a movement based on PMC axis control.

Alarm

Number	Message	Contents
OT0021	+ OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the negative direction during the stroke check before movement.
OT0022	- OVERTRAVEL (PRE-CHECK)	The tool exceeded the limit in the positive direction during the stroke check before movement.

7

ALARMS AND SELF DIAGNOSTIC FUNCTIONS

7.1 INTERNAL POSITION COMPENSATION DATA DISPLAY FUNCTION

The function for displaying internal position compensation data displays the data used in the CNC for each function while the following functions are executed:

- .Cutter compensation
- .Tool length compensation
- .Drilling canned cycle
- .Coordinate rotation
- .Three-dimensional coordinate conversion
- .Programmable mirror image
- .Scaling
- .Three-dimensional tool compensation
- .Three-dimensional cutter compensation

Display

Follow the procedure described in Section III.x.x, "Programs" to display the internal position compensation data screen. The right half of the screen displays, like the program check screen, position data (selected from relative coordinates, absolute coordinates, and machine coordinates), the remaining amount of travel, modal values, and the actual speed. The left half of the screen displays position compensation information for each function and the program being executed

Cutter compensation

Fig.7.1 (a) Cutter compensation

MODE

Displays ON in reverse video when the current mode is cutter compensation mode, and displays OFF in reverse video when the current mode is other than cutter compensation mode.

OFFSET VECTOR

Displays the cutter compensation vector created for each block.

Start-Up

Displays the startup cancel type.

Neglect Limit

Displays the limit for a small amount of movement generated by compensation that can be neglected (parameter No. 6010).

Tool Offset

Displays the currently selected tool compensation number and tool offset data.

Tool length offset

Fig.7.1 (b) Tool length offset

MODE

Displays ON in reverse video when the current mode is tool length compensation mode, and displays OFF in reverse video when the current mode is other than tool length compensation mode.

OFFSET VECTOR

Displays the tool length compensation vector created for each block. Any axis can be subject to tool length compensation

depending on the parameter setting. Therefore, offset vectors for all axes are displayed.

Tool Offset

Displays the currently selected tool compensation number and tool offset data.

Canned cycle for drilling

Fig.7.1 (c) Canned cycle for drilling

MODE

Displays ON in reverse video when the current mode is canned drilling cycle mode, and displays OFF in reverse video when the current mode is other than canned drilling cycle mode.

INITIAL POINT

Displays the absolute position of the initial point.

POINT R

Displays the absolute position of point R.

POIN Z

Displays the absolute position of point Z.

NUMBER OF REPETITIONS

Displays the specified value (CMD) and current value (ACT) of the number of repetitions.

CUT IN VALUE

Displays the depth of cut for G73 and G83.

SHIFT VALUE

Displays the shift for G76 and G87.

Coordinate System rotation

Fig.7.1 (d) Coordinate System rotation**MOOD**

Displays ON in reverse video when the current mode is coordinate system rotation mode, and displays OFF in reverse video when the current mode is other than coordinate system rotation mode.

ROTATION CENTER

Displays the absolute position of the center of coordinate rotation.

ANGLE OF ROTATION

Displays the angle of coordinate rotation.

Three-dimensional coordinate conversion 13.2 (e)

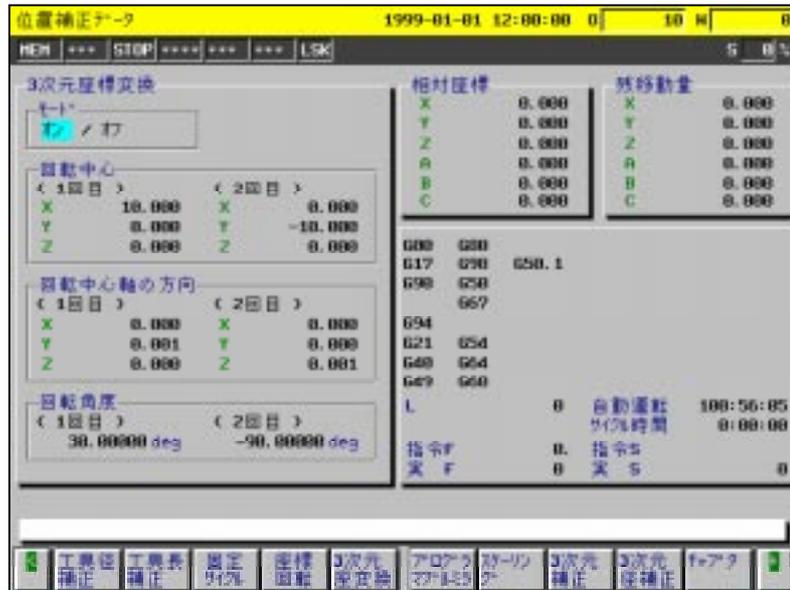


Fig.7.1 (e) Three-dimensional coordinate conversion

l n c d

Displays ON in reverse video when the current mode is three-dimensional coordinate conversion mode, and displays OFF in reverse video when the current mode is other than three-dimensional coordinate conversion mode.

ROTATION CENTER

Displays the absolute position of the rotation center for three-dimensional coordinate conversion.

DIRECTION OF CENTER AXIS OF ROTATION

Displays the direction of the rotation center axis for three-dimensional coordinate conversion

ROTATION ANGLE

Displays the rotation angle for three-dimensional coordinate conversion.

Programmable mirror image

Fig.7.1 (f) Programmable mirror image

MODE

Displays ON in reverse video when the current mode is programmable mirror image mode, and displays OFF in reverse video when the current mode is other than programmable mirror image mode.

MIRROR CENTER

Displays the absolute coordinates of the mirror center position. For axes not specified for programmable mirroring, no value is displayed. Therefore, when 0 is indicated, the mirror center is located at position 0.

Scaling

Fig.7.1 (g) Scaling

MODE

Displays ON in reverse video when the current mode is scaling mode, and displays OFF in reverse video when the current mode is other than scaling mode.

SCALING CENTER

Displays the absolute position of the scaling center.

SCALE FACTOR

Displays the scale factor for each axis.

Three- dimensional tool compensation

Fig.7.1 (h) Three- dimensional tool compensation

MODE

Displays ON in reverse video when the current mode is three-dimensional tool compensation mode, and displays OFF in reverse video when the current mode is other than three-dimensional tool compensation mode.

OFFSET VECTOR

Displays the three-dimensional tool compensaion vector created for each block.

Tool Offset

Displays the currently selected tool compensation number and tool offset data.

Three dimensional cutter compensation

Fig.7.1 (i) Three dimensional Cutter compensation

MODE

Three-dimensional cutter compensation has two modes: Tool side offset mode and leading edge offset mode. When the current mode is either of these mode, ON is displayed in reverse video. When the current mode is other than these modes, OFF is displayed in reverse video.

OFFSET VECTOR

Displays the three-dimensional tool compensation vector created for each block.

Tool Offset

Displays the currently selected tool compensation number and tool offset data.

7.2 COORDINATE SYSTEM RELATED DATA DISPLAY

Offset data related to coordinate systems is displayed on the diagnosis screen described in Section III.7.3.

Explanation

The workpiece coordinate system used for automatic operation is defined with several offsets / shifts.

The controlled axes, the machine coordinates, absolute coordinates, and offsets / shifts are related as follows:

Machine coordinate	Absolute coordinate	
	{External workpiece zero	
	{Currently valid offset among those caused by G54 to G59	
	{Shift amount of the coordinate system at manual intervention made in the manual absolute off state	1
	{Shift amount of the coordinate system by manual interrupt in a function such as simultaneous automatic and manual operation	2
	{Offset caused by G92	3
	{Offset caused by G52	4
	{Shift amount of the coordinate system by a move command issued with the machine locked	5

Data 1 to 5 in the above expression can be checked on the diagnosis screen.

	Diagnosis number	Indication
1	4100	Manual shift
2	4101	Manual interruption value
3	4102	G92 offset
4	4103	G52 offset
5	4104	Machine lock offset

7.3 PERIODIC MAINTENANCE SCREEN

The periodic maintenance screen provides a guideline indicating when consumables that require periodic replacement (such as the battery for control unit backup and the LCD unit backlight) are to be replaced and checked if their service lives are set in advance.

Displaying periodic maintenance screens

There are three periodic maintenance screens.

Status display screen

To display the status display screen, follow the procedure described in Section III.x.x, "System." The statuses of items subject to periodic maintenance can be checked. The statuses of up to ten items can be displayed.

Fig.7.3 (a) Status Display Screen

Count status

The mark displayed to the left of each item number indicates the counting status.

Indication	Counting status
None	Counting stopped
	Counting in progress
	Life expired

Item number

A sequence number from 01 to 10.

Item Name

The name of an item subject to periodic maintenance is indicated. If the percentage (%) of the remaining service life is not greater than the percentage specified in parameter No. 2239, the item name is displayed in red to alert the operator.

Life

The service life of an item subject to periodic maintenance is indicated (in hours).

Remain

The remaining service life of an item subject to periodic maintenance is indicated (in hours). If the percentage (%) of the remaining service life is not greater than the percentage specified in parameter No. 2239, the item name is displayed in red to alert the operator. The remaining time is displayed in hours, so a value of less than one hour is not displayed. When 20 hours is indicated as the remaining time, for example, the actual remaining time falls within the range of 20 hours to less than 21 hours.

Count Type

Displays the type of service life counting.

Indication	Meaning
NO COUNT	Counting is not performed (stopped)
ALL TIME	The expiration time is counted since a service life is set.
POWER ON	The power-on time is counted.
AUTO START	The automatic operation time is counted.
CUTTING	The cutting time is counted.

Item setting screen

This screen is displayed by pressing the [SET ITEM] soft key on the status display screen. On the item setting screen, an item can be selected to set its periodic maintenance data such as the service life and count type.

Fig.7.3 (b) Item Setting Screen

Item selection menu screen

This screen is displayed by pressing the [SET ITEM] soft key on the item setting screen. With the item selection menu screen, a list of items selected on the setting screen can be created.

Fig.7.3 (c) Item Selection Menu Screen

Machine

A menu of item names that can be set is displayed.

CNC

A menu of system-defined item names is displayed.

Setting periodic maintenance data

Set periodic maintenance data on the item setting screen.

Setting an item name

£ Setting by MDI key input

- 1) Position the cursor to a desired item name by using the



cursor keys .

- 2) Key in an item name in the input format, explained later.
- 3) Press the [INPUT] soft key or the  key.

The item name is set, and the service life, remaining time, and count type are initialized.

- 4) When an item name is keyed in, and the [+INPUT] soft key is pressed, the item name is added to an already-set item name. In this case, the life time, remaining time, and count type are not initialized.

£ Setting by selection from the item selection menu screen

- 1) Position the cursor to a desired item name by using the

cursor keys . 

- 2) Press the [SELECTITEM] soft key.
The menu screen appears.
- 3) Select a desired name from the item names registered on the item selection menu screen by positioning the cursor to that name.
- 4) Press the [SELECTEND] soft key.
The item setting screen is redisplayed, and the selected item name is set.

NOTE

1 An already set item name cannot be overwritten by an item name selected from the item selection menu screen. Delete the existing item name, then set a new item name.

2 When the item of the battery for the control unit or the battery for the absolute-position detector is selected and set, its life time, like other data, is managed by using the life time set on the periodic maintenance screen. Therefore, life time management is not related to the battery alarm detected by the system.

£ Input format

E E E E E	E E E E E E E E E	E E E E E
Alphanumeric	Kanji/hiragana	Alphanumeric

Describe alphanumeric characters directly. Kanji and hiragana character code must be enclosed by an asterisk (*). Either FANUC code or JIS code can be selected as the character code by setting bit 6 (JIS) of parameter No. 2286. A name can consist of up to 24 alphanumeric characters (onekanji or hiragana character is counted as two alphanumeric characters).

Example: To enter "TABLE," specify the following:

FANUC code	.TABLE*13AE08DA0738*
JIS code	.TABLE*3D61336A4C7D*

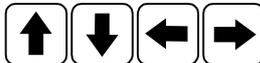
NOTE

Symbols *, [,], (,), and ;

cannot be included in an item name.

Setting the service life

- 1) Position the cursor to the desired service life by using

the cursor keys . 

- 2) Key in a time. The set value must be between 0 and 999999 (hours).

- 3) Press the [INPUT] soft key or the  key.

The service life and remaining time are set to the same value, and the count type is initialized.

- 4) When a time is keyed in, after which the [+INPUT] soft key is pressed, the set value is added to the existing service life. The value is also added to the remaining time.

NOTE

If an item name is not set, the service life cannot be set.

Setting the remaining time

- 1) Position the cursor to the desired remaining time by

using the cursor keys .



- 2) Key in a time. The set value must range from 0 to 999999 (hours).

- 3) Press the [INPUT] soft key or the key.



- 4) When a time is keyed in, and the [+INPUT] soft key is pressed, the set value is added to the existing remaining time.

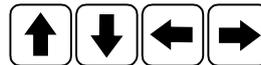
NOTE

If no item name is set, the remaining time cannot be set.

Setting the count type

- 1) Position the cursor to a desired count type by using the

cursor keys .



- 2) Select a count type by pressing the [NO COUNT], [ALL TIME], [POWER ON], [AUTO START], or [CUTTING] soft key.

Deleting an item

- 1) Press the [CLEAR ITEM] soft key.
- 2) Key in the item number (1 to 10) you want to delete.
- 3) Press the [EXEC] soft key.

Creating an item selection menu

Display the item selection menu screen.

Setting an item name

- 1) Position the cursor to a desired item name by using the

cursor  keys . Only the machine menu can be edited.

- 2) Key in an item name in the input format described previously.

- 3) Press the [INPUT] soft key or  key.

- 4) When an item name is keyed in, then the [+INPUT] soft key is pressed, the item name is added to the already set item name.

- 5) Press the [SELECT END] soft key.

The item setting screen is redisplayed.

Deleting an item name

- 1) Press the [CLEAR ITEM] soft key.

- 2) Key in an item number (1 to 10) you want to delete.

- 3) Press the [EXEC] soft key.

Only the machine menu can be deleted.

Setting item input/output

The periodic maintenance data and machine menu can be input/output to and from external devices.

Inputting setting items

Method 1 (Neither the input file name nor number need be specified.)

- (1) Select EDIT mode.

- (2) To input periodic maintenance data, display the item setting screen. To input the machine menu, display the item selection menu screen.

- (3) Press the [READ] soft key.

- (4) Press the [PERIODMAINTENANCE] soft key.

Method 2 (The input file is specified with its file name.)

- (1) Select EDIT mode.
- (2) To input periodic maintenance data, display the item setting screen. To input the machine menu, display the item selection menu screen.
- (3) Press the [READ] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME] soft key.
- (7) Press the [PERIODMAINTE] soft key.

Method 3 (The input file is specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) To input periodic maintenance data, display the item setting screen. To input the machine menu, display the item selection menu screen.
- (3) Press the [READ] soft key.
- (4) Press the [FAIL#] soft key.
- (5) Enter a file number.
- (6) Press the [PERIODMAINTE] soft key.

Method 4 (The input file is specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) To input periodic maintenance data, display the item setting screen. To input the machine menu, display the item selection menu screen.
- (3) Press the [READ] soft key.
- (4) Press the address key.
- (5) Enter a file number.
- (6) Press the [PERIODMAINTE] soft key.

Method 5 (The input file is specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) To input periodic maintenance data, display the item setting screen. To input the machine menu, display the item selection menu screen.
- (3) Press the address key.

- (4) Enter a file number.
- (5) Press the [READ] soft key.

NOTE

- | | |
|---|--|
| 1 | If an item name is omitted for an item number in the input data, the data corresponding to that item number is deleted. |
| 2 | When data is input from a FANUC Floppy Cassette, FANUC FA Card, or FANUC Handy File, the input file can be specified using a file name or file number. |
| 3 | When data is input from a memory card, the input file can be specified using a file name. |

Outputting setting items

Method 1 (Neither the output file name nor number need be specified.)

- (1) Select EDIT mode.
- (2) To output periodic maintenance data, display the item setting screen. To output the machine menu, display the item selection menu screen.
- (3) Press the [] soft key.
- (4) Press the [PERIODMAINTE] soft key.

Method 2 (The output file is specified with its file name.)

- (1) Select EDIT mode.
- (2) To output periodic maintenance data, display the item setting screen. To output the machine menu, display the item selection menu screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [PERIODMAINTE] soft key.

Method 3 (The output file is specified with its file number (one of three methods))

- (1) Select EDIT mode.
- (2) To output periodic maintenance data, display the item setting screen. To output the machine menu, display the item selection menu screen.
- (3) Press the [PANCH] soft key.
- (4) Press the [(FILE#)] soft key.

- (5) Enter a file number.
- (6) Press the [PERIODMAINTE] soft key.

Method 4 (The output file is specified with its file number (two of three methods))

- (1) Select EDIT mode.
- (2) To output periodic maintenance data, display the item setting screen. To output the machine menu, display the item selection menu screen.
- (3) Press the [PANCH] soft key.
- (4) Press the address key.
- (5) Enter a file number.
- (6) Press the [PERIODMAINTE] soft key.

Method 5 (The output file is specified with its file number (three of three methods))

- (1) Select EDIT mode.
- (2) To output periodic maintenance data, display the item setting screen. To output the machine menu, display the item selection menu screen.
- (3) Press the address key.
- (4) Enter a file number.
- (5) Press the [] soft key.

NOTE	
1	If an item name is omitted for an item number, the data corresponding to the item number is not output.
2	Items selected and set from the CNC menu are not output.
3	When periodic maintenance data is output to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, or a memory card by using setting item output method 1 (when neither an output file name nor file number is specified), the file name is set to PMNTSTS.TXT.
4	When the machine menu is output to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, or a memory card by using setting item output method 1 (when neither an output file name nor file number is specified), the file name is set to PMNTMCN.TXT.
5	When a FANUC Floppy Cassette, FANUC FA Card, or FANUC Handy File is used for output, an output file can be specified using a file name or file number.
6	When a memory card is used for output, an output file can be specified using a file name.

Input/output file format

Periodic maintenance data

G10	L60	P1	m	nA	q	p	G
G10	L60	P2	m	nA	q	p	G
G10	L60	P3	m	nA	q	p	G
			E				
			E				
			E				

Item selection menu (machine) data

G10	L61	P1	m	n	G
G10	L61	P2	m	n	G
G10	L61	P3	m	n	G
			E		
			E		
			E		

G10 L60 . Specifies periodic maintenance data.

G10 L61 . Specifies item selection menu (machine) data.

P . Item number (1 to 10)

[n] . Item name enclosed in []. Item names must conform to the input format described previously.

A . Life time (0 to 999999)

R . Remaining time (0 to 999999)

- Q . Count type
- 0 . No counting
- 1 . All time
- 2 . Power on
- 3 . Automatic operation
- 4 . Cutting

Input from an NC program

Data can also be input by executing an input/output file as an NC program.

NOTE

- 1 No macro statement can be specified in a G10L60(61) command block.
- 2 An item name must be specified following G10L60(61). If an item name is specified in an invalid position, the item name is assumed to have been omitted. As a result, that data having the corresponding item number is deleted.

Input/output on another screen

Data input/output on the periodic maintenance screens can also be input/output on the floppy disk list screen and memory card screen. For details, see Section III.x.x, "Floppy Disk List Screen," and Section III.x.x, "Memory Card Screen."

Alarm

Number	Message	Contents
SR0125	ILLEGAL EXPRESSION FORMAT	The description of the custom macro statement is erroneous. The format of the parameter data is erroneous.

7.4 MAINTENANCE INFORMATION SCREEN

The maintenance information screen allows service persons from FANUC and the machine tool builder to record a maintenance history of the machine.

For details, refer to "FANUC Series 15i/150i-MODEL A Maintenance Manual" (B-63325JA).

7.5 SYSTEM LOG SCREEN

When a system alarm is issued, displayed information is recorded in backup memory. The recorded information can then be checked and downloaded to a memory card and host computer even after the machine is started again.

Display

Follow the procedure described in Section III.x.x, "System" to display the system log screen. The displayed information is almost the same as that displayed on the system alarm screen.

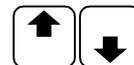
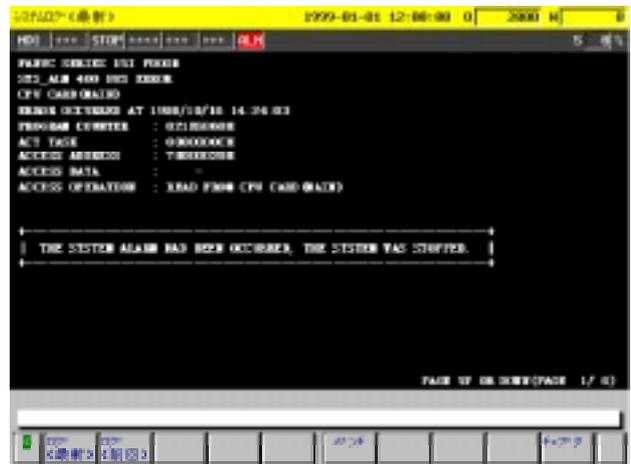


Fig.7.5 (a) Examples of the System Alarm Screen (Left) and System Log Screen (Right)

Selecting display data

Page change

Use the page change keys to display the previous

and next pages.

Selecting a system log

The past two system logs are recorded. To change the displayed system log, use the [LOG (NEW)] or [LOG (OLD)] soft key.

NOTE

When no system log has been saved into backup memory, the message "NO LOG DATA" is displayed. If the data saved into backup memory is illegal, a message "CAN NOT DISP LOG DATA" is displayed.

Outputting a system log

Method 1 (when neither an output file name nor file number is specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [LOG (NEW)], [LOG (OLD)], [ALL], or [THIS DATA] soft key.

Method 2 (when an output file name is specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Key in a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [LOG (NEW)], [LOG (OLD)], [ALL], or [THIS DATA] soft key.

NOTE

- 1 When the [LOG (NEW)], [LOG (OLD)], [ALL], or [THIS DATA] soft key is pressed, the following is output:
When the [LOG (NEW)], [LOG (OLD)], [ALL], or [THIS DATA] soft key is pressed, the following is output:
- 2 When tool offset data is output to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, or a memory card using method 1 (when neither an output file name nor file number is specified), the following file name is assigned:
System log (new): SYSLOG_N.TXT
System log (old): SYSLOG_O.TXT
When both the new and old system logs are output at the same time:
SYSLOG_A.TXT
- 3 When no system log is saved in backup memory, or when the saved data is illegal, the warning message "DATA NOT FOUND" is displayed.

A system log can be output not only to this screen, but also to the floppy disk list screen and memory card screen. For details, see Section III.x.x, "Floppy Disk List Screen," and Section III.x.x, "Memory Card Screen."

7.5.1 Displaying the contents of memory

The CNC memory contents, starting from a specified address, can be displayed.

For details, refer to the "FANUC Series 15i/150i-MODEL A Maintenance Manual" (B-63325JA).

NOTE

This screen is used for CNC maintenance.
Do not use this screen for any other purpose.

7.5.2

e \ f

CNC.....

.....FANUC Series 15i/150i-MODEL A(B-63325JA).....

```
{  * CNC      - g p      • B
» Ⓔ " O -   g p ~ • ¥ ~ ~ s " ç - " † ç B
```


8.1 INPUTTING AND OUTPUTTING PROGRAMS

This section explains the operations for inputting programs from and outputting programs to external input/output units, as well as the related functions.

8.1.1 Inputting programs

Programs can be registered from external input/output units.

Registering a single program

Procedure

Method 1 (Neither the input file name nor number need be specified)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the [1_PROGRAM] soft key.
- (5) Press the [NEW] soft key.

Method 2 (The input file is specified with its file name)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [1_PROGRAM] soft key.
- (8) Press the [NEW] soft key.

Method 3 (The input file is specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [1_PROGRAM] soft key.
- (7) Press the [NEW] soft key.

Method 4 (The input file is specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the address key.
- (5) Enter a file number.
- (6) Press the [1_PROGRAM] soft key.
- (7) Press the [NEW] soft key.

Registering a program with a new program number

A program can be registered with a new keyed-in program number.

Procedure

£ Methods in which neither the input file name nor number need be specified

Method 1

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the [(PROGRAM#)] soft key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 2

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the address key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 3

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.

- (3) Press  the address key.
- (4) Enter a program number.
- (5) Press the [READ] soft key.

£ Methods in which the input file is specified with its file name

Method 1

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [(PROGRAM#)] soft key.
- (8) Enter a program number.
- (9) Press the [EXEC] soft key.

Method 2

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press  the address key.
- (8) Enter a program number.
- (9) Press the [EXEC] soft key.

£ Methods in which the input file is specified with its file number

Method 1

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [(PROGRAM#)] soft key.

- (7) Enter a program number.
- (8) Press the [EXEC] soft key.

Method 2

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the address key.
- (5) Enter a file number.

- (6) Press the address key.
- (7) Enter a program number.
- (8) Press the [EXEC] soft key.

Method 3

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the address key.
- (4) Enter a file number.
- (5) Press the address key.
- (6) Enter a program number.
- (7) Press the [READ] soft key.

Registering multiple NC programs

Procedure

Method 1 (Neither the input file names nor numbers need be specified)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the [ALL] soft key.

Method 2 (The input files are specified with their file names)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [ALL] soft key.

Method 3 (The input files are specified with their file numbers (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [ALL] soft key.

Method 4 (The input files are specified with their file numbers (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press  the address key.
- (5) Enter a file number.
- (6) Press the [ALL] soft key.

Registering additional programs

A program can be loaded and merged with the program selected on the CNC.

Procedure

Method 1 (Neither the input file name nor number need be specified)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.

- (4) Press the [1_PROGRAM] soft key.
- (5) Press the [ADD] soft key.

Method 2 (The input file is specified with its file name)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [1_PROGRAM] soft key.
- (8) Press the [ADD] soft key.

Method 3 (The input file is specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [1_PROGRAM] soft key.
- (7) Press the [ADD] soft key.

Method 4 (The input file is specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [READ] soft key.
- (4) Press  the address key.
- (5) Enter a file number.
- (6) Press the [1_PROGRAM] soft key.
- (7) Press the [ADD] soft key.

Stopping registration

While a program is being loaded, pressing the [STOP] soft key causes loading of the program to stop. Once stopped, loading cannot be resumed.

Explanation

£ Program end codes in files and program numbers

Usually, M02, M30, and M99 are program end M codes. The available program end codes differ depending on the settings of parameter NPE (bit 3 of No. 2200) and NM9 (bit 4 of No. 2200), as shown below.

Depending on the available program end codes, the program numbers differ, as does the way in which programs are separated.

Table 8.1.1 (b) Program End M Codes

Parameter	NPE=0 ◊NM9=0	NPE=0 ◊NM9=1	NPE=0 ◊NM9=0 or 1
Program end M code	M02, M30, M99	M02, M30	None
Program number	O number or the sequence number of the start block		O number only
Single program	<ul style="list-style-type: none"> - Program number to program end - Program number to program end Program number to the block immediately preceding the O number 		O number to the block immediately preceding the O number

£ Registerable programs

Programs from 00000001 to 99999999 can be registered. How programs are to be handled is determined with the following parameters:

Parameter RDL (bit 0 of No. 2200)

When all programs are to be registered in the foreground at the same time

0: The programs are added after the previously registered programs

1: The programs are registered after the previously registered programs are deleted.

Parameter REP (bit 1 of No. 2200)

Programs with the same program numbers as those registered previously:

0: Cannot be registered.

1: Registered after those registered previously are deleted.

Parameter NE8 (bit 0 of No. 0011)

Programs with numbers 8000 to 8999:

0: Can be edited (can be registered).

1: Cannot be edited (cannot be registered).

Parameter NE9 (bit 0 of No. 2201)

Programs with numbers 9000 to 9999:

0: Can be edited (can be registered).

1: Cannot be edited (cannot be registered).

Warning

Message	Description
ALREADY EXISTS	An attempt was made to register a program with the same program number as that of a previously registered program in program memory.
WRITE PROTECT	An attempt was made to register a program that is write-protected with parameter NE8 (bit 0 of No. 0011) or NE9 (bit 0 of No. 2201).

8.1.2 OUTPUTTING PROGRAMS

Programs can be output to external input/output units.

Outputting the currently displayed program

Procedure

Method 1 (Neither the output file name nor number need be specified)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the [THIS PROGRAM] soft key.

Method 2 (The output file is specified with its file name)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [THIS PROGRAM] soft key.

Method 3 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [THIS PROGRAM] soft key.

Method 4 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press  the address key.
- (5) Enter a file number.
- (6) Press the [THIS PROGRAM] soft key.

Outputting a specified single program

Procedure

£ Neither the input file name
nor number need be specified

Method 1

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the [(PROGRAM#)] soft key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 2

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press  the address key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 3

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press  the address key.
- (4) Enter a program number.
- (5) Press the [PUNCH] soft key.

£ The output file is specified with its file name

Method 1

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.

- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [(PROGRAM#)] soft key.
- (8) Enter a program number.
- (9) Press the [EXEC] soft key.

Method 2

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the address key.
- (8) Enter a program number.
- (9) Press the [EXEC] soft key.

£ Specified with its file number**Method 1**

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [(PROGRAM#)] soft key.
- (7) Enter a program number.
- (8) Press the [EXEC] soft key.

Method 2

- (1) Select xxx mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the address key.
- (5) Enter a file number.
- (6) Press the address key.

- (7) Enter a program number.
- (8) Press the [EXEC] soft key.

Method 3

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the address key.
- (4) Enter a file number.
- (5) Press the address key.
- (6) Enter a program number.
- (7) Press the [PUNCH] soft key.

Outputting specified multiple programs

A specified range of programs can be output.

Procedure

£ Neither the output file name nor number need be specified

Method 1

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the [(PROGRAM#)] soft key.
- (5) Enter the number of the first program to be output.
- (6) Press the [, (PROGRAM#)] soft key.
- (7) Enter the number of the last program to be output.
- (8) Press the [EXEC] soft key.

Method 2

- (1) Select xxx mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the address key.

(5) Enter the number of the first program to be output.

(6) Press the address key.

(7) Press the address key.

(8) Enter the number of the last program to be output.

(9) Press the [EXEC] soft key.

Method 3

(1) Select EDIT mode.

(2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.

(3) Press the address key.

(4) Enter the number of the first program to be output.

(5) Press the address key.

(6) Press the address key.

(7) Enter the number of the last program to be output.

(8) Press the [PUNCH] soft key.

£ The output file is specified with its file name

Method 1

(1) Select EDIT mode.

(2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.

(3) Press the [PUNCH] soft key.

(4) Press the ["FILE NAME] soft key.

(5) Enter a file name.

(6) Press the [FILE NAME"] soft key.

(7) Press the [(PROGRAM#)] soft key.

(8) Enter the number of the first program to be output.

(9) Press the [, (PROGRAM#)] soft key.

(10) Enter the number of the last program to be output.

(11) Press the [EXEC] soft key.

Method 2

(1) Select EDIT mode.

- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the address key.
- (8) Enter the number of the first program to be output.
- (9) Press the address key.
- (10) Press the address key.
- (11) Enter the number of the last program to be output.
- (12) Press the [EXEC] soft key.

£ Specified with its file number

Method 1

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [(PROGRAM#)] soft key.
- (7) Enter the number of the first program to be output.
- (8) Press the [, (PROGRAM#)] soft key.
- (9) Enter the number of the last program to be output.
- (10) Press the [EXEC] soft key.

Method 2

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the address key.
- (5) Enter a file number.
- (6) Press the address key.

(7) Enter the number of the first program to be output.

(8) Press the address key.

(9) Press the address key.

(10) Enter the number of the last program to be output.

(11) Press the [EXEC] soft key.

Method 3

(1) Select EDIT mode.

(2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.

(3) Press the address key.

(4) Enter a file number.

(6) Press ...the address key.

(6) Enter the number of the first program to be output.

(7) Press ...the address key.

(8) Press ...the address key.

(9) Enter the number of the last program to be output.

(10) Press the [PUNCH] soft key.

Outputting all programs

Procedure

Method 1 (Neither the output file name nor number need be specified)

(1) Select EDIT mode.

(2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.

(3) Press the [PUNCH] soft key.

(4) Press the [ALL] soft key.

Method 2 (The output file is specified with its file name)

(1) Select EDIT mode.

(2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.

- (3) Press the [PUNCH] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [ALL] soft key.

Method 3 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [ALL] soft key.

Method 4 (Specified with its file number (one of two methods))

- (1) Select xxx mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [PUNCH] soft key.
- (4) Press  the address key.
- (5) Enter a file number.
- (6) Press the [ALL] soft key.

Outputting a portion of the currently displayed program

That portion of the currently displayed program that starts from the current cursor position can be output.

Procedure

Method 1 (Neither the output file name nor number need be specified)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Position the cursor to the beginning of the portion you want to punch out.
- (4) Press the [PUNCH] soft key.
- (5) Press the [HERE] soft key.

Method 2 (The output file is specified with its file name)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Position the cursor to the beginning of the portion you want to punch out.
- (4) Press the [PUNCH] soft key.
- (5) Press the ["FILE NAME] soft key.
- (6) Enter a file name.
- (7) Press the [FILE NAME"] soft key.
- (8) Press the [HERE] soft key.

Method 3 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Position the cursor to the beginning of the portion you want to punch out.
- (4) Press the [PUNCH] soft key.
- (5) Press the [(FILE#)] soft key.
- (6) Enter a file number.
- (7) Press the [HERE] soft key.

Method 4 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Position the cursor to the beginning of the portion you want to punch out.
- (4) Press the [PUNCH] soft key.

(5) Press  the address key.

- (6) Enter a file number.
- (7) Press the [HERE] soft key.

Pausing/stopping the output and canceling the feed

Pausing the output

While a program is being output, pressing the [PAUSE] soft key causes output of the program to pause. Subsequently, the output can be resumed, or the program can be output one block at a time.

- (1) While a program is being output, press the [PAUSE] soft key.
The output pauses after the block being output when the pause request is accepted has been output. A slight delay may occur prior to communication actually stopping.
- (2) To resume the output of the program, press the [EXEC] soft key.
- (3) To output a single block, press the [EXEC SINGLE] soft key. The output pauses again.
- (4) To end program output, press the [%_FEED] soft key. A % (end of record) and feed (portion without holes) are output before program output ends.

Stopping the output

While a program is being output, pressing the [STOP] soft key causes the output of the program to stop. Once stopped, output cannot be resumed.

Canceling the feed

While the leading or trailing feed (portion without holes) of tape is being output, pressing the [FEED CANCEL] soft key causes output of the feed to stop.

While the leading feed is being output: The program is output immediately.

While the trailing feed is being output: The feed output stops immediately.

8.1.3 Collating programs

Programs entered from external input/output units can be collated with programs in program memory.

Collating a single program

Procedure

Method 1 (Neither the file name nor number need be specified)

- (1) Select EDIT mode.

- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [VERIFY] soft key.
- (4) Press the [1_PROGRAM] soft key.

Method 2 (Specified with its file name)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [VERIFY] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [1_PROGRAM] soft key.

Method 3 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [VERIFY] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [1_PROGRAM] soft key.

Method 4 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [VERIFY] soft key.
- (4) Press  the address key.
- (5) Enter a file number.
- (6) Press the [1_PROGRAM] soft key.

Explanation

A program loaded from an external input/output unit can be collated with a program retrieved from program memory.

If no mismatches are found, the collated program is displayed and collation ends.

If a mismatch is found, alarm PS400 PROGRAM NOT MATCH is issued and the cursor is positioned to the mismatching word.

Collating multiple programs successively

Procedure

Method 1 (Neither the file name nor number need be specified)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [VERIFY] soft key.
- (4) Press the [ALL] soft key.

Method 2 (Specified with its file name)

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [VERIFY] soft key.
- (4) Press the ["FILE NAME] soft key.
- (5) Enter a file name.
- (6) Press the [FILE NAME"] soft key.
- (7) Press the [ALL] soft key.

Method 3 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [VERIFY] soft key.
- (4) Press the [(FILE#)] soft key.
- (5) Enter a file number.
- (6) Press the [ALL] soft key.

Method 4 (Specified with its file number (one of two methods))

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [VERIFY] soft key.
- (4) Press  the address key.
- (5) Enter a file number.
- (6) Press the [ALL] soft key.

Explanation

Multiple programs can be loaded from an external input/output unit one after another and collated with programs retrieved from program memory, one program at a time.

If no mismatches are found, the program collated last is displayed and collation ends.

If a mismatch is found, alarm PS400 PROGRAM NOT MATCH is issued and the cursor is positioned to the mismatching word.

NOTE

When parameter NPE (bit 3 of No. 2200) is equal to 1, multiple programs cannot be collated one after another.

Resuming the collation

After a mismatch is found in a program, collation can be resumed.

Procedure

- (1) If a mismatch is found, alarm PS400 PROGRAM NOT MATCH is issued.

In this state, the program displayed on the program text screen is that loaded from the external input/output unit. The cursor is positioned to the mismatching word.

- (2) Press the [CANCEL] soft key on the alarm screen to cancel the alarm.

After the alarm is canceled, the program displayed on the program text screen is that registered in program memory.

- (3) On the program text screen, check the mismatch (word to which the cursor is positioned) in the program registered in program memory.

- (4) If the mismatch is the word immediately preceding ;(EOB) or is ;(EOB), position the cursor to the first word in the block after the next.

Otherwise, position the cursor to the first word in the next block.

- (5) Press the [VERIFY] soft key.
- (6) Press the [HERE] soft key.

Alarm

Number	Message	Contents
PS0400	PROGRAM NOT MATCH	The program in memory does not match the program stored on tape. Multiple programs cannot be matched continuously when parameter No. 2200#3 is set to "1". Set parameter No. 2200#3 to "0" before executing a match.

8.1.4 Keys and program encryption

Programs can be secured by setting a password and a program security range with the appropriate parameters.

1 Security with a password and a security range

When the password and security range parameters are specified, the display, editing, and input/output operations are disabled for the programs within the security range.

This prevents the custom macro programs created by the machine tool builder from being accidentally changed or deleted by the end user. This also provides security because the contents of the programs are not displayed.

2 Input/output of encrypted programs

The programs in the security range can be encrypted before being output. Once encrypted, the programs cannot be decrypted. In addition, encrypted programs can be input directly.

This enables machine tool builders to encrypt their custom macro programs and supply the encrypted programs for maintenance purposes to the machines to be delivered. When programs are supplied in this way, security is ensured when the contents of the program memory are lost for some reason and FANUC service personnel or end users need to perform restore operations.

Locking/unlocking

When the programs in the security range are secured, the program memory is said to be locked.

If they are not secured, the program memory is said to be unlocked.

Parameter

Parameter PSW (No. 2210)

Set the password required to lock the program memory. A password other than 0 can be set. The password is not displayed. A password can be set when no password is set (PSW = 0) or the program memory is unlocked.

Parameter KEY (No. 2211)

When KEY is set equal to PSW, the program memory is unlocked. The value set for the parameter is never displayed. Each time the CNC is started, 0 is set for the parameter. This

ensures that the program memory is locked whenever the CNC is started.

Parameters PMIN (No. 2212) and PMAX (No. 2213)

Set the desired security range. The value set for PMIN must not be greater than that set for PMAX. The programs in the range of PMIN to PMAX are secured. If PMIN is set to 0, it is regarded as being 9000. If PMAX is set to 0, it is regarded as being 9999.

A range can be set when no password is set or the program memory is unlocked.

NOTE

- 1 For security, the values set for PSW and KEY are not displayed. For the same reason, PSW, PMIN, and PMAX can be specified only when no password is set or the program memory is unlocked. Set a password, taking great care to avoid a situation where the program memory cannot be unlocked because a password is set incorrectly.
- 2 If the password is forgotten, the programs within the security range must be deleted to nullify the password.
- 3 The [+INPUT] key used to specify PSW and KEY behave in the same way as the [INPUT] soft key.

Example: When 99 is set for KEY, pressing



and [+INPUT] changes the value to.

- 4 These four parameters are not output with parameter output. They are ignored when an attempt is made to input them with parameter input.

Inputting/outputting and collating programs

When a program is encrypted, a password is output. The password is used to load the program.

£ **Outputting all programs**

Locked/unlocked	Results
Locked	All the programs outside the security range are output in the normal way.
Unlocked	All the programs within the security range are encrypted and output.
Password not set	All the programs in the program memory are output in the normal way.

£ **Outputting a single program**

Outputting specified multiple programs

Locked/unlocked	Results
Locked	If a program is outside the security range, it is output in the normal way. If it is within the security range, a warning is issued.
Unlocked	If a program is outside the security range, it is output in the normal way. If it is within the security range, it is encrypted and output.
Password not set	The program is output in a normal way.

£ **Inputting an un-encrypted program**

Locked/unlocked	Results
Locked	If the program is outside the security range, it is input. If it is within the security range, it is not input.
Unlocked, or password not set	The program is input.

£ **Inputting an encrypted program**

Password set in the system and password of the program	Results
Password set in the system	Alarm PS580 ENCODE ALARM (PSWD&KEY) is issued.
Password for the program	The program is input.
Password not set in the system	The program is input. The PSW in the file is set for parameter No. 2210.

£ **Collating a program with an un-encrypted program**

The program is always collated.

£ **Collating a program with an encrypted program**

Password set in the system and password of	Results
--	---------

the program	
Password set in the system	Alarm PS580 ENCODE ALARM (PSWD&KEY) is issued.
Password set in the system = Password of the program, or password not set in the system	The program is collated.

<p>NOTE</p> <p>1 To encrypt programs, set parameter EIA (bit 4 of No. 0000) to 0 (to specify that the punch code is ISO).</p> <p>2 An encrypted program cannot be registered as an additional program ([READ]-[1_PROGRAM]-[ADD]).</p> <p>3 Programs outside the security range cannot be input from encrypted programs.</p>
--

Program display

- 1 On the program directory screen, all program numbers are displayed together with comments.
- 2 On the program text screen:
 When the program memory is locked, programs within the security range are not displayed.
 When the program memory is unlocked, all the programs are displayed.
 The change from the unlocked state to the locked state or from the locked state to the unlocked state is not applied to the display immediately. The change takes effect when, for example, a program search is performed.

Editing and deleting programs

When the program memory is locked, the programs within the security range cannot be edited or deleted. When the program memory is locked, an attempt to delete all programs results in only those programs outside the security range being deleted.

Warning

Message	Description
WRITE PROTECTION	An attempt was made to edit or delete a program within the security range when the program memory was locked.
INVISIBLE PROGRAM	An attempt was made to select or output a program within the security range when the

	program memory was locked.
--	----------------------------

Alarm

Number	Message	Description
PS0580	ENCODE ALARM (PSWD&KEY)	An attempt was made to load or collate a program, but the password set in the system did not match the loaded password.
PS0581	ENCODE ALARM (PARAMETER)	An attempt was made to encrypt and output a program when the specified punch code is EIA (parameter EIA (bit 4 of No. 0000 = 1).

8.1.5 Stored program lengths and number of registerable programs

The following stored program lengths are available:

The number of registerable NC programs is as follows:

Stored program length	80	160	320	640	1280	2560	5120
Number of registerable programs (when expanded)	100(100)	100(200)	100(400)	100(1000)	100(1000)	100(1000)	100(1000)

8.2 Cursor

Editing operations such insertion, alteration, and deletion are performed on the character string to which the cursor is positioned. Basically, the cursor is displayed on a word such as a program number, G code command, or address command. Custom macro commands are not in form of words, in which case, the cursor is displayed differently.

This chapter explains the cursor and how to position it.

8.2.1 Cursor/editing units/words

Cursor and editing units

The cursor refers to a pointer, displayed in yellow in reverse video on a character string.

The character string to which the cursor is positioned is subject to an editing operation, and is called an editing unit. Editing units are separated by any of the following characters:

- Address
- First # on the left side of an assignment statement
- /, (, ;
- =
- First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT, and PCLOS

Editing units separated by characters other than = will have a single blank inserted at the beginning.

Editing units separated by = are connected to the preceding editing units.

Words and editing units

A word is an editing unit that consists of an address and numeric value (expression). Strictly speaking, "words" and "editing units" are different. In Section III.x, "EDITING PROGRAMS" in this chapter, the term word is used to mean editing unit.

8.2.2 Positioning the cursor

Registering a single program

- 1 Use  the...cursor key to position the cursor forward by one word. When the key is pressed and held down, the cursor moves repeatedly.
- 2 Use  the...cursor key to position the cursor back by one word. When the key is pressed and held down, the cursor moves repeatedly
- 3 Use  the...cursor key to position the cursor to the beginning of the next block. When the key is pressed and held down, the cursor moves repeatedly.
- 4 Use  the...cursor key to position the cursor to the beginning of the previous block. When the key is pressed and held down, the cursor moves repeatedly.
- 5 Use the...cursor  key to display the next page and position the cursor to the beginning of the page. When the key is pressed and held down, the cursor moves repeatedly.
- 6 Use the...cursor  key to display the previous page and position the cursor to the beginning of the page. When the key is pressed and held down, the cursor moves repeatedly.
- 7 Use either of the following operations to position the cursor to the end of the program.
 - (1) Press the [FRWRD SEARCH] soft key.
 - (2) Press the [BOTTOM] soft key.

- 8 Use either of the following operations to position the cursor to the beginning of the program.
 - (1) Press the [BKWRD SEARCH] soft key.
 - (2) Press the [TOP] soft key.

- 9 Press the [REWIND] soft key to position the cursor to the beginning of the program.

8.3 Program Number Search

A program can be selected from among those in program memory.

Searching for the immediately preceding (succeeding) program

Procedure

Method 1

- (1) Select EDIT or MEM mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.
- (4) Press the [NEXT PRGRAM] ([PRVIOS PRGRAM]) soft key.

Method 2

- (1) Select EDIT or MEM mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.
- (4) Press the [(PROG#)] soft key.
- (5) Press the [EXEC] soft key.

Method 3

- (1) Select EDIT or MEM mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press  the...address key.
- (4) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.

Searching for a program by specifying a program number

Procedure

Method 1

- (1) Select EDIT or MEM mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.
- (4) Press the [(PROG#)] soft key.

- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 2

- (1) Select EDIT or MEM mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.
- (4) Press the...address key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 3

- (1) Select EDIT or MEM mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the...address key.
- (4) Enter a program number.
- (5) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.

Warning

Message	Description
DATA NOT FOUND	The program for which a search was made cannot be found.

8.4 Word Search/Address Search

It is possible to search for a specified word, address, or character string through an NC program and position the cursor to it.

8.4.1 Word search

Word search

Procedure

Method 1

- (1) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.
- (2) Enter the character string to search for.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter the character string to search for.
- (2) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.

Specifiable character strings

- 1 NC commands
Examples: G00, X100., M99
- 2 Macro statements
Examples: #100=#102+1, IF[#300EQ0]GOTO100
- 3 Function names and variables
Examples: SIN in #1=SIN[#2], #102 in #100=#102+1
- 4 Portions of expressions
Examples: +1 in #100=#102+1, EQ in IF[#300EQ0]GOTO100
- 5 Symbols (only those listed below)
#, +,, =, :, [], (

Limitation

- 1 Even when an attempt is made to search for a portion of a word, such as a function name or variable, the cursor is positioned to the entire word.
- 2 The / and /n in an optional block skip are not distinguished from those in a macro statement.

- 3 Even comment sections are within the scope of the search.
Example: If an attempt is made to search for G90 through the program having a comment section (JOB1 G90MODE), the cursor will be positioned to the comment section.
- 4 It is not possible to search for an address command or function name by specifying a portion of it.
Example: It is not possible to search for X100. by specifying X1.
- 5 It is not possible to search for a command by specifying an equivalent command if the command and its equivalent are written differently.
Example: It is not possible to search for G1 by specifying G01.

Warning

Message	Description
DATA NOT FOUND	The character strings for which a search was made cannot be found.

8.4.2 Address search**Address search****Procedure****Method 1**

- (1) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.
- (2) Enter the address to search for.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter the address to search for.
- (2) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.

Limitation

Even comment sections are within a scope of a search.

Example: If an attempt is made to search for Y through the program having the comment section (JOB2 XY-PLANE), the cursor will be positioned to the comment section.

Warning

Message	Description
DATA NOT FOUND	The Address for which a search was made cannot be found.

8.4.3 Search repeat function

Once a word/address search has been performed, the search can be repeated under the same conditions with a simple operation.

Search repeat function

Procedure

- (1) Perform a word or address search.
- (2) Press the [FRWRD SEARCH] ([BKWRD SEARCH]) soft key.
- (3) Press the [REPEAT] soft key.

Warning

Message	Description
CAN NOT REPEAT (FUNCTION IS DIFFERENT)	On the current screen or in the current operation mode, the search cannot be repeated.
CAN NOT REPEAT (BUFFER IS EMPTY)	An attempt was made to use the repeat function without first performing a word/address search.

8.5 Inserting words and blocks

Words and blocks can be inserted at the cursor position.

Inserting a word or block

Procedure

- (1) Position the cursor to the point where you want to insert a word or block.
- (2) Key in the word you want to insert.
Multiple words and blocks can be entered until the key-in buffer becomes full.
- (3) Press the  key.

Relationship between the cursor and the insertion position

The insertion position is determined by the setting of the following parameter:

Parameter NIS (bit 6 of No. 2218)

- 0: Inserts a word or block after the cursor(in conformance with FS15-B).
- 1: Inserts a word or block ahead of the cursor.

8.6 Changing words and blocks

- 1 The word at the cursor position can be overwritten with an entered word or block.
- 2 A word, address, or character string in the program being edited can be replaced.

8.6.1 Overwriting words

Overwriting words

Procedure

- (1) Position the cursor to the word to be overwritten.
- (2) Key in a new word.
Multiple words and blocks can be entered until the key-in buffer is full.

- (3) Press the  key.

Limitation

It is possible to replace a single word with multiple words or blocks. The procedure described in this section cannot, however, replace multiple words with a single word. To do this, see Section III.x.x.x, "Replacing words and character strings."

8.6.2 Replacing words and character strings

Confirming each replacement

Procedure

Method 1

- (1) Press the [ALTER] soft key.
- (2) Key in the word or character string to be replaced. A list of multiple words can also be keyed in.
- (3) Press the [(WORD)] soft key.
- (4) Key in a new word or character string. Multiple words can also be keyed in.
- (5) Press the [EXEC SINGLE] soft key.

The system searches for the character string to be replaced and the cursor is positioned to the string. If a list of multiple

words is to be replaced, the cursor is positioned to the last word.

- (6) To replace the character string, press the [EXEC SINGLE] soft key.

The character string is replaced and the system searches for the next occurrence of the character string.

- (7) To search for the next occurrence of the character string without performing replacement, press the [SKIP] soft key.

The system searches for the next occurrence without replacing the character string.

- (8) To replace all the remaining occurrences of the character string, press the [EXEC] soft key.

- (9) To end the replacement prior to the completion of the search, press the [STOP] soft key.

- (10) Once replacement ends, the total number of replacements is displayed.

Method 2

- (1) Press the [ALTER] soft key.

- (2) Key in the word or character string to be replaced, followed by a new word or character string, as shown in the example.

Multiple words to be replaced and multiple new words can be entered.

Example: To replace G90 with G91, enter G90,G91.

- (3) The subsequent steps are the same as (5) to (10) of method 1.

Method 3

- (1) Key in the word or character string to be replaced, followed by a new word or character string, as shown in the example.

Multiple words to be replaced and multiple new words can be entered.

Example: To replace G90 with G91, enter G90,G91.

- (2) Press the [ALTER] soft key.

The system searches for the character string to be replaced and the cursor is positioned to the string. If a list of multiple words is to be replaced, the cursor is positioned to the last word.

- (3) The subsequent steps are the same as (6) to (10) of method 1.

Replacing all the occurrences of a character string without confirming each occurrence

Procedure

Method 1

- (1) Press the [ALTER] soft key.
- (2) Key in the word or character string to be replaced. A list of words can also be keyed in.
- (3) Press the [(WORD)] soft key.
- (4) Key in a new word or a character string. Multiple words can also be keyed in.
- (5) Press the [EXEC] soft key.
The system searches for the first occurrence of the character string to be replaced and the cursor is positioned to the string. If a list of multiple words is to be replaced, the cursor is positioned to the last word.
- (6) Press the [EXEC] soft key.
All the occurrences of the character string are replaced.
- (7) To end the replacement before the completion of the search, press the [STOP] soft key.
- (8) When replacement ends, the total number of replacements is displayed.

Method 2

- (1) Press the [ALTER] soft key.
- (2) Key in the word or character string to be replaced, followed by a new word or character string, as shown in the example. Multiple words to be replaced and multiple new words can be entered.
Example: To replace G90 with G91, enter G90,G91.
- (3) The subsequent steps are the same as (5) to (8) in method 1.

Method 3

- (1) Key in the word or character string to be replaced, followed by a new word or character string, as shown in the example. Multiple words to be replaced and multiple new words can be entered.
Example: To replace G90 with G91, enter G90,G91.
- (2) Press the [ALTER] soft key.
The system searches for the first occurrence of the character string to be replaced and the cursor is positioned to the string. If a list of multiple words is to be replaced, the cursor is positioned to the last word.

(3) The subsequent steps are the same as (6) to (8) in method 1.

Character string that can be replaced

- 1 NC commands
Examples: G00, X100., M99
- 2 Macro statements
Examples: #100 = #102+1, IF[#300EQ0]GOTO100
- 3 Function names and variables
Examples: SIN in #1=SIN[#2], #102 in #100= #102+1
- 4 Portions of expressions
Examples: +1 in #100= #102+1, EQ in IF[#3000EQ0]
GOTO100
- 5 Symbols (only those listed below)
#.+......=.;[.](

Limitation

- 1 Only the words that appear after the word to which the cursor is positioned when replacement starts can be replaced. If the word to which the cursor is positioned when replacement starts is the same as that to be replaced, that word will not be replaced.
- 2 Even when a portion of a word, such as a function name or variable, is to be replaced, the cursor is positioned to the entire word.
- 3 The / and /n in an optional block skip are not distinguished from those in a macro statement.
- 4 Even comment sections are within the scope of replacement.
Example: If an attempt is made to replace G90 in a program having the comment section (JOB1 G90MODE), the G90 in the comment section can be replaced.
- 5 It is not possible to replace an address command or function name by specifying a portion of it.
Example: It is not possible to replace DPRNT with BPRNT, using DP,BP [CHANGE].
- 6 It is not possible to replace a command by specifying an equivalent command if the command and its equivalent are written differently.

Example: It is not possible to replace G1 with G02 by specifying G01,G02 [CHANGE].

Warning

Message	Description
DATA NOT FOUND	Not a single occurrence of the character string to be replaced can be found.

8.6.3 Replacing addresses

Confirming each replacement

Procedure

Method 1

- (1) Press the [ALTER] soft key.
- (2) Key in the address to be replaced.
- (3) Press the [(Address)] soft key.
- (4) Key in a new address.
- (5) Press the [EXEC SINGLE] soft key.
The system searches for the address to be replaced and the cursor is positioned to the address.
- (6) To replace the address, press the [EXEC SINGLE] soft key.
The address is replaced and the system searches for the next occurrence of the address.
- (7) To search for the next occurrence of the address without performing replacement, press the [SKIP] soft key.
The system searches for the next occurrence without replacing the address.
- (8) To replace all the remaining occurrences of the address, press the [EXEC] soft key.
- (9) To end the replacement prior to the completion of the search, press the [STOP] key.
- (10) When replacement ends, the total number of replacements is displayed.

Method 2

- (1) Press the [ALTER] soft key.
- (2) Key in the address to be replaced, followed by a new address, as shown in the example.
Example: To replace U with X, enter U,X.
- (3) The subsequent steps are the same as (5) to (10) of method 1.

Method 3

- (1) Key in the address to be replaced, followed by a new address, as shown in the example.

Example: To replace U with X, enter U,X.

- (2) Press the [ALTER] [soft key].

The system searches for the address to be replaced and the cursor is positioned to the address.

- (3) The subsequent steps are the same as (6) to (10) of method 1.

Replacing all the occurrences of an address without confirmation

Procedure**Method 1**

- (1) Press the [ALTER] soft key.
- (2) Key in the address to be replaced.
- (3) Press the [(ADDRESS)] soft key.
- (4) Key in a new address.
- (5) Press the [EXEC] soft key.

The system searches for the first occurrence of the address and the cursor is positioned to the address.

- (6) Press the [EXEC] soft key.

All the occurrences of the address to be replaced are replaced.

- (7) To end the replacement prior to the end of the search, press the [STOP] soft key.
- (8) When the replacement ends, the total number of replacements is displayed.

Method 2

- (1) Press the [ALTER] soft key.
- (2) Key in the address to be replaced, followed by a new address, as shown in the example.
Example: To replace U with X, enter U,X.
- (3) The subsequent steps are the same as (5) to (8) of method 1.

Method 3

- (1) Key in the address to be replaced, followed by a new address, as shown in the example.

Example: To replace U with X, enter U,X.

- (2) Press the [ALTER] soft key.

All the occurrences of the address to be replaced are replaced.

- (3) The subsequent steps are the same as (6) to (8) of method 1.

Limitation

- 1 Only those words that appear after that to which the cursor is positioned when replacement starts can be replaced. If the word to which the cursor is positioned when replacement starts is the same as that to be replaced, that word will not be replaced.
- 2 Even comment sections are within a scope of replacement.
Example: If an attempt is made to replace Y in the program having the comment section (JOB2 XY-PLANE), the Y in the comment section can be replaced.

Warning

Message	Description
DATA NOT FOUND	Not a single occurrence of the address to be replaced can be found.

8.7 DELETING WORDS AND BLOCKS

The word at the cursor position can be deleted, as well as selected multiple words and blocks.

Deleting a single word

Procedure

(1) Position the cursor to the word to be deleted.

(2) Press the  key.

Deleting multiple words or blocks

Use cutting and pasting to delete multiple words or blocks. See "Deleting multiple words or blocks" in Section III-9.xx, "Cutting and Pasting."

8.8 CUTTING AND PASTING

The following four basic operations can be combined to copy and move words and blocks.

- 1 Move a specified range to the copy buffer. (Cutting)
- 2 Copy a specified range to the copy buffer. (Copying)
- 3 Insert the contents of the copy buffer. (Pasting)
- 4 Use the contents of the copy buffer to create a new program. (Pasting a program)

8.8.1 Basic operations

Cutting

Procedure

- (1) Position the cursor to the first word of the range to be moved.
- (2) Press the [SELECT] soft key.
- (3) Position the cursor and select the range to be moved, which is displayed in white in reverse video.
- (4) Press the [CUT] soft key.

Copying

Procedure

- (1) Position the cursor to the first word of the range to be copied.
- (2) Press the [SELECT] soft key.
- (3) Position the cursor and select the range to be copied, which is displayed in white in reverse video.
- (4) Press the [COPY] soft key.

Pasting

Procedure

Method 1

- (1) Position the cursor to the position at which the contents of the copy buffer are to be inserted.
- (2) Press the [PASTE] soft key.

- (3) Press the [HERE] soft key.

The contents of the copy buffer are inserted ahead of the cursor.

Method 2

- (1) Position the cursor to the position at which the contents of the copy buffer are to be inserted.
- (2) Press the [PASTE] soft key.
- (3) Press the [(COUNT)#] soft key.
- (4) Key in the desired number of times.
- (5) Press the [HERE] soft key.

The contents of the copy buffer are inserted ahead of the cursor the specified number of times.

Method 3

- (1) Position the cursor to the position at which the contents of the copy buffer are to be inserted.
- (2) Press the [PASTE] soft key.

- (3) Press the  address key.

- (4) Key in the desired number of times.
- (5) Press the [HERE] soft key.

The contents of the copy buffer are inserted ahead of the cursor the specified number of times.

Pasting a program

Procedure

Method 1

- (1) Press the [PASTE] soft key.
- (2) Press the [(PROG)#] soft key.
- (3) Enter a program number.
- (4) Press the [EXEC] soft key.

A program is created with the contents of the copy buffer.

Method 2

- (1) Press the [PASTE] soft key.
- (2) Press the [(COUNT)#] soft key.
- (3) Key in the desired number of times.
- (4) Press the [(PROG)] soft key.
- (5) Enter a program number.

- (6) Press the [EXEC] soft key.

A program is created, with the contents of the copy buffer being copied the specified number of times.

Method 3

- (1) Press the [PASTE] soft key.

- (2) Press the address key.

- (3) Key in the desired number of times.

- (4) Press the [(PROG)#] soft key.

- (5) Enter a program number.

- (6) Press the [EXEC] soft key.

A program is created, with the contents of the copy buffer being copied the specified number of times.

NOTE

- 1 The copy buffer can contain hold up to about 4000 characters. If an attempt is made to cut or copy words or blocks exceeding this capacity, the warning "NO MORE SPACE" is issued. When this warning is issued, nothing will be cut or copied.
- 2 There is only one copy buffer. If, after the copy buffer has been used in EDIT mode, the copy buffer is used in MDI mode, the contents of the buffer in EDIT mode will be lost.
- 3 The contents of the copy buffer are retained upon reset, mode change, or screen change, but are lost when the power is turned off.
- 4 If, while a program range is selected, a mode change or reset is performed, the range is automatically deselected.

Warning

Message	Description
NO MORE SPACE	An attempt was made to cut or copy words or blocks exceeding the capacity of the copy buffer.

8.8.2 Copying, moving, and deleting words and blocks

Copying a word or block

Procedure

- (1) Copy the word or block to be copied.
- (2) Select the program to which it is to be copied.

- (3) Position the cursor to the desired point.
- (4) Paste the word or block.

Moving a word or block

Procedure

- (1) Copy the word or block to be moved.
- (2) Select the program to which it is to be moved.
- (3) Position the cursor to the desired point.
- (4) Paste the word or block.

Deleting multiple words or blocks

Procedure

The effects of accidentally deleting and inserting words and blocks can be canceled. This is called undoing.

8.9 CANCELING THE EFFECTS OF EDITING WORDS AND BLOCKS

The effects of accidentally deleting and inserting words and blocks can be canceled. This is called undoing.

Explanation

While a program is being edited in EDIT/MDI mode, the immediately preceding editing operation, such as deletion or insertion, is stored.

Undoing refers to the function that cancels the effect of the immediately preceding editing operation.

The following editing operations can be canceled by undoing.

- . Deletion, overwriting, and insertion of words
- . Cutting and pasting

Other operations (such as the replacement of words and the alteration of program numbers) cannot be canceled.

Stored editing operation

The stored editing operation is erased upon any of the following operations:

- . Restoring a program to its previous state by undoing.
- . Replacing a word or address in the program on which the editing operation has been performed.
- . Merging or copying the program on which the editing operation has been performed.
- . Registering or deleting the program on which the editing operation has been performed.
- . Changing the program number of the program on which the editing operation has been performed.
- . Condense the program memory (in EDIT mode only).
- . Perform an operation or a reset (in MDI mode only).

Undoing

Procedure

- (1) Select the program you want to restore to its previous state by undoing.
- (2) Place the program in the same editing status as that when the editing operation was performed.

Example: To restore the program that was edited in the right portion of the screen in MULTI EDIT mode (simultaneous editing of two programs) in the foreground, place the program in the right portion of the screen in MULTI EDIT mode in the foreground.

(3) Press the [UNDO] soft key.

(4) Press the [EXEC] soft key.

The cursor returns to the point to which it was positioned when the editing operation was performed.

(5) Press the [EXEC] soft key.

Execute UNDO to return the program to its previous state.

NOTE

- 1 If UNDO is stopped while an NC program is being restored to its previous state with UNDO, the NC program returns to the state existing before UNDO was started. The stored editing operation is, however, erased.
- 2 UNDO may fail if restoring a program to its previous state would leave an extremely small amount of free space in program memory.
- 3 UNDO may fail if the program security parameters or DI signal differs from that when the editing operation was performed.

8.10 COPYING PROGRAMS

The entire contents of a single program can be copied to create a new one.

Copying a program

Procedure

Method 1

- (1) Select the program to be copied.
- (2) Press the [COPY PRGRAM] soft key.
- (3) Press the [(PROG)#] soft key.
- (4) Key in the program number of the program to be created.
- (5) Press the [EXEC] soft key.

Method 2

- (1) Select the program to be copied.
- (2) Press the [COPY PRGRAM] soft key.
- (3) Press the address key.
- (4) Key in the program number of the program to be created.
- (5) Press the [EXEC] soft key.

Method 3

- (1) Select the program to be copied.
- (2) Press the address key.
- (3) Key in the program number of the program to be created.
- (4) Press the [COPY PRGRAM] soft key.

8.11 MERGING PROGRAMS

The program being edited can be merged with any program.

Merging programs

Procedure

Method 1

- (1) Select the first program.
- (2) Press the [MERGE PRGRAM] soft key.
- (3) Press the [(PROG)#] soft key.
- (4) Key in the program number of the program to be merged with the first one.
- (5) Press the [EXEC] soft key.

Method 2

- (1) Select the first program.
- (2) Press the [MERGE PRGRAM] soft key.
- (3) Press the address key.
- (4) Key in the program number of the program to be merged with the first one.
- (5) Press the [EXEC] soft key.

Method 3

- (1) Select the first program.
- (2) Press the address key.
- (3) Key in the program number of the program to be merged with the first one.
- (4) Press the [MERGE PRGRAM] soft key.

8.12 DELETING PROGRAMS

Any program can be deleted.

Deleting a single program

Procedure

Method 1

- (1) Press the [DELETE PRGRAM] soft key.
- (2) Press the [(PROG)#] soft key.
- (3) Key in the program number of the program you want to delete.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Press the [DELETE PRGRAM] soft key.
- (2) Press the address key.
- (3) Key in the program number of the program you want to delete.
- (4) Press the [EXEC] soft key.

Method 3

- (1) Press the address key.
- (2) Key in the program number of the program you want to delete.
- (3) Press the [DELETE PRGRAM] soft key.

Deleting the currently selected program

Procedure

- (1) Press the [DELETE PRGRAM] soft key.
- (2) Press the [THIS] soft key.

Deleting multiple programs

Multiple programs can be deleted by specifying a range of program numbers.

Procedure

Method 1

- (1) Press the [DELETE PRGRAM] soft key.
- (2) Press the [(PROG)#] soft key.
- (3) Key in the number of the first program you want to delete.
- (4) Press the [(PROG)#] soft key.
- (5) Key in the number of the last program you want to delete.
- (6) Press the [EXEC] soft key.

Method 2

- (1) Press the [DELETE PRGRAM] soft key.
- (2) Press the address key.
- (3) Key in the number of the first program you want to delete.
- (4) Press the address key.
- (5) Press the address key.
- (6) Key in the number of the last program you want to delete.
- (7) Press the [EXEC] soft key.

Method 3

- (1) Press the address key.
- (2) Key in the number of the first program you want to delete.
- (3) Press the address key.
- (4) Press the address key.
- (5) Key in the number of the last program you want to delete.
- (6) Press the [DELETE PRGRAM] soft key.

Deleting all programs

Procedure

- (1) Press the [DELETE PRGRAM] soft key.
- (2) Press the [ALL] soft key.

8.13 RESTORING A DELETED PROGRAM

A program deleted accidentally can be restored. This is called undeleting.

Explanation

A single program deleted accidentally can be restored. Restoration is not possible if:

- 1 Multiple or all programs are deleted.
- 2 A single program is deleted, followed by one of the following operations:
 - . Creation of a new program
 - . Editing of multiple pages (such as cutting and pasting multiple blocks)
 - . Condensation of the program memory
 - . Turning off of the power to the NC

Undeleting a program

Procedure

- (1) Press the [UNDEL PRGRAM] soft key.
- (2) Press the [EXEC] soft key.

NOTE

A program may not be undeleted if the program security parameters or the DI signal differs from that when the program was deleted.

8.14 BACKGROUND EDITING

The editing of a program while another program is being executed or edited is called background editing.

Background editing allows searching, editing, registration, and deletion of programs in the same way as in ordinary editing (foreground editing).

Starting background editing

Procedure

Method 1

- (1) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen or program directory screen.
- (2) Press the [BACK EDIT] soft key.
- (3) Press the [(PROG)#] soft key.
- (4) Enter a program number.
- (5) Press the [EXEC] soft key.

Method 2

- (1) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen or program directory screen.
- (2) Press the [BACK EDIT] soft key.
- (3) Press the  address key.
- (4) Enter a program number.
- (5) Press the [EXEC] soft key.

Method 3

- (1) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen or program directory screen.
- (2) Press the  address key.
- (3) Enter a program number.
- (4) Press the [BACK EDIT] soft key.

Method 4

- (1) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen or program directory screen.
- (2) Press the [BACK EDIT] soft key.
- (3) Press the [PRVIOSPRGRAM] soft key.
The program that last underwent background editing is selected.

Ending background editing

Procedure

Press the [BACKGROUND EDIT END] soft key.

Limitation

- 1 When an attempt is made to background edit the program that is currently in the foreground, a warning "PROGRAM IN OPERATION" may be issued.
- 2 When an attempt is made to operate a program that is currently being edited in the background, a warning "START REJECT (BACKGROUND EDITING)" may be issued.
- 3 Background editing does not allow the deletion of all programs.

Alarm

n background editing, PS/SR alarms may be issued in the same way as in program editing in the foreground. To distinguish them from alarms in the foreground, these alarms are turned into BG alarms. The alarms do not affect the foreground operations. They can be reset without affecting the background operation, by using the [CANCEL] soft key on the alarm screen.
Alarms in the foreground do not affect background editing.

Number	Message	Contents
PS0077	PROGRAM IN USE	An attempt was made in the foreground to execute a program being edited in the background. The currently edited program cannot be executed, so end editing and restart program execution.

Warning

Message	Description
PROGRAM IN OPERATION	An attempt was made to background edit the program currently selected in the foreground.

START REJECT (BACKGROUND EDITING)	An attempt was made to operate the program being background edited.
--------------------------------------	---

8.15 EDITING TWO PROGRAMS SIMULTANEOUSLY

The program text screen can be divided into right and left portions, so that two programs can be edited simultaneously.

Display

Fig.8.15 (b) Editing Two Programs Simultaneously

The program text screen can be divided into right and left portions, so that two programs can be edited simultaneously.

Starting the simultaneous editing of two programs

Procedure

Method 1

- (1) Press the [MULTI EDIT] soft key.
- (2) Press the [(PROG)#] soft key.
- (3) Enter a program number.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Press the [MULTI EDIT] soft key.
- (2) Press the  address key.
- (3) Enter a program number.
- (4) Press the [EXEC] soft key.

Method 3

- (1) Press  the...address key.
- (2) Enter a program number.
- (3) Press the [MULTI EDIT] soft key.

Method 4

- (1) Press the [MULTI EDIT] soft key.
- (2) Press the [PRVIOUS PRGRAM] soft key..
The program that was last edited in the right portion is selected.

When the simultaneous editing of two program is started, the program in the right portion always becomes active.

Replacing the right and left programs with each other

Procedure

Press the [RIGHT/LEFT] soft key.

Ending the simultaneous editing of two programs

Procedure

Press the [M_EDITEND] soft key.
The program edited in the left portion is always selected.

8.16 EDITING PROGRAMS IN OPERATION

During automatic operation, programs can be edited by stopping automatic operation with the feed hold or single block function. Automatic operation can then be resumed.

Editing programs in operation

Procedure

- (1) Stop automatic operation, using the feed hold or single block function. Make a note of the cursor position.
- (2) Enter EDIT mode to edit programs.
- (3) Return the cursor to the previous position, which you noted in step (1).
- (4) Return to the previous operation mode to resume automatic operation.

Limitation

The available editing operations depend on the type of the program that is being operated.

- 1 If a main program is being operated
.All editing operations are available
- 2 If a subprogram is being operated
.If a subprogram is being operated
.Editing is possible only if the operation has been memory operation.
.Editing is possible in word units only.

8.17 CONDENSING THE PROGRAM MEMORY

In program memory, NC programs are first registered in contiguous areas. When an NC program is edited, however, it is divided into several blocks. The more frequently it is edited, the more it is fragmented. When an NC program is fragmented, the speed of MEM operation will slow down and more program memory space is required.

For this reason, it is necessary to condense the program memory to collect the program into contiguous areas.

Condensing the program memory for the displayed NC program

Procedure

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section ..x.x, "Programs" to display the program text screen.
- (3) Press the [CONDENSE] soft key.
- (4) Press the [THIS] soft key.

Condensing the program memory for a specified single NC program

Procedure

Method 1

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [CONDENSE] soft key.
- (4) Press the [(PROG#)] soft key.
- (5)
- (6) Press the [EXEC] soft key.

Method 2

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section III.x.x, "Programs" to display the program text screen.
- (3) Press the [CONDENSE] soft key.
- (4) Press the  address key.

- (5) Enter the number of the program for which the program memory is to be condensed.
- (6) Press the [EXEC] soft key.

Method 3

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section ..x.x, "Programs" to display the program text screen.
- (3) Press the  address key.
- (4)
- (5) Press the [CONDENSE] soft key.

Condensing the program memory for all NC programs

Procedure

- (1) Select EDIT xxx mode.
- (2) Follow the procedure described in Section ..x.x, "Programs" to display the program text screen.
- (3) Press the [CONDENSE] soft key.
- (4) Press the [ALL] soft key.

8.18 CREATING PROGRAMS USING THE MDI PANEL

Using the functions described in Section .-9, "EDITING PROGRAMS," programs can be created in EDIT mode.

Creating a program using the MDI panel

Procedure

- (1) Select EDIT mode.
- (2) Follow the procedure described in Section ..x.x, "Programs" to display the program text screen.
- (3) Press the ...  address key.
- (4) Enter a program number.
- (5) Press  the...key.
- (6) Follow the operations described in Section .-9, "EDITING PROGRAMS" to create a program.

8.19 PLAYBACK

Playback is a function in which teach in mode is selected, so that the machine can be moved along an axis to the desired machine position, and the position is inserted into the machining program.

Explanations

Teach in mode is that mode in which both EDIT and manual modes are selected simultaneously. In this mode, the editing of NC programs and the manual movement of the machine along an axis can be performed simultaneously.

Teach in mode is divided into five operation modes depending on the manual mode that is selected. In each operation mode, the corresponding manual operation can be performed.

- 1 Teach in and jog feed mode
- 2 Teach in and incremental feed mode
- 3 Teach in and manual handle feed mode
- 4 Teach in and jog feed & manual handle feed mode
- 5 Teach in and incremental feed & manual handle feed mode

When teach in mode is selected, and an axis address is keyed in and a word is inserted, the axis address is inserted into the machining program, with the absolute coordinate position being automatically added. This allows the creation of a machining program using machine positions resulting from manual movement as program coordinates.

Apart from axis addresses, program addresses (such as O, N, G, M, S, T, P, Q, and R) can be edited with the same editing operations as those for ordinary program editing.

Creating a program in THCH IN mode

Procedure

- (1) Follow the procedure described in Section ..x.x, "Programs" to display the program text screen.
- (2) Select teach in mode.
When the system enters teach in mode, the program text screen changes to the dedicated layout.

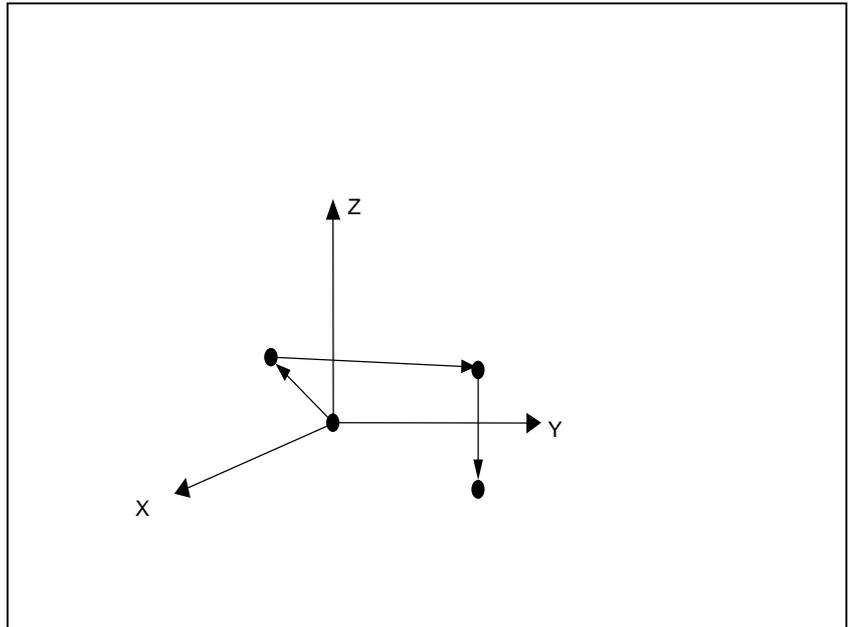
Fig.8.19 (c) Program Text Screen in Teach In Mode

- (3) Move the machine to the desired position, using jog, incremental, or handle feed.
- (4) Register or retrieve the number of the program to be edited, and position the cursor where you want to insert a machine position on each axis.
- (5) Key in an axis address and press the key. 

The absolute coordinate position of the entered axis address is loaded and inserted into the program as an axis command. The coordinates inserted with this method become absolute command values.

(Example) Absolute coordinate position X 12.300 (input in mm,
IS-B) X12.300

Exercise



```
O1234;
N1 G90 G00 X10. Y0 Z10.;
N2 G00 X3.025 Y23.723;
N3 G01 Z-0.325 F300.;
N4 M02;
```

Fig.8.19 (d) Sample Program

○ $(10.0, 0.0, 10.0)$

○ $(3.025, 23.723, 10.0)$

○ $(10.0, 0.0, 10.0)$

(1) Select teach in manual handle mode.

(2) Display the program text screen.



○ $(3.025, 23.723, -0.325)$

(3) Press the address key.

(4) Key in program number 1 2 3 4.

(5) Press the  key.

(6) Press the  key.

(7) Position the machine to P0, using the manual pulse generator.

(8) Register the absolute coordinate position of P0 as the data for the first block.

Key in N1G90G00XYZ, and then press the  and  keys. Block N1 is inserted.

(9) Position the machine to P1, using the manual pulse generator.

(10) Register the absolute coordinate position of P1 as the data for the second block.

Key in N2G00XY, and then press the  and 

keys. Block N2 is inserted.

(11) Register the absolute coordinate position of P2 as the data for the third block.

Key in N3G01ZF300, and then press the  and 

keys. Block N3 is inserted.

(12) Register the fourth block.

Key in N4M02, and then press the  and 

keys. Block N4 is inserted.

Registering machine positions with compensation

When, in teach in mode, an axis address is keyed in, followed by a numeric value, the keyed-in numeric value is added to the absolute coordinate position before the position is registered. This is useful for applying compensation to the position to be registered.

- 1 If the numeric value to be entered has more digits than the least input increment, it is rounded off to a value with the same number of digits as the least input increment.

(Example) When absolute coordinate position X0.0 (input in mm, IS-B) is keyed in, followed by X.1234567



and the key, "X1.235" will be inserted.

- 2 If the coordinates to be inserted exceed the maximum allowable command value, the warning "DATA IS OUT OF RANGE" is issued.

Limitation

- 1 In teach in mode, do not edit macro statements. If the commands used in macros contain a character matching the axis address, the system regards it as being the axis address and adds the absolute coordinate position.
(Example) When "#100= #100 XOR #101" is keyed in, the X in the exclusive-OR command "XOR" is regarded as being an axis address and the absolute coordinate position is added after the "X."
- 2 If an attempt is made to insert a block exceeding 200 characters, the warning "KEY-IN BUFFER FULL" is issued.
- 3 Parallel axis commands cannot be created.

Warning

Message	Description
DATA IS OUT OF RANGE	The coordinate value to be inserted exceeds the maximum specifiable value.
KEY-IN BUFFER FULL	An attempt was made to insert a block containing more than 200 characters

8.20 AUTOMATICALLY INSERTING SEQUENCE NUMBERS

During programming, the sequence number of the next block can be automatically inserted each time ;(EOB) is inserted.

The initial sequence number and the increment must be set for the appropriate parameters.

Automatically inserting sequence numbers

Procedure

- (1) Set the initial sequence number for parameter No. 0030 and the increment for parameter No. 0031. Set 1 for parameter SQN (bit 1 of No. 0010).
- (2) Create a new program.
- (3) The sequence number of the next block is inserted each

time is keyed in.

- . When multiple blocks are input, the sequence number of the next block is inserted if the last word is ;(EOB).
- . To change the sequence numbers to be automatically inserted in the middle, overwrite the automatically inserted sequence number immediately preceding the program end (%) with a new sequence number. The sequence numbers to be automatically inserted will not change even if a sequence number in the middle of the program or the sequence number keyed in from the MDI is overwritten.

NOTE

- 1 The sequence number returns to the initial value each time a new program is created. It does not return to the initial value when the system searches for an existing program.
- 2 The sequence number returns to the initial value when it exceeds N99999999.
- 3 Even if a sequence number is keyed in as a word from the MDI, the sequence numbers to be automatically inserted will not be affected.
- 4 If 64 or more characters are entered to the key-in buffer at one time, sequence numbers will not be inserted automatically.

8.21 STAMPING THE MACHINING TIME

When a program is executed during memory operation, the time required for the main program is measured and displayed, and the result is stamped (inserted) on the program as a comment section.

Displaying the machining time

The machining time is displayed on the machining time display screen, program directory screen, and tool path drawing screen.

£ Machining time display screen

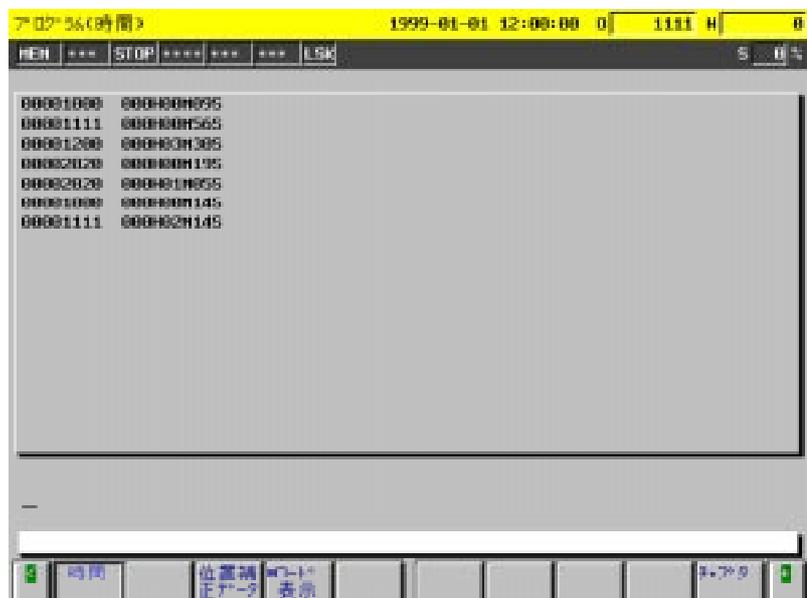


Fig.8.21 (a) Machining Time Display Screen

Follow the procedure described in Section ..x.x, "Programs" to display the machining time display screen. This screen displays the measurement results. The results for up to ten programs executed during memory operation are displayed, in the order in which the programs are executed. When more results are obtained, the results obtained first are discarded.

£ Program directory screen

Fig.8.21 (b) Program Directory Screen

Follow the procedure described in Section .x.x, "Programs" to display the program directory screen. On the program name list on this screen, stamped machining times are displayed instead of program sizes when parameter TIM (bit 4 of No. 0011) is set to 1. In the second or third word of a program number block, a comment section in the format of "(hhhHmmMssS)" is displayed as the machining time.

£ Tool path drawing screen

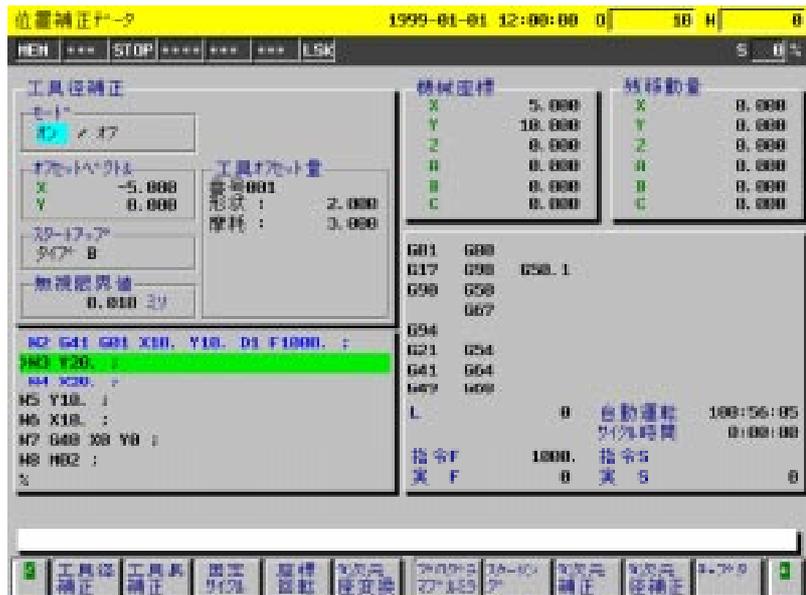


Fig.8.21 (c) Tool Path Drawing Screen

Follow the procedure described in Section III.x.x, "Drawing" to display the tool path drawing screen. This screen can display the time being measured in real-time.

Measuring the machining time

- 1 The time from the start of a program until a reset, M02, or M30 is measured as the machining time.
- 2 If, during measurement, an alarm occurs, causing the program to stop, the time until a reset is measured.
- 3 If the program is set up to be executed repeatedly, using M02 or M30, the time until the first FIN of M02 or M30 is returned is measured.

Stamping (inserting) the machining time

Procedure

- (1) Select EDIT mode.
 - (2) Follow the procedure described in Section ..x.x, "Programs" to display the program text screen.
 - (3) Follow the procedure described in Section ..x.x, "Program number search" to select the program into which the machining time is to be inserted.
 - (4) Press the [INSERT TIME] soft key.
- . If multiple measurement results exist for a single program on the machining time display screen, the newest measurement result will be stamped.
 - . If an attempt is made to stamp the machining time on a program for which no measurement result is displayed on the machining time display screen, the attempt is ignored.
 - . If the program number block consists of multiple words, the time is stamped as the third word.

9

SETTING AND DISPLAYING DATA

9.1 DISPLAYING AND SETTING TOOL OFFSET VALUES

Tool offset values, tool length compensation values, and cutter compensation values are specified by D codes (cutter compensation) or H codes (tool length compensation).

The compensation values corresponding to the D codes or H codes are displayed and set on the tool offset screen.

Displaying the tool offset screen

To display the tool offset screen, follow the procedure described in Section III.x.x, "Offset Setting." The screen provides three types of layout for tool offset memory A, B, and C.

£ Tool offset memory A

No distinction is made between D codes and H codes, as well as between geometry and wear.

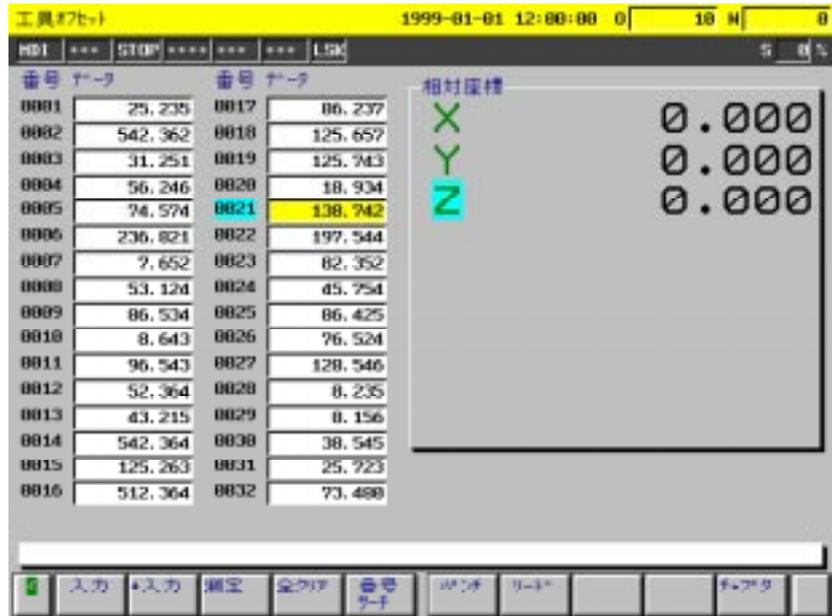


Fig. 9.1 (a) Tool Offset Memory A

£ Tool offset memory B

No distinction is made between geometry and wear, but distinction is made between D codes and H codes.

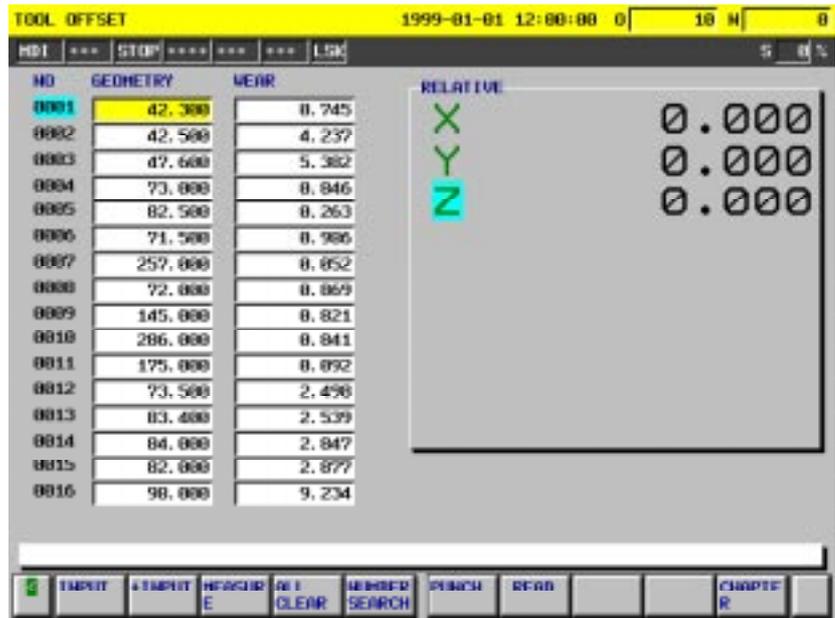


Fig. 9.1 (b) Tool Offset Memory B

£ Tool offset memory C

A distinction is made between D codes (radius) and H codes (length), and between geometry and wear.

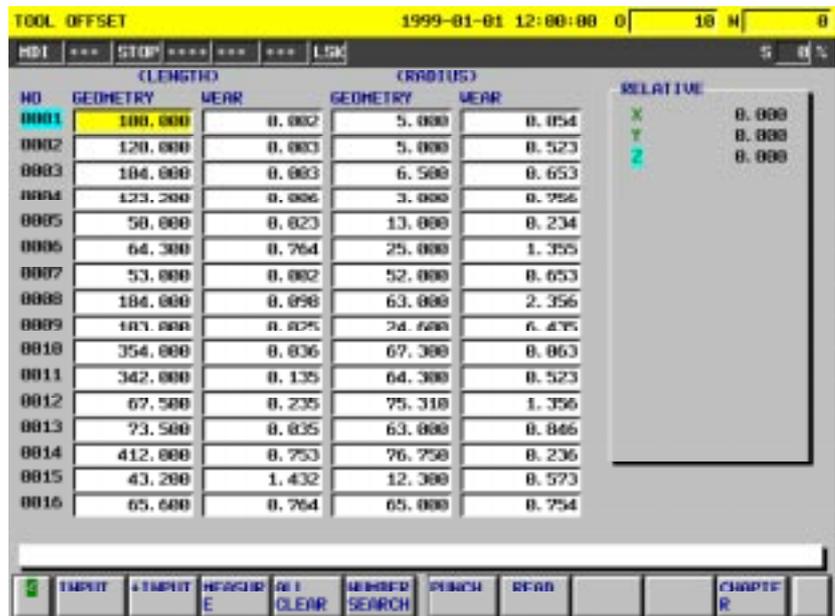


Fig.9.1 (c) Tool Offset Memory C

Setting/updating tool offsets

▣ Moving the cursor

Method 1

Using the page keys   and the cursor keys    , position the cursor to the offset to be set/updated.

Method 2

- (1) Press the [NUMBER SEARCH] soft key.
- (2) Key in the number of the offset to be set/updated.
- (3) Press the [EXEC] soft key.

Method 3

- (1) Key in the number of the offset to be set/updated.
- (2) Press the [NUMBER SEARCH] soft key.

£ Entering offsets (absolute input)

Method 1

- (1) Press the [INPUT] soft key.
- (2) Key in an offset.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in an offset.
- (2) Press the [INPUT] soft key.

Method 3

- (1) Key in an offset.
- (2) Press the  key.

£ Entering offsets values (incremental input)

Method 1

- (1) Press the [+ INPUT] soft key.
- (2) Key in an incremental or decremental value relative to the current offset.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in an incremental or decremental value with respect to the current offset.
- (2) Press the [+ INPUT] soft key.

£ Offset increment system

As an offset, a number with or without a decimal point can be entered. The increment system of an offset depends on the setting of the parameters, as indicated below.

Parameter				Offset increment system (valid data range)	
OFE	OFD	OFC	OFA	Metric input	Inch input
0	0	0	1	0.01 ()9999.99)	0.001 ()999.999)
0	0	0	0	0.001 ()9999.999)	0.0001 ()999.9999)
0	0	1	0	0.0001 ()9999.9999)	0.00001 ()999.99999)
0	1	0	0	0.00001 ()9999.99999)	0.000001 ()999.999999)
1	0	0	0	0.000001 ()999.999999)	0.0000001 ()99.9999999)

OFA : Bit 0 of parameter No. 6002

OFC : Bit 1 of parameter No. 6002

OFD : Bit 0 of parameter No. 6004

OFE : Bit 0 of parameter No. 6007

Clearing tool offsets

£ Tool offset memory A

- (1) Press the [ALL CLEAR] soft key.
- (2) Press the [ALL].[GEOMETRY].[WEAR] soft key.
No distinction is made between geometry and wear, so that pressing any of these keys clears all the offset values to 0.

£ Tool offset memory B

- (1) Press the [ALL CLEAR] soft key.
- (2) When the [ALL] soft key is pressed, all offsets are set to 0.
- (3) When the [GEOMETRY] soft key is pressed, the geometric offsets are set to 0.

- (4) When the [WEAR] soft key is pressed, the wear offsets are set to 0.

£ Tool offset memory C

- (1) Press the [ALL CLEAR] soft key.
- (2) When the [ALL] soft key is pressed, all offsets are set to 0.
- (3) When the [GEOMETRY] soft key is pressed, the geometric offsets for both length and radius are set to 0.
- (4) When the [WEAR] soft key is pressed, the wear offsets for both length and radius are set to 0.

Tool offset input/output

£ Tool offset input

Method 1 (Neither the input file name nor number need be specified)

- (1) Select MDI mode.
- (2) Press the [READ] soft key.
- (3) Press the [TOOL OFFSET] soft key.

Method 2 (The input file is specified with its file name)

- (1) Select MDI mode.
- (2) Press the [READ] soft key.
- (3) Press the ["FILE NAME] soft key.
- (4) Enter a file name.
- (5) Press the [FILE NAME"] soft key.
- (6) Press the [TOOL OFFSET] soft key.

Method 3 (The input file is specified with its file number (one of two methods))

- (1) Select MDI mode.
- (2) Press the [READ] soft key.
- (3) Press the [(FILE#)] soft key.
- (4) Enter a file number.
- (5) Press the [TOOL OFFSET] soft key.

Method 4 (The input file is specified with its file number (one of two methods))

- (1) Select MDI mode.
- (2) Press the [READ] soft key.
- (3) Press the address key.
- (4) Enter a file number.
- (5) Press the [TOOL OFFSET] soft key.

Method 5 (The input file is specified with its file number (one of three methods))

- (1) Select MDI mode.
- (2) Press the  address key.
- (3) Enter a file number.
- (4) Press the [READ] soft key.

£ Tool offset output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [TOOL OFFSET] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [TOOL OFFSET] soft key.

Method 3 (The output file is specified with its file number (one of two methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [TOOL OFFSET] soft key.

Method 4 (The output file is specified with its file number (one of two methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the  address key.
- (3) Enter a file number.
- (4) Press the [TOOL OFFSET] soft key.

Method 5 (The output file is specified with its file number (one of three methods))

- (1) Press the  address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

1. When method 1 of tool offset output is used (when neither an output file name nor a file number is specified) for output to the FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, or memory card, file name TOOL-OFS.TXT is selected.
2. For input to or output from a FANUC Floppy Cassette, FANUC FA Card, or FANUC Handy File, an input/output file can be specified using a file name or file number.
3. For input to or output from a memory card, an input/output file can be specified using a file name.

£ Input/output using other screens

Tool offset input/output is possible on the floppy directory screen and memory card screen in addition to the tool offset screen. For details, see Section III.x.x, "Floppy Directory Screen" and Section III.x.x, "Memory Card Screen."

Measurement

The [MEASURE] soft key is used for tool length measurement. For details of this function, see Section III.x, "Tool Length Measurement."

9.2 DISPLAYING AND SETTING WORKPIECE ORIGIN OFFSETS

The workpiece origin offset for each workpiece coordinate system (G54 to G59 and G54.1 P1 to G54.1 P48) and an external workpiece origin offset are displayed and set.

Displaying the workpiece origin offset screen

To display the workpiece origin offset screen, follow the procedure described in Section III.x.x, "Offset Setting."

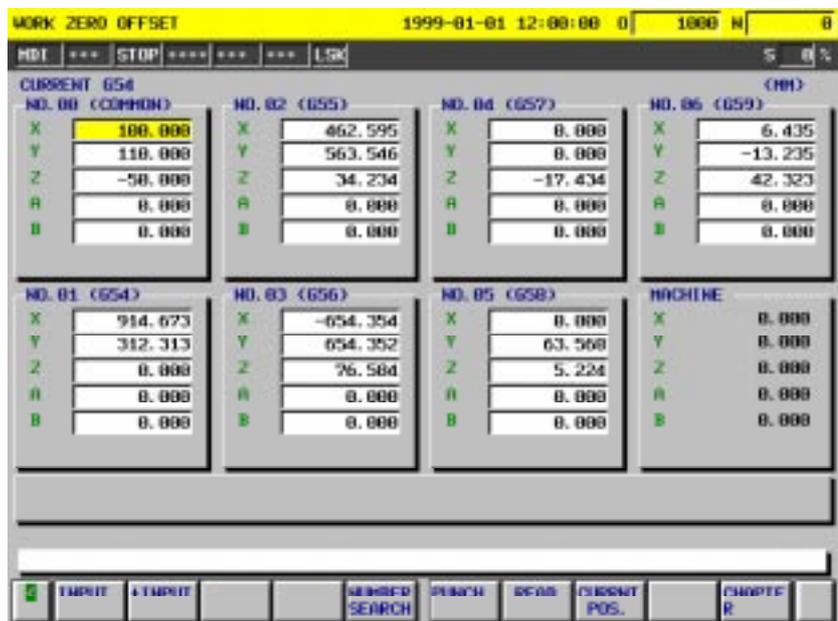


Fig. 9.2 (a) Workpiece Origin Offset Screen

£ Selecting display axes

By setting bit 4 (ZND) of parameter No. 1009, axes for which offsets are displayed and set on the workpiece origin offset screen can be selected.

Setting/updating workpiece origin offsets

£ Positioning the cursor

- Using the cursor keys  , the cursor can be moved up and down.

- 2 Using the cursor keys  , the cursor can be moved from one workpiece coordinate system to another without changing the axis for which the cursor is displayed.
- 3 The page keys   enables paging. Each time paging is performed, the cursor moves to the top of a page.

£ Searching for a workpiece coordinate system

A workpiece coordinate system can be found using a number indicated in the table below.

No.	Found workpiece coordinate system
0	External workpiece origin offset No. 00 (common)
1 ~6	Workpiece coordinate system No. 01 (G54) to No. 06 (G59)
P1 ~48	Additional workpiece coordinate system P01 to P48 (G54.1)

Method 1

- (1) Press the [NUMBER SEARCH] soft key.
- (2) Key in the number of a workpiece coordinate system to be found.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in the number of a workpiece coordinate system to be found.
- (2) Press the [NUMBER SEARCH] soft key.

£ Entering offsets (absolute input)

Method 1

- (1) Press the [INPUT] soft key.
- (2) Key in an offset.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in an offset.
- (2) Press the [INPUT] soft key.

Method 3

- (1) Key in an offset.

- (2) Press the  key.

£ Entering offsets (incremental input)

Method 1

- (1) Press the [+INPUT] soft key.
- (2) Key in an incremental or decremental value relative to the current offset.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in an incremental or decremental value relative to the current offset.
- (2) Press the [+INPUT] soft key.

Workpiece origin offset input/output

£ Workpiece origin offset input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [WORK OFFSET] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [WORK OFFSET] soft key.

Method 3 (The input file is specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE #)] soft key.
- (3) Enter a file number.
- (4) Press the [WORK OFFSET] soft key.

Method 4 (The input file is specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press the  address key.
- (3) Enter a file number.
- (4) Press the [WORK OFFSET] soft key.

Method 5 (The input file is specified with its file number (three of three methods))

- (1) Press the address key.
- (2) Enter a file number.
- (3) Press the [READ] soft key.

£ Workpiece origin offset output

Method 1.when neither an output file name nor a file number is specified.

- (1) Press the [PUNCH] soft key.
- (2) Press the [WORK OFFSET] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [WORK OFFSET] soft key.

Method 3 (The input file is specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE #)] soft key.
- (3) Enter a file number.
- (4) Press the [WORK OFFSET] soft key.

Method 4 (The input file is specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the address key.
- (3) Enter a file number.
- (4) Press the [WORK OFFSET] soft key.

Method 5 (The input file is specified with its file number (three of three methods))

- (1) Press the address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

- | | |
|---|--|
| 1 | When method 1 of workpiece origin offset output is used (when neither an output file name nor a file number is specified) for output to the FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, or memory card, file name EXT-WKZ.TXT is selected. |
| 2 | For input to or output from the FANUC Floppy Cassette, FANUC FA Card, or FANUC Handy File, an input/output file can be specified using a file name or file number. |
| 3 | For input to or output from a memory card, an input/output file can be specified using a file name. |

£ Input/output using other screens

Workpiece origin offset input/output is possible on the floppy directory screen and memory card screen in addition to the workpiece origin offset screen. For details, see Section III.x.x, "Floppy Directory Screen" and Section III.x.x, "Memory Card Screen."

Manual workpiece origin setting

The [CURRENT POS.] soft key is used for manual workpiece origin setting. For details of this function, see Section III.x.x, "Manual Workpiece Origin Setting."

9.3 DISPLAY AND SETTING OF SETTING PARAMETERS

Parameters that can be set by input (setting parameters) can be displayed and set on the setting parameter screen.

Displaying setting parameters

To display the setting parameter screen, follow the procedure described in Section III.x.x, "Offset Setting."

£ Setting parameter screen

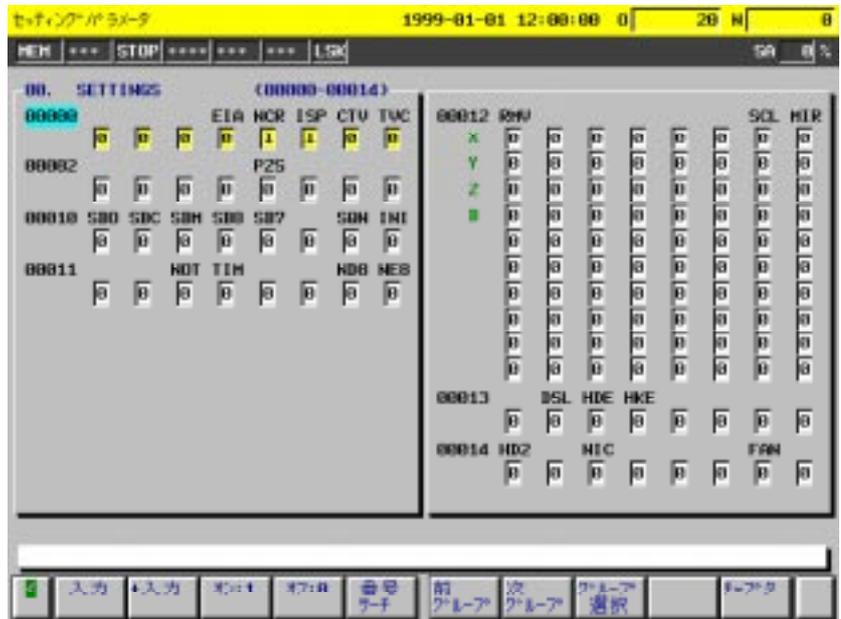


Fig. 9.3 (a) Setting Parameter Screen

£ Display data

Setting parameter numbers, names, and settings are displayed. For parameters of axis type, axis names are also displayed. Setting parameters are divided into several groups according to their applications, and a group name is displayed in the upper left corner of the screen. (The names of some setting parameters are not displayed.)

£ Types of setting parameters

According to the types of settings, setting parameters are divided into three types: bit type, integer type, and real number type.

1. Bit type parameters

This type of parameter consists of 8 bits, each of which is set to 0 or 1. A bit name is displayed above each bit position. (No name is displayed above some bit positions.)

2. Integer type parameters

In this type of parameter, an integer is set.

3. Parameters of real number type

In this type of parameter, a real number is set.

A setting with a decimal point is displayed, and the unit of a setting is also displayed.

In addition, parameters of axis type are supported to enable the setting of a value for each axis. According to the types of settings, parameters of axis type are divided into three types: bit axis type, integer axis type, and real number axis type.

An axis name is displayed to the left of the setting for each axis.

However, no axis name is displayed for axes beyond the maximum number of controlled axes.

4. Parameters of bit axis type

5. Parameters of integer axis type

6. Parameters of real number axis type

Setting of setting parameters

£ Mode setting

Select MDI mode, or enter the emergency stop state by pressing the emergency stop button.

£ Data protection key

Turn on the data protection key (KEY2).

£ Positioning the cursor

Position the cursor to the setting parameter you want to set.

Method 1

Using the page keys and   the cursor keys



, position the cursor.

Method 2

- (1) Press the [NUMBER SEARCH] soft key.
- (2) Key in a setting parameter number.
- (3) Press the [EXEC] soft key.

Method 3

- (1) Key in a setting parameter number.
- (2) Press the [NUMBER SEARCH] soft key.

£ Entering settings (absolute input)

Method 1

- (1) Press the [INPUT] soft key.
- (2) Enter a number you want to set.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter a number you want to set.
- (2) Press the [INPUT] soft key.

Method 3

- (1) Enter a number you want to set.

- (2) Press the  key.

For details of the input method, see Section X.X.X, "Data input method".

£ Entering settings (incremental input)

Method 1

- (1) Press the [+INPUT] soft key.
- (2) Key in an incremental or decremental value relative to the current setting.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in an incremental or decremental value relative to the current setting.
- (2) Press the [+INPUT] soft key.

For details of the input method, see Section X.X.X, "Data input method".

NOTE

For parameters of bit type and bit axis type, incremental input is impossible. Pressing the [+INPUT] soft key sets absolute input mode.

£ Bit on : 1

When the [ON:1] soft key is pressed, the bit of a parameter of bit type/bit axis type to which the cursor is positioned is set to 1. If the cursor is displayed at all the bit positions, all the bits are set to 1.

£ Bit off : 0

When the [OFF:0] soft key is pressed, the bit of a parameter of bit type/bit axis type to which the cursor is positioned is set to 0. If the cursor is displayed at all the bit positions, all the bits are set to 0.

Selecting a group

On the group selection screen, the setting parameters of a desired group can be easily displayed.

£ Group selection screen

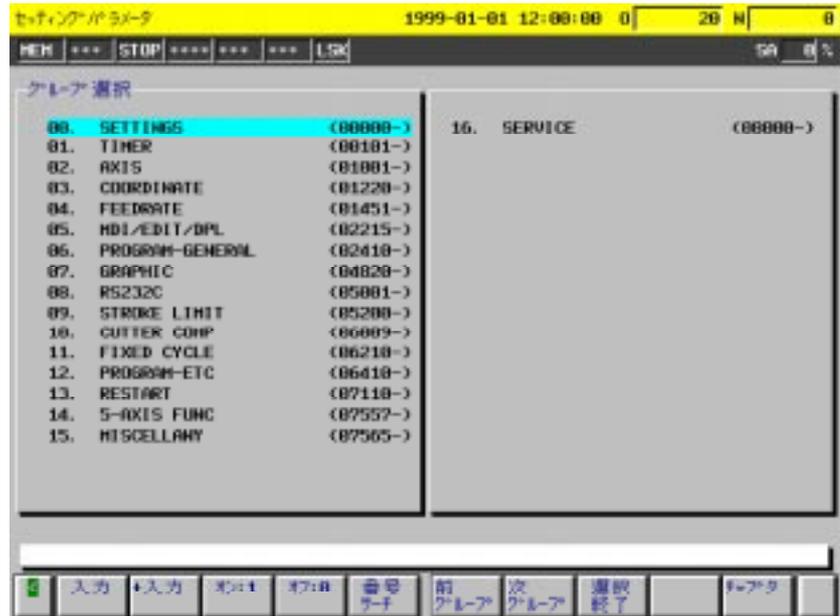


Fig. 9.3 (b) Group Selection Screen

£ Displaying the group selection screen

On the setting parameter screen, press the [SELECTGROUP] soft key.

Positioning the cursor

Position the cursor to the group you want to display.

Method 1

Using the cursor keys ,     position the cursor.

Method 2

- (1) Press the [NUMBER SEARCH] soft key.
- (2) Key in the number of the group that you want to display.
- (3) Press the [EXEC] soft key.

Method 3

- (1) Key in the number of the group that you want to display.
- (2) Press the [NUMBER SEARCH] soft key.

£ Closing the group selection screen

Pressing the soft key terminates the group selection screen to return to the parameter screen.

9.4 DISPLAYING AND SETTING CUSTOM MACRO VARIABLE VALUES

The values of the local variables (#1 to #33), common variables (#100 to #199, #200 to #499, #500 to #999), and system variables (#1000 and up) are displayed.

Values can be set in the common variables.

Displaying custom macro variable values

To display the custom macro screen, follow the procedure described in Section III.x.x, "Offset Setting."

The custom macro screen displays the numbers and values of custom macro variables.

For a custom macro variable that has a variable name assigned, the variable name is also displayed.

If the value of a custom macro variable is null, Data Empty is displayed as the variable value.

If a variable value is greater than the displayable range, +OVERFLOW is displayed.

If a value is less than the displayable range, -OVERFLOW is displayed.

£ Custom macro screen

The numbers and values of the custom macro variables are displayed.

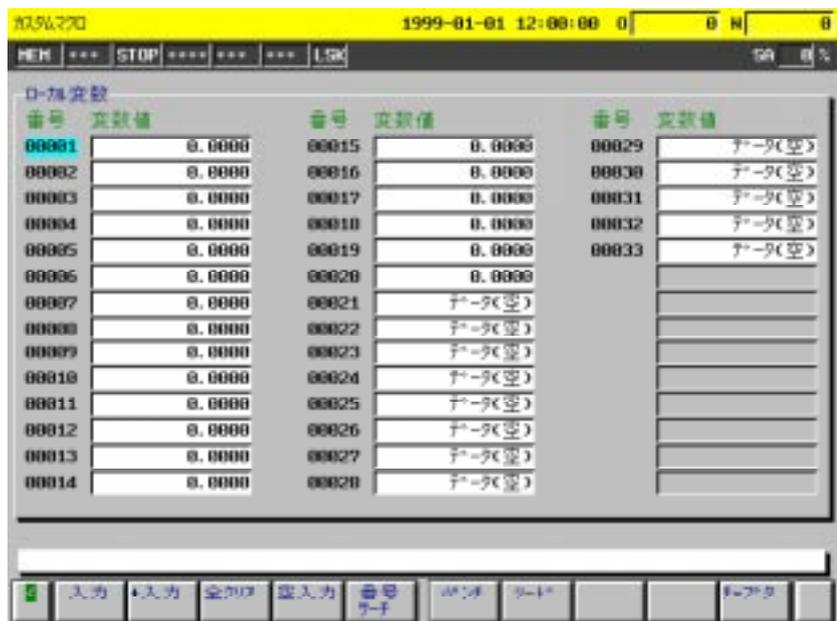


Fig. 9.4 (c) Custom Macro Screen without Variable

£ Custom macro screen
 (#500 to #549, and system variables)

The numbers, names, and values of the custom macro variables are displayed.

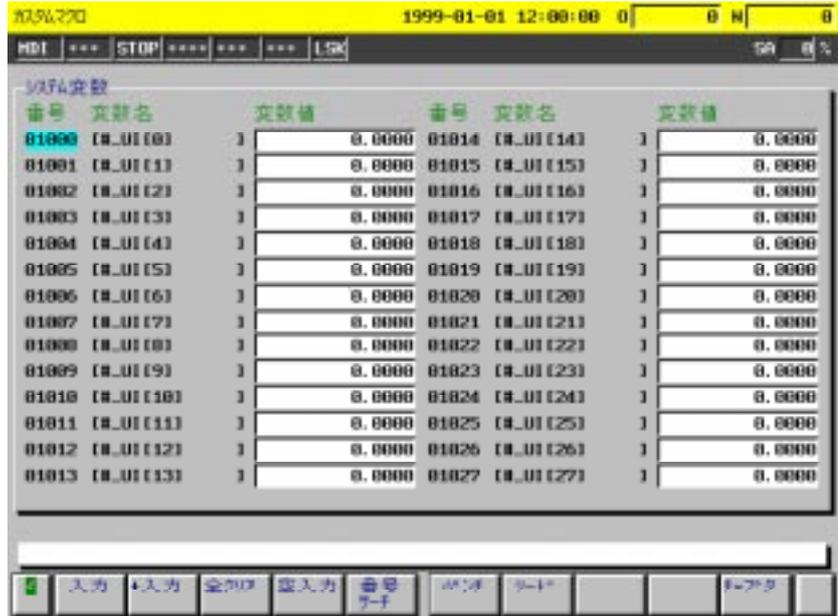


Fig. 9.4 (d) Custom Macro Screen with Variable Names

£ Undisplayable variable values

For undisplayable variable values produced by arithmetic operations, the following indications are used:

Variable value range	Indication for variable value
0 < Variable value < +0.00000000001	+ UNDER FLOW
0 > Variable value > -0.00000000001	- UNDER FLOW
Variable value > 9999999999999	+ OVER FLOW
Variable value < -9999999999999	- OVER FLOW

Setting custom macro variable values

A value can be set in a common variable. No value can be set in a custom macro variable other than the common variables.

The common variables are classified as common variable 1 and common variable 2, as follows:

Common variable 1 (#100 to #199): Not held when the power is turned off

Common variable 2 (#200 to #999): Held when the power is turned off

£ Mode setting

Select MDI mode, or enter emergency stop state by pressing the emergency stop button.

£ Positioning the cursor

Position the cursor to the custom macro variable you want to set.

Method 1

Using the page keys  and  the cursor keys , , , , position the cursor.

Method 2

- (1) Press the [NUMBER SEARCH] soft key.
- (2) Key in the number of a custom macro variable.
- (3) Press the [EXEC] soft key.

Method 3

- (1) Key in the number of a custom macro variable.
- (2) Press the [NUMBER SEARCH] soft key.

£ Entering settings (absolute input)

Method 1

- (1) Press the [INPUT] soft key.
- (2) Enter a number you want to set.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter a number you want to set.
- (2) Press the [INPUT] soft key.

Method 3

- (1) Enter a number you want to set.
- (2) Press the  key.

For details of the input method, see Section X.X.X, "Data input method".

£ Entering settings (incremental input)

Method 1

- (1) Press the [+INPUT] soft key.
- (2) Key in an incremental or decremental value relative to the current setting.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in an incremental or decremental value relative to the current setting.
- (2) Press the [+INPUT] soft key.

For details of the input method, see Section X.X.X, "Data input method".

£ Input of "null"

Press the [INPUT OF NULL] soft key.

Data Empty is displayed as the variable value.

£ Clearing common variable 1/2 to 0

Common variable 1 and common variable 2 can be cleared to 0 at the same time.

When clearing both common variable 1 and common variable 2 to 0

- (1) Press the [ALL CLEAR] soft key. The soft key display changes to the operation guide key display.
- (2) Press the [ALL] soft key.

When clearing only common variable 1 to 0

- (1) Press the [ALL CLEAR] soft key. The soft key display changes to the operation guide key display.
- (2) Press the [COMMON VARIABLE 1] soft key.

When clearing only common variable 2 to 0

- (1) Press the [ALL CLEAR] soft key. The soft key display changes to the operation guide key display.
- (2) Press the [COMMON VARIABLE 2] soft key.

NOTE

By setting parameter No. 7036 to No. 7039, the common variables in a specified variable number range can be made unwritable.

9.5 INPUT/OUTPUT OPERATION OF CUSTOM MACRO COMMON VARIABLES

The value of a custom macro common variable (#200 to #999) can be output to an external input/output device.

A value can be set in a custom macro common variable (#200 to #999) from an external input/output device.

This section describes the method of custom macro common variable input from and output to an external input/output device.

Output operation of custom macro common variables

Custom macro common variables stored in CNC memory can be output to an external input/output device.

Output procedure

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [COMMON VARIABLE 2] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [COMMON VARIABLE 2] soft key.

Method 3 (The input file is specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE #)] soft key.
- (3) Enter a file number.
- (4) Press the [COMMON VARIABLE 2] soft key.

Method 4 (The input file is specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the  address key.
- (3) Enter a file number.

- (4) Press the [COMMON VARIABLE 2] soft key.

Method 5 (The input file is specified with its file number (three of three methods))

- (1) Press the  address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

Output format

The value of a custom macro common variable is output as double-precision floating-point data in a hexadecimal bit image.

When the value is 0
When the value is <null>
When the value is an ordinary number
Variable name

NOTE

The value of a custom macro common variable cannot be output in conventional custom macro statement program format.

Input operation of custom macro common variables

Custom macro common variables can be entered from an external input/output device.

Mode setting

Set MDI mode.

Input procedure

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [COMMON VARIABLE 2] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [COMMON VARIABLE 2] soft key.

Method 3 (The input file is specified with its file number (one of two methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [COMMON VARIABLE 2] soft key.

Method 4 (The input file is specified with its file number (one of two methods))

- (1) Press the [READ] soft key.
- (2) Press the  address key.
- (3) Enter a file number.
- (4) Press the [COMMON VARIABLE 2] soft key.

Method 5 (The input file is specified with its file number (one of two methods))

- (1) Press the  address key.
- (2) Enter a file number.
- (3) Press the [READ] soft key.

The entered value of a custom macro common variable is automatically stored into CNC memory.

Procedure for inputting from a file of custom macro statement program format

Custom macro common variables can be entered from a file of conventional custom macro statement program format.

- (1) Enter the file as described in Section III.x.x.x, "Program input operation."

- (2) Select MEM mode.
- (3) Press the cycle start button to execute the read program.
This stores the custom macro common variable values into CNC memory.

9.6 DISPLAYING AND SETTING PARAMETERS

When the CNC is connected to a machine, the parameters are set to determine the machine specifications and functions and to make full use of the capability of the servo motor and so forth. The parameter setting varies from one machine to another. So, refer to the parameter table prepared by the machine tool builder.

Usually, the parameters need not be changed by the end user.

Displaying parameters

Follow the procedure described in Section III.x.x, "System" to display the parameter screen.

£ Parameter display screen

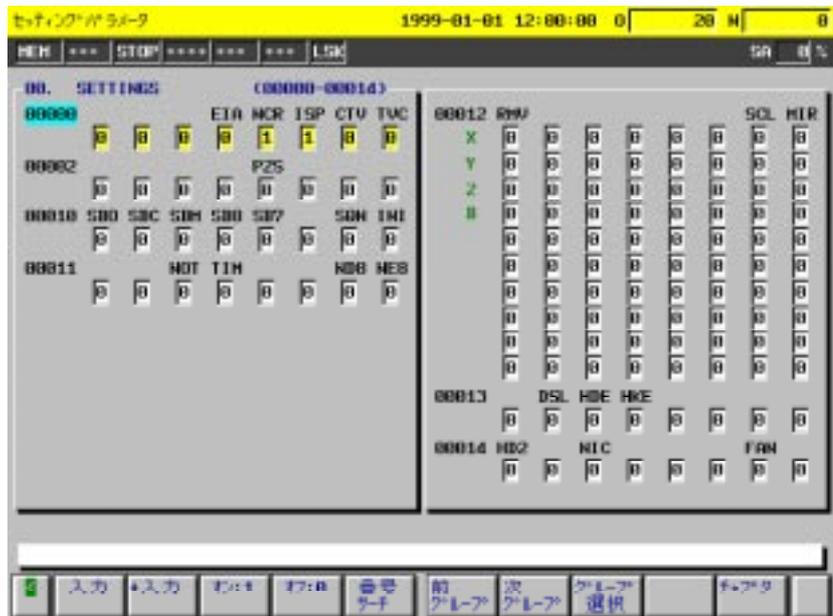


Fig. 9.6 (e) Parameter Screen

£ Display data

Parameter numbers, names, and settings are displayed. For the parameters of axis type, axis names are displayed. For the parameters of spindle type, spindle names are displayed. Parameters are divided into several groups according to their applications, and a group name is displayed in the top left corner of the screen.

£ Types of parameters

According to the types of settings, parameters are divided into three types: bit type, integer type, and real number type.

1. Bit type parameters

This type of parameter consists of 8 bits, each of which is set to 0 or 1. A bit name is displayed above each bit position. (No name is displayed above some bit positions.)

2. Integer type parameters

In this type of parameter, an integer is set.

3. Real number type parameters

In this type of parameter, a real number is set.

A setting with a decimal point is displayed, and the unit of setting is also displayed.

Parameters of axis type are available which set a value for each axis. According to the types of settings, parameters of axis type are divided into three types: bit axis type, integer axis type, and real number axis type.

An axis name is displayed to the left of the setting of each axis.

However, no axis name is displayed for axes beyond the maximum number of controlled axes.

4. Bit axis type parameters**5. Integer axis type parameters****6. Real number axis type parameters**

Parameters of spindle type are available which set a value for each spindle. According to the types of settings, parameters of spindle type are divided into three types: bit spindle type, integer spindle type, and real number spindle type.

A spindle name is displayed to the left of the setting for each spindle.

7. Bit spindle type parameters**8. Integer spindle type parameters****9. Real number spindle type parameters**

Setting parameters

⌘ Mode setting

Select MDI mode, or enter the emergency stop state by pressing the emergency stop button.

⌘ Setting parameter PWE

On the setting parameter screen, set bit 0 (PWE) of parameter No. 8000 for allowing setting input to 1.

For the method of setting, see Section 11.X.X, "Displaying and setting of setting parameters."

⌘ Positioning the cursor

Position the cursor to the parameter you want to set.

Method 1

Using the page keys and   the cursor keys



, position the cursor.

Method 2

- (1) Press the [NUMBER SEARCH] soft key.
- (2) Key in a parameter number.
- (3) Press the [EXEC] soft key.

Method 3

- (1) Key in a parameter number.
- (2) Press the [NUMBER SEARCH] soft key.

£ Entering settings (absolute input)

Method 1

- (1) Press the [INPUT] soft key.
- (2) Enter a number you want to set.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter a number you want to set.
- (2) Press the [INPUT] soft key.

Method 3

- (1) Enter a number you want to set.

- (2) Press the  key.

For details of the input method, see Section X.X.X, "Data input method".

£ Entering settings (incremental input)

Method 1

- (1) Press the [+INPUT] soft key.
- (2) Key in an incremental or decremental value relative to the current setting.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in an incremental or decremental value relative to the current setting.
- (2) Press the [+INPUT] soft key.

For details of the input method, see Section X.X.X, "Data input method".

NOTE

For parameters of bit type, bit axis type, and bit spindle type, incremental input is impossible.

Pressing the [+INPUT] soft key sets absolute input mode.

£ Bit on : 1

When you press the [ON:1] soft key, the bit of a parameter of bit type/bit axis type/bit spindle type to which the cursor is positioned is set to 1.

If the cursor is displayed for all bit positions, all bits are set to 1.

£ Bit off : 0

When you press the [OFF:0] soft key, the bit of a parameter of bit type/bit axis type/bit spindle type to which the cursor is positioned is set to 0.

If the cursor is displayed for all bit positions, all bits are set to 0.

£ Termination of setting

Upon completion of parameter setting, set bit 0 (PWE) of parameter No. 8000 for allowing setting input to 0.

Selecting a group

On the group selection screen, the parameters of a desired group can easily be displayed.

£ Group selection screen

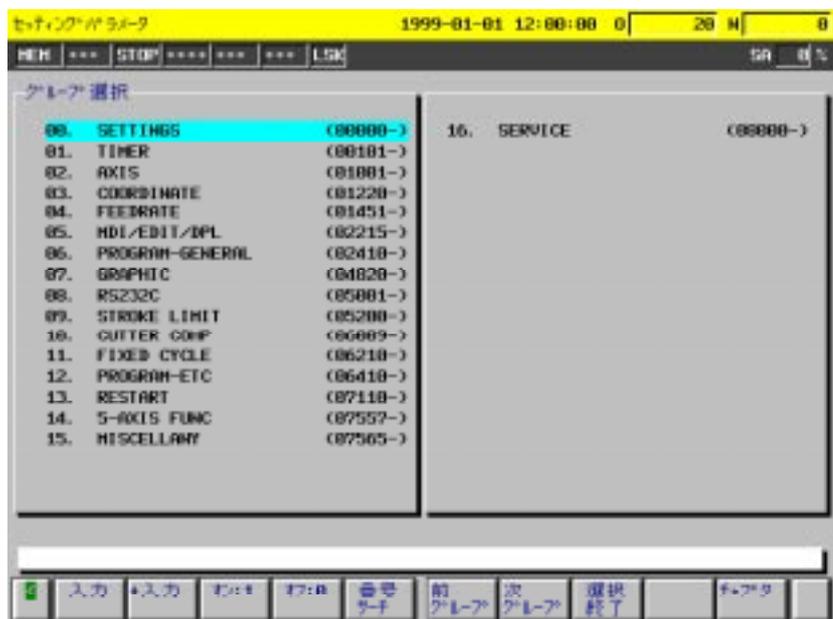


Fig. 9.6 (f) Group Selection Screen

£ Displaying the group selection screen

On the parameter screen, press the [SELECTGROUP] soft key.

£ Positioning the cursor

Position the cursor to the group you want to display.

Method 1

Using the cursor keys ,     position the cursor.

Method 2

- (1) Press the [NUMBER SEARCH] soft key.
- (2) Key in the number of the group you want to display.
- (3) Press the [EXEC] soft key.

Method 3

- (1) Key in the number of the group you want to display.
- (2) Press the [NUMBER SEARCH] soft key.

£ Terminating the group selection

Pressing the soft key terminates the group selection screen to return to the parameter screen.

9.7 DISPLAYING AND SETTING PITCH ERROR COMPENSATION DATA

Pitch error compensation data is set to match the characteristics of a machine connected to the CNC.

Pitch error compensation data varies from one machine to another, and any modification made to the pitch error compensation data degrades the machine precision.

In general, the end user should not modify the pitch error compensation data.

Displaying and setting pitch error compensation data

£ Pitch error compensation screen

The screenshot shows a CNC control interface with a yellow title bar 'ピッチ誤差補正' and a status bar 'MDI STOP LSR'. The main display area is titled 'ピッチ誤差補正データ' and contains a table of data points. The table has 6 columns, each with a '番号' (Number) and a 'データ' (Data) field. The data values are all 0. The bottom of the screen shows a control panel with buttons for '入力', '+入力', 'データ', 'リセット', and 'クリア'.

番号	データ										
0000	0	0014	0	0028	0	0042	0	0056	0	0070	0
0001	0	0015	0	0029	0	0043	0	0057	0	0071	0
0002	0	0016	0	0030	0	0044	0	0058	0	0072	0
0003	0	0017	0	0031	0	0045	0	0059	0	0073	0
0004	0	0018	0	0032	0	0046	0	0060	0	0074	0
0005	0	0019	0	0033	0	0047	0	0061	0	0075	0
0006	0	0020	0	0034	0	0048	0	0062	0	0076	0
0007	0	0021	0	0035	0	0049	0	0063	0	0077	0
0008	0	0022	0	0036	0	0050	0	0064	0	0078	0
0009	0	0023	0	0037	0	0051	0	0065	0	0079	0
0010	0	0024	0	0038	0	0052	0	0066	0	0080	0
0011	0	0025	0	0039	0	0053	0	0067	0	0081	0
0012	0	0026	0	0040	0	0054	0	0068	0	0082	0
0013	0	0027	0	0041	0	0055	0	0069	0	0083	0

Fig. 9.7 (g) Pitch Error Compensation Screen

£ Display data

The numbers and values of the pitch error compensation data are displayed.

Pitch error compensation data ranging from No. 0 to No. 1279 (1280 points) is displayed.

For bidirectional pitch error compensation, additional data ranging from No. 10000 to No. 11279 (1280 points) is displayed.

Setting pitch error compensation data

For the method of setting, refer to the connection manual.

9.8 Manual Tool Length Measurement

A numeric value displayed as a relative position can be placed in offset memory as an offset by using the soft keys.

To do this, first choose the tool offset screen. This screen shows a relative position. Then, choose a reference tool, and manually move the reference tool until it touches a machine-specific point. Next, reset the reference position displayed on the screen to zero. Then, manually move the tool to be measured until it touches the same machine-specific point. At this time, the relative position display indicates the difference between the reference tool and the tool to be measured. This difference can be set as an offset.

Displaying the tool offset screen

To display the tool offset screen, follow the procedure described in Section III.x.x, "Offset Setting." The screen has three layouts for tool offset memories A, B, and C. An example of memory A is shown below:

Fig. 9.8 (a) Tool Offset Memory A

Tool length measurement

- Selecting the offset to be changed

Procedure 1

Position the cursor to the offset to be changed or set by pressing a

page key   or a cursor key    .

Procedure 2

- (1) Press the [NO. SEARCH] soft key.
- (2) Enter a number corresponding to the desired offset.
- (3) Press the [EXEC] soft key.

Procedure 3

- (1) Enter a number corresponding to the desired offset.
- (2) Press the [NO. SEARCH] soft key.

- Selecting a measurement axis

Procedure 1

- (1) Press the [MEASURE] soft key.
- (2) Enter the name of a desired axis.
- (3) Press the [SELECT] soft key.

Procedure 2

- (1) Enter the name of a desired axis.
- (2) Press the [MEASURE] soft key.

Procedure 3

- (1) Press the [MEASURE] soft key.
- (2) Press the [NEXT AXIS] or [PREV AXIS] soft key.

The name of the measurement axis, selected from the axes of relative position display, is displayed in reverse video. If the LXY bit (bit 4 of parameter 6000) is set to 0, the Z-axis is selected at power-up. (The tool length compensation axis is always the Z-axis.)

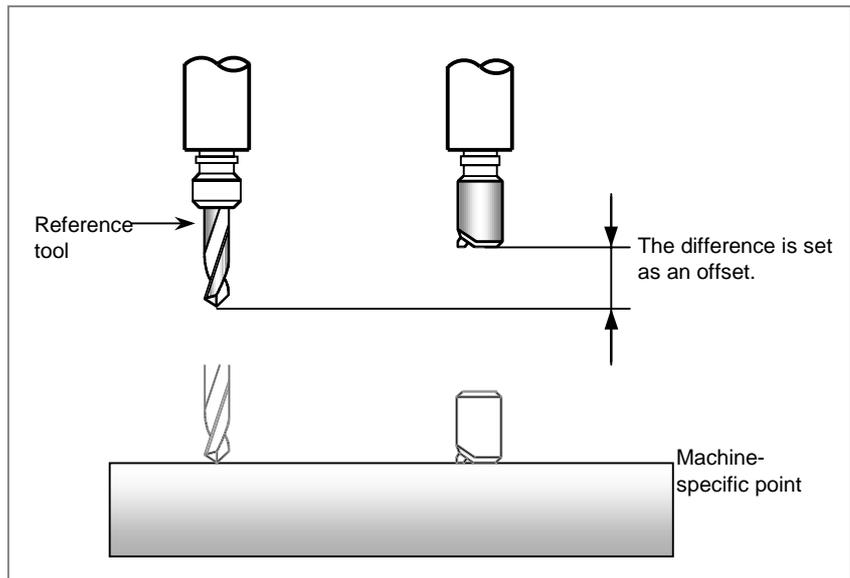
- Measuring the tool length

First, choose and manually move a reference tool until it touches the machine-specific point. Then, press the [MEASURE] soft key -> [ORIGIN] soft key to zero the counter (relative position display).

Next, choose the tool to be measured and manually move the tool until it touches the machine-specific point. The counter (relative position

display) shows the difference between the reference tool and the tool to be measured, which is the offset.

Then, press the [MEASURE] soft key -> [INPUT] soft key. The offset shown in the relative position display is written into the offset memory to which the cursor is positioned.



When the function of tool offset by tool number is provided

If the function of tool offset by tool number is supported, manual tool length measurement can be performed in the same way.

To display the screen for setting tool offset by tool number, follow the procedure described in Section III.x.x, "Offset Setting."

Fig. 9.8 (b) Screen for Setting Tool Offset by Tool Number**- Selecting the offset to be changed**

Position the cursor to a desired offset (length or radius) by

pressing a page key   or a cursor key
   

- Selecting the measurement axis**- Measuring the tool length**

The operation is the same as that performed when the function for applying tool offset by tool number is not provided.

9.9 MANUAL WORKPIECE ORIGIN SETTING

On the workpiece origin offset screen, a workpiece origin offset can be set so that the current machine position is the origin of a selected workpiece coordinate system or a specified position.

Displaying the workpiece origin offset screen

To display the workpiece origin offset screen, follow the procedure described in Section III.x.x, "Offset Setting."

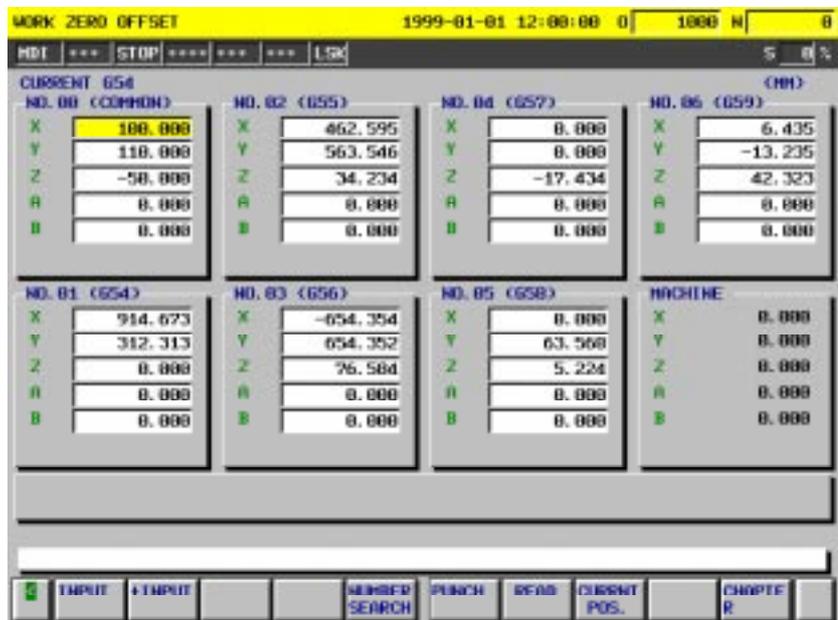


Fig. 9.9 (a) Workpiece Origin Offset Screen

Selecting a workpiece coordinate system and axis

Selecting a workpiece coordinate system

Select the workpiece coordinate system you want to set. A workpiece coordinate system can be found using a number indicated in the table below.

No.	Found workpiece coordinate system
0	External workpiece origin offset No. 00 (common)
1 to 6	Workpiece coordinate system

	No. 01 (G54) to No. 06 (G59)
P1 to 48	Additional workpiece coordinate system P01 to P48 (G54.1)

Method 1

- (1) Press the [NUMBER SEARCH] soft key.
- (2) Key in the number of the workpiece coordinate system you want to find.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in the number of the workpiece coordinate system you want to find.
- (2) Press the [NUMBER SEARCH] soft key.

Selecting an axis

Position the cursor to a desired axis.

When specifying the current position (machine coordinates) as the origin of a selected workpiece coordinate system

- (1) Press the [CURRENT POSITION] soft key.
- (2) Press the [EXEC] soft key.

When specifying the current position (machine coordinates) as a point () in a selected workpiece coordinate system

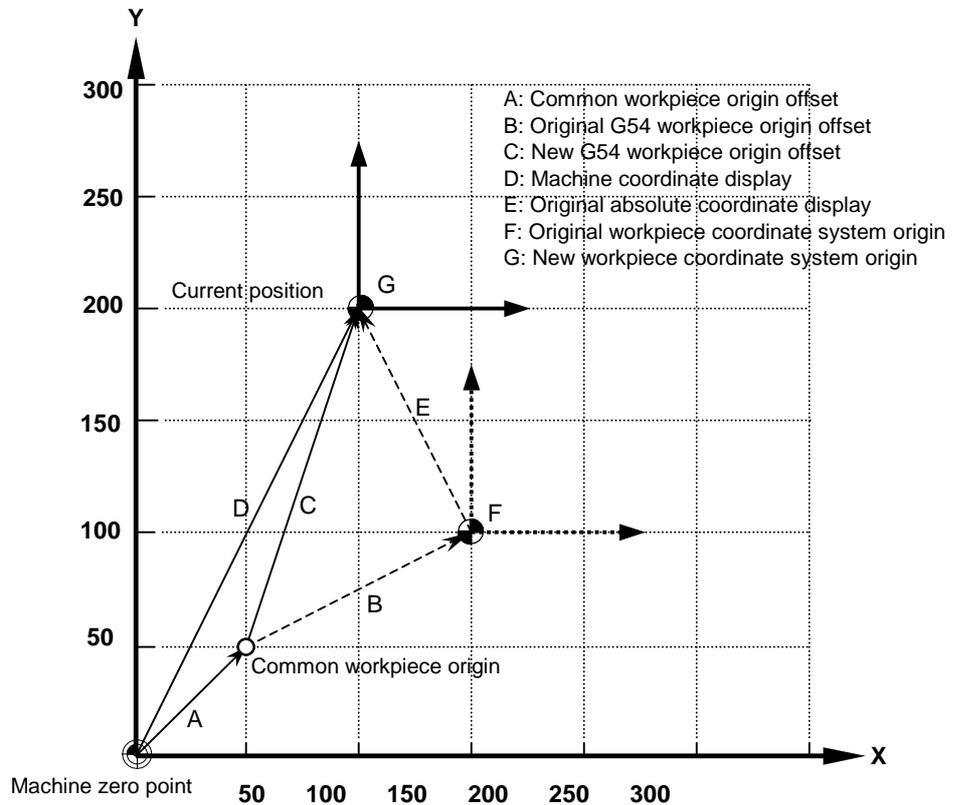
- (1) Press the [CURRENT POSITION] soft key.
- (2) Enter the coordinates of . from the MDI panel.
- (3) Press the [EXEC] soft key.

NOTE

When the cursor is positioned to workpiece origin offset No. 00 (common), an attempt to perform manual workpiece origin setting displays a warning message indicating that the common workpiece origin cannot be entered.

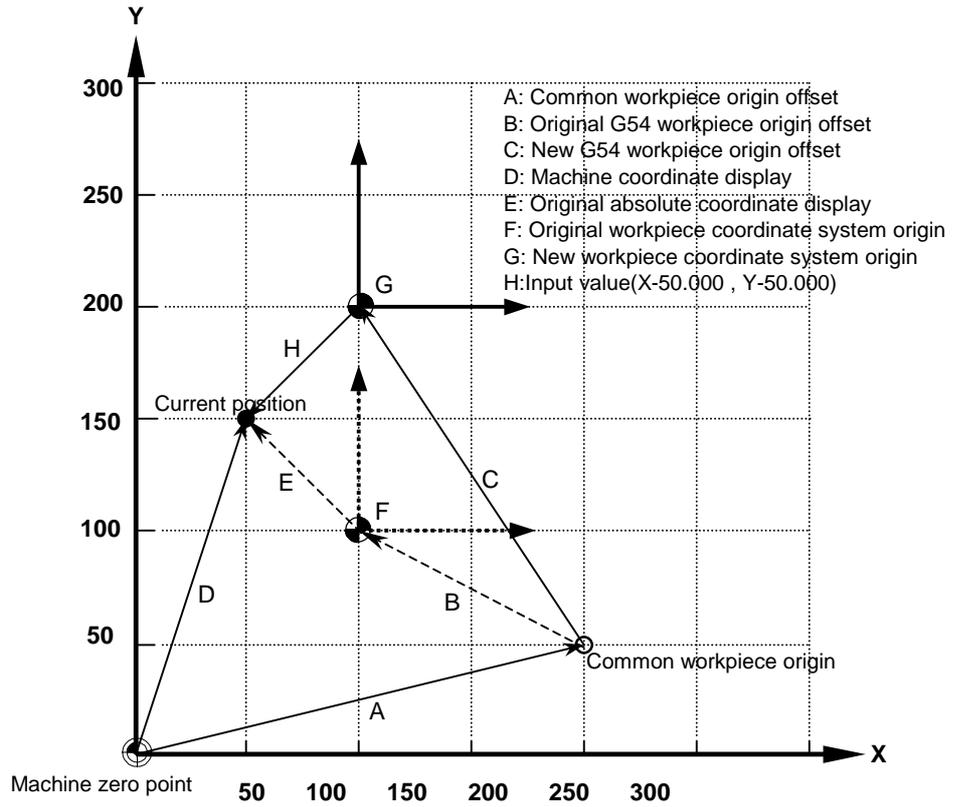
Operation example 1. When a common workpiece origin offset value and workpiece coordinate system No. 1 (G54) are set

@ When an operation is performed so that the current position (machine coordinates: X100.000,Y200.000) is the origin of workpiece coordinate system No. 1



	No. 00 (common)	No.01 (G54)	Machine coordinates	Absolute coordinates
Before setting	X 50.000	X 100.000	X 100.000	X -50.000
	Y 50.000	Y 50.000	Y 200.000	Y 100.000
After setting	No. 00 (common)	No.01 (G54)	Machine coordinates	Absolute coordinates
	X 50.000	X 50.000	X 100.000	X 0.000
	Y 50.000	Y 150.000	Y 200.000	Y 0.000

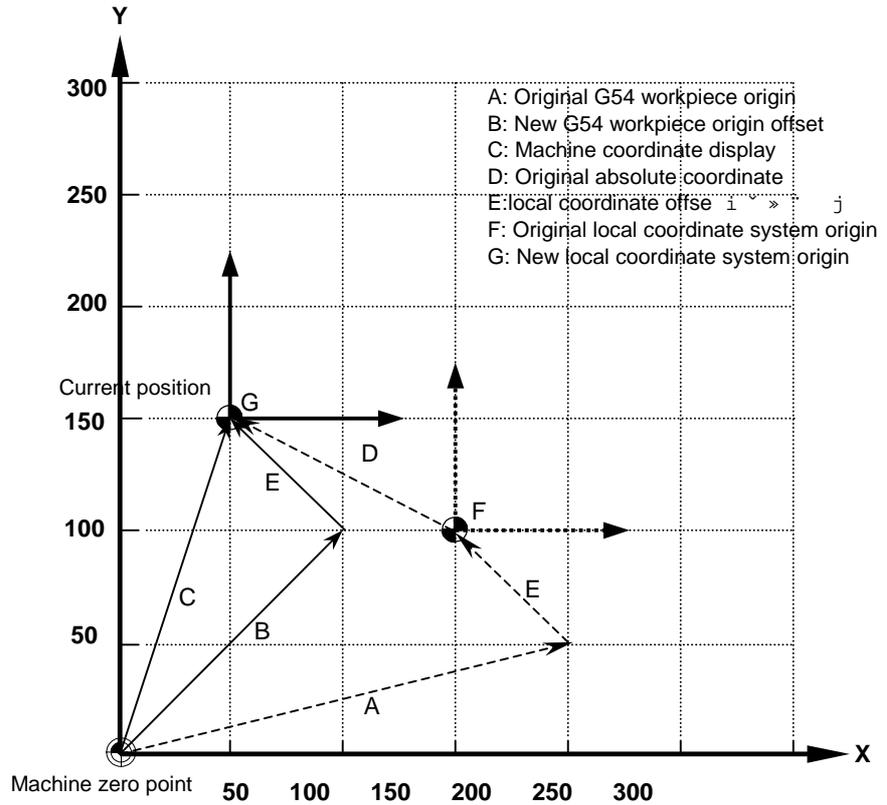
A **When an operation is performed so that the current position (machine coordinates: X100.000,Y200.000) is (X-50.000,Y-50.000) in workpiece coordinate system No. 1**



	No. 00 (common)	No.01 (G54)	Machine coordinates	Absolute coordinates
Before setting	X 200.000	X -100.000	X 50.000	X -50.000
	Y 50.000	Y 50.000	Y 150.000	Y 50.000
After setting	No. 00 (common)	No.01 (G54)	Machine coordinates	Absolute coordinates
	X 200.000	X -100.000	X 50.000	X -50.000
	Y 50.000	Y 150.000	Y 150.000	Y -50.000

Operation example 2. When a local coordinate system (G52) is set

When an operation is performed so that the current position (machine coordinates: X50.000, Y150.000) is the origin of workpiece coordinate system No. 1 (provided that the common workpiece origin offset is 0 for X and Y)

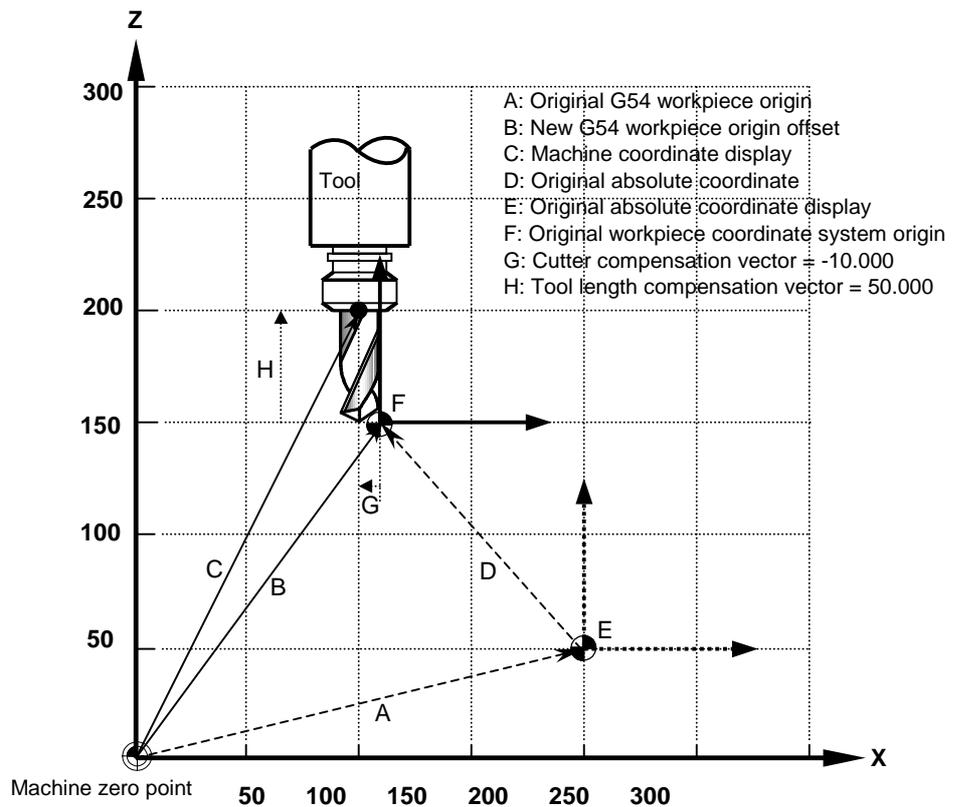


	No. 00 (common)	No.01 (G54)	Machine coordinates	Absolute coordinates
Before setting	X 0.000	X 200.000	X 50.000	X -100.000
	Y 0.000	Y 50.000	Y 150.000	Y 50.000
After setting	X 0.000	X 100.000	X 50.000	X 0.000
	Y 0.000	Y 100.000	Y 150.000	Y 0.000

The origin of the new G54 workpiece origin offset is not located at the current position but shifted from the position by the local coordinate offset set with G52. In other words, the local coordinate system (G52) is not canceled even if the workpiece origin is set manually.

Operation example 3. When tool length compensation and cutter compensation are enabled

When an operation is performed so that the current position (machine coordinates: X100.000, Z200.000) is the origin of workpiece coordinate system No. 1 (provided that the common workpiece origin offset is 0 for X and Y)



	No. 00 (common)	No.01 (G54)	Machine coordinates	Absolute coordinates
Before setting	X 0.000	X 200.000	X 100.000	X -90.000
	Z 0.000	Z 50.000	Z 200.000	Z 100.000
After setting	No. 00 (common)	No.01 (G54)	Machine coordinates	Absolute coordinates
	X 0.000	X 110.000	X 100.000	X 0.000
	Z 0.000	Z 150.000	Z 200.000	Z 0.000

NOTE
 In the table, a setting in which the absolute coordinates exclude the distance moved by tool length compensation/cutter compensation is assumed (with bit 2 (DTL) of parameter No. 2202 set to 1, and bit 3 (DCR) of parameter No. 2202 set to 1).

9.10 Tool Length/Workpiece Origin Measurement

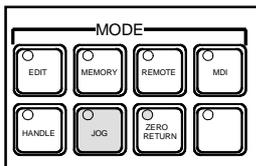
The length of a tool can be measured by executing a program command for automatic tool length measurement (G37) (Section II.14.2, "Automatic Tool Length Measurement" in Programming) or by manually moving the tool to the top surface of the workpiece or the like (Section II-9.8, "Manual Tool Length Measurement"). With tool length/workpiece origin measurement, tool length measurement can be simplified, and the setup for machining can be easily made in a short time. The workpiece origin offset can also be measured easily.

On the tool length offset measurement screen, a T code, M code or reference position return can be specified by a manual numeric command.

Procedure for measuring a tool length offset

A tool length offset can be measured by manually bringing the tool into contact with a workpiece or a reference block.

For details of the actual operation, refer to the manual supplied by the machine tool builder.



- 1 **Place the machine in the tool replacement position by manual reference position return or another suitable operation.**
- 2 **Press the HANDLE or JOG switch among the mode selection switches.**
- 3 **Turn the tool offset measurement mode switch on the machine operator's panel to ON. The screen is automatically replaced by the tool length offset measurement screen, as shown below. On the screen, status indication OFST blinks.**

The tool length offset measurement screen for offset memories A and B (geometry compensation is distinguished from wear compensation) slightly differs from that for memory C (geometry compensation is distinguished from wear compensation, and cutter compensation is distinguished from length compensation).

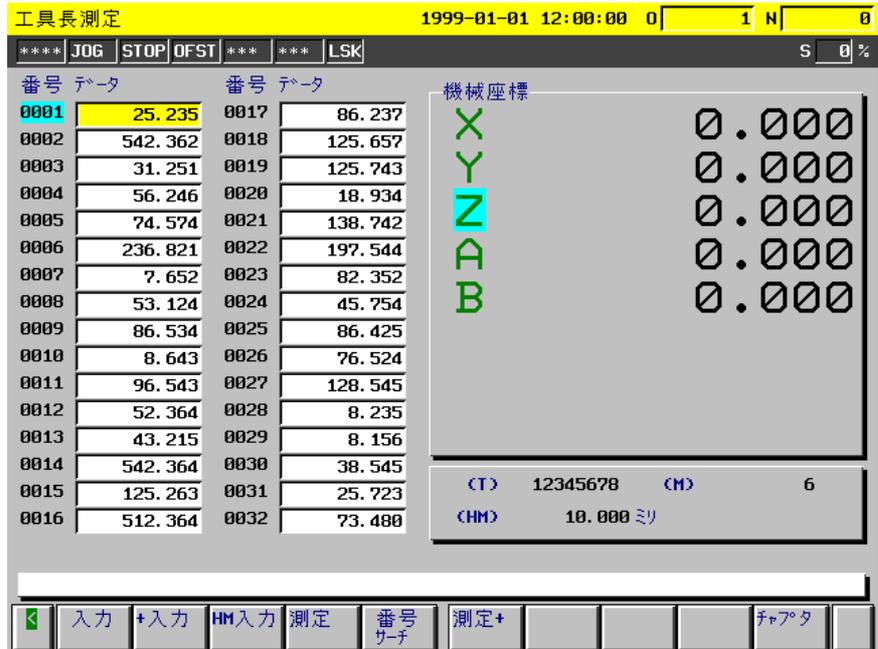


Fig. 9.10 (a) Tool Length Offset Measurement Screen (Offset Memory A)

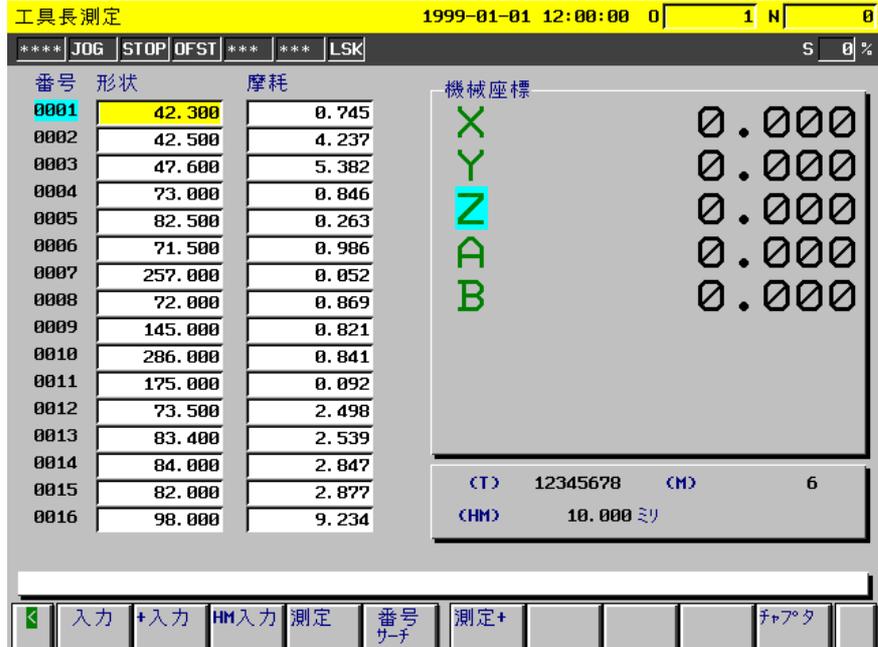


Fig. 9.10 (b) Tool Length Offset Measurement Screen (Offset Memory B)

工具長測定						1999-01-01 12:00:00 0		1 N		0	
*** JOG STOP OFST *** ** LSK								S		0%	
(長さ)			(半径)			機械座標					
番号	形状	摩耗	形状	摩耗							
0001	100.000	0.002	5.000	0.054	X	0.000					
0002	120.000	0.003	5.000	0.523	Y	0.000					
0003	104.000	0.003	6.500	0.653	Z	0.000					
0004	123.200	0.006	3.000	0.756	A	0.000					
0005	50.000	0.023	13.000	0.234	B	0.000					
0006	64.300	0.764	25.000	1.355							
0007	53.000	0.002	52.000	0.653							
0008	183.000	0.098	63.000	2.356							
0009	102.000	0.025	24.600	6.435							
0010	354.000	0.036	67.300	0.063							
0011	342.000	0.135	64.300	0.523							
0012	67.500	0.235	75.310	1.356							
0013	73.500	0.035	63.000	0.846							
0014	412.000	0.753	76.750	0.236							
0015	43.200	1.432	12.300	0.573							
0016	65.600	0.764	65.000	0.754							

(T)	12345678
(M)	6
(HM)	10.000ミリ

入力	+入力	HM入力	測定	番号	測定+				チャプタ
----	-----	------	----	----	-----	--	--	--	------

Fig. 9.10 (c) Tool Length Offset Measurement Screen (Offset Memory C)

NOTE

The values of addresses T and M can be reset to 0. In the MEM or MDI mode, the values of the modal T and M codes are displayed.

- 4 **Enter the distance between the reference measurement surface and the actual reference surface by using the numeric keys, then press the [HM_INP] soft key. The reference measurement surface and actual reference surface are described below.**
- 5 **Select the desired tool.**
While status indication OFST is blinking on the tool length offset measurement screen, a T code and M code can be programmed in manual handle feed or jog feed mode (manual numeric command). Key in Ttttt, and then press the cycle start button on the machine operator's panel or MDI panel. Then, select the tool to be measured by Ttttt. Next, M06 is generally entered and executed to position the tool to the spindle. Now, the target tool is set at the spindle position. Position the cursor to the tool offset number corresponding to the tool length offset to be set. Generally, the operator positions the cursor on the offset number. Some machines can automatically position the cursor to the corresponding tool offset number upon the completion of tool selection if the QNI bit (bit 6 of parameter 6002) is set to 1.
- 6 **Bring the tool into contact with the measurement surface of the workpiece or reference block by means of manual handle feed or jog feed.**

- 7 **Press the [MEASURE] soft key. The tool length offset is set in the tool offset memory. In tool offset memory B or C, the tool length offset is set as a tool geometry offset, and the tool wear offset is set to 0. The cursor does not move from the tool offset number. To automatically position the cursor to the next tool offset number after the setting is completed, press the [MEASUR+] soft key.**
- 8 **When the tool length offset is set, the tool automatically moves to the tool replacement position set by the TC2 bit (bit 0 of parameter 6003) and the TC3 bit (bit 1 of parameter 6003).**
- 9 **The procedure for measuring the tool length offset of a single tool is completed. For other tools, repeat steps 5 to 8.**
- 10 **When the tool length offset measurement of all tools is completed, turn off the tool length offset measurement mode switch on the machine operator's panel. OFST stops blinking on the tool length offset measurement screen.**

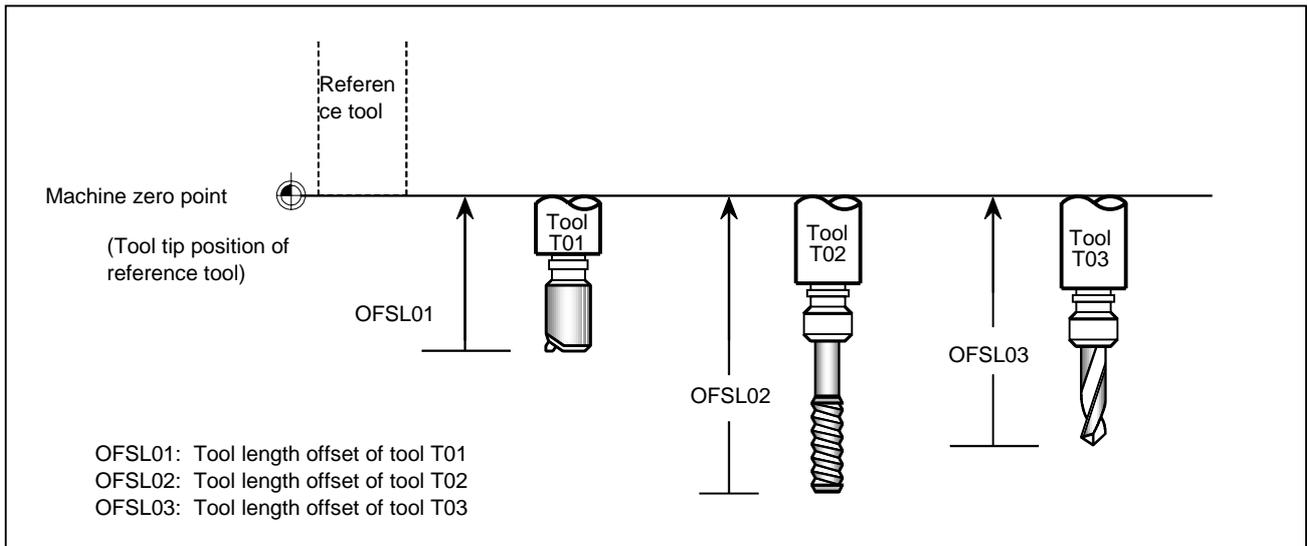
Description

- Definition of the tool length offset

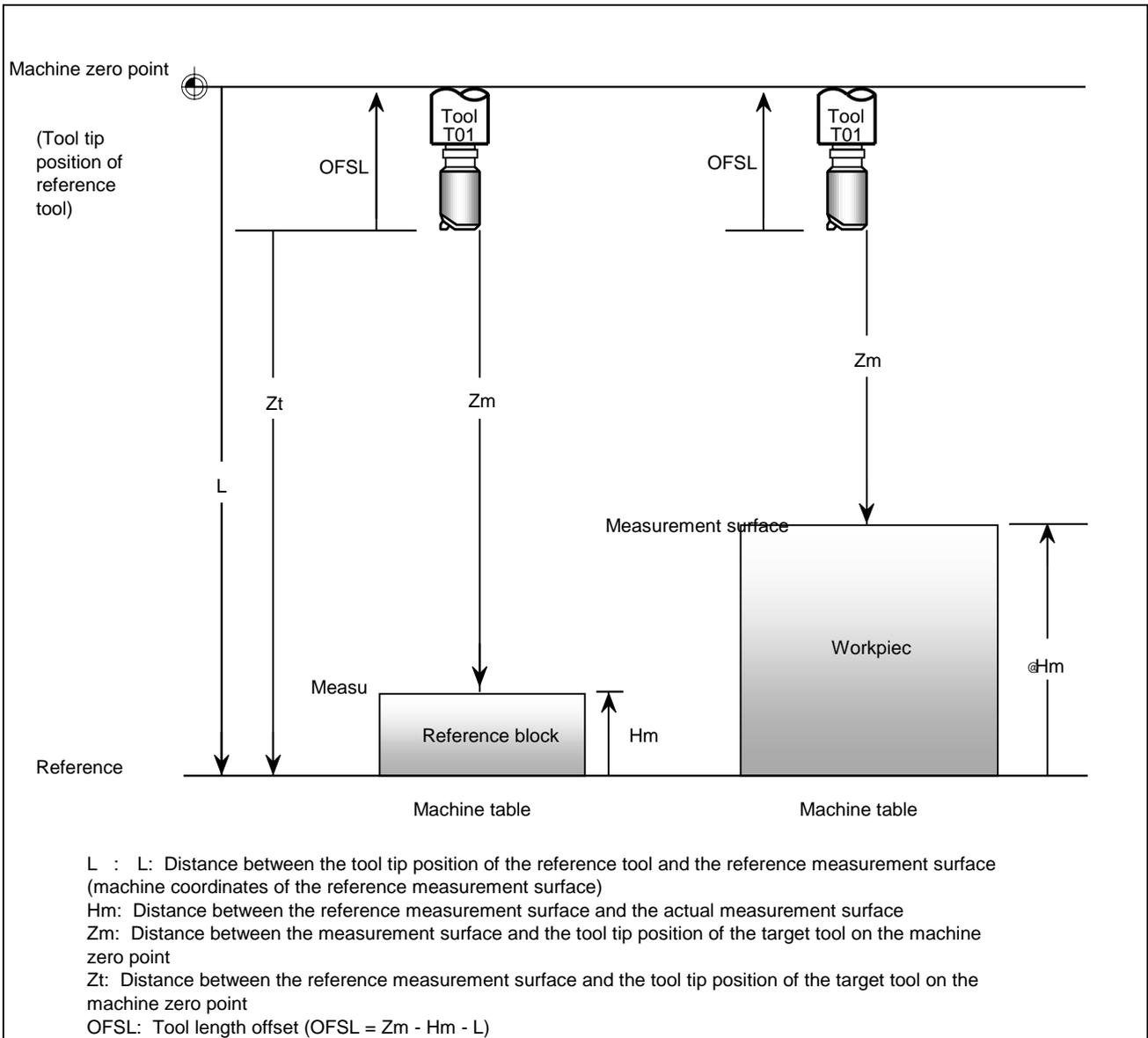
The tool length offset can generally be defined in two methods, as described below. The two methods share a single view that a tool length offset is the distance between the tool tip of a target tool and the tool tip of a reference tool.

(1) Method 1

The actual tool length is used as the tool length offset. In this method, the reference tool is a virtual tool of which tip is on the machine zero point when the machine is on the zero point on the Z-axis of the machine coordinate system. The tool length offset is the difference between the tool tip position of the target tool and the tool tip position of the reference tool, or the distance along the Z-axis between the machine zero point and the tool tip position of the target tool when the machine is at the machine zero point of the Z-axis.



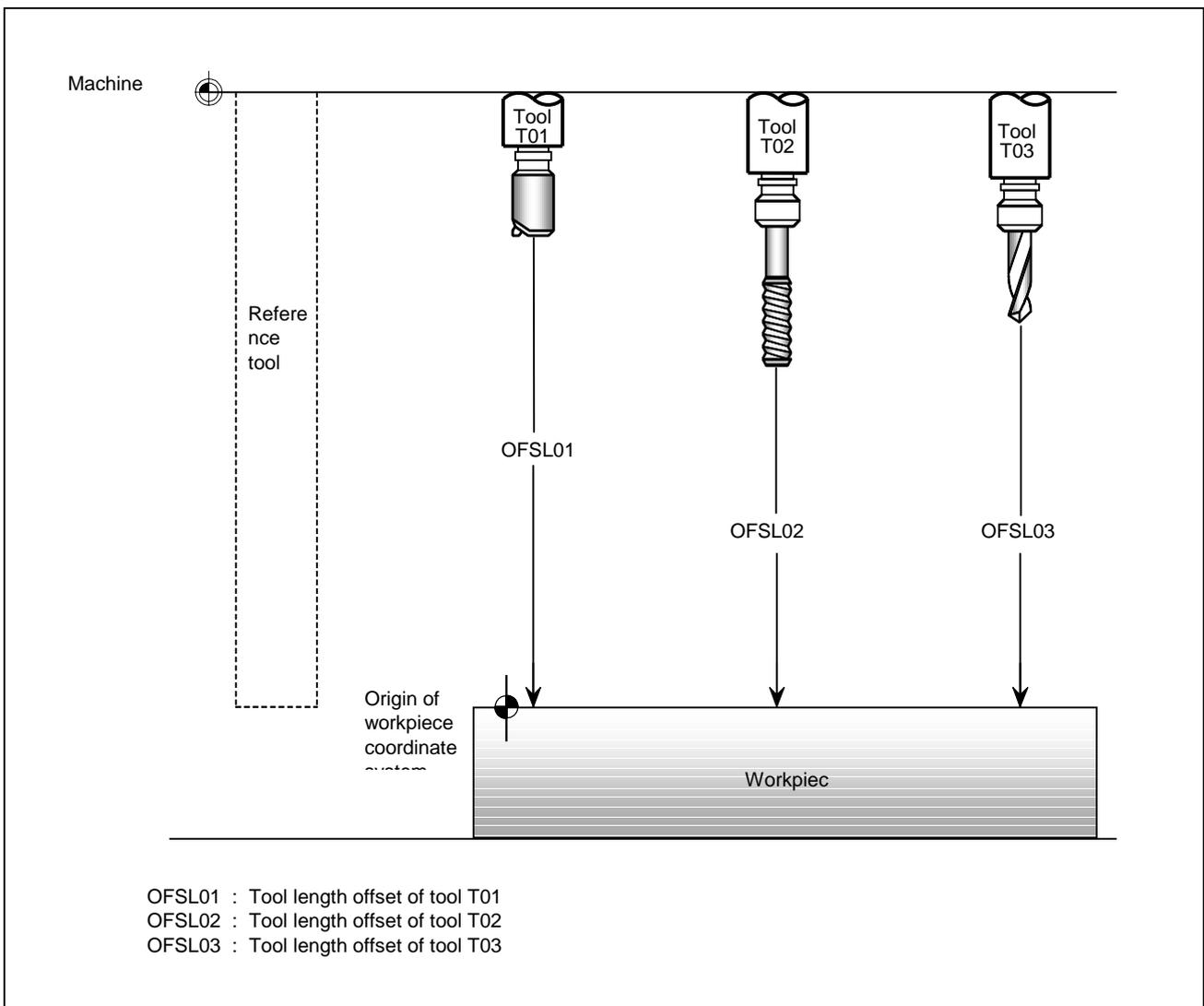
When a tool length is measured, the tool is manually moved until the tool tip comes into contact with the top surface of a workpiece or reference block. This surface is referred to as the measurement surface. This measurement surface cannot be the top surface of the machine table because doing so would cause the machine to be damaged. However, suppose that the top surface of the machine table is used as the measurement surface. The distance between the machine zero point and the top surface of the machine table is a machine-specific value that is referred to as distance L. Given that distance L is set in parameter 6024 and that the measurement surface is the top surface of the machine table, tool length offset OFSL can be easily obtained from L and the machine coordinates Zt of the tool when the tool touches the top surface of the machine table. Since the actual measurement surface cannot be the top surface of the machine table, the top surface of the machine table is defined as a reference measurement surface, and the distance between the reference measurement surface and the actual measurement surface, which is the height of a workpiece or reference block (Hm), is set separately (see the procedure). Then, tool length offset OFSL can be obtained as shown below:



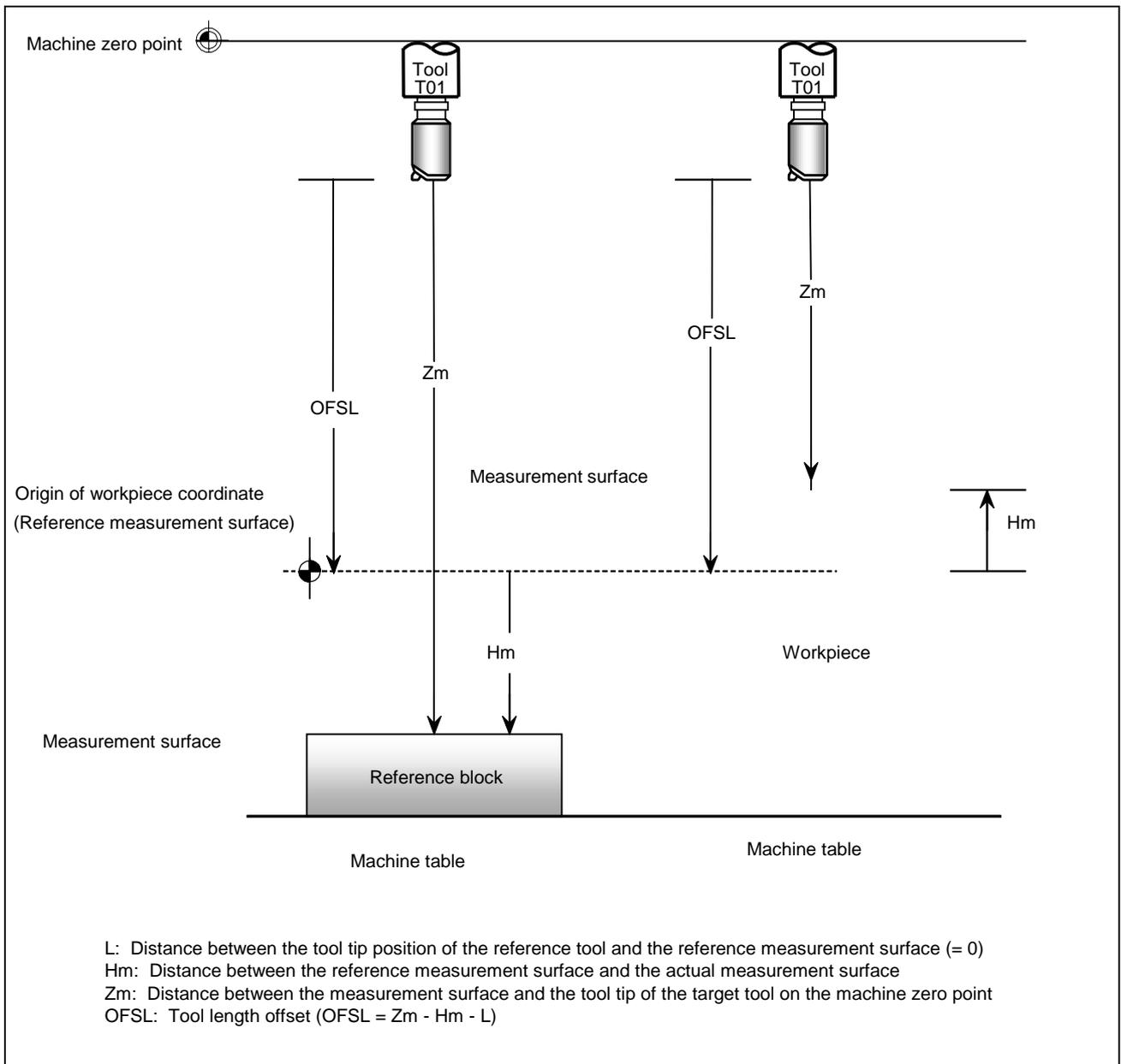
If the tool does not wear, an advantage of this method is that the tool length offset does not have to be adjusted for each new workpiece because the actual tool length is set as the tool length offset. If multiple workpieces are cut at one time, the need to adjust the tool length offset can be eliminated by assigning the workpiece coordinate system of each workpiece to G54 to G49 and setting the individual workpiece origin offsets.

(2) Method 2

The tool length offset is defined as the distance between the origin of the workpiece coordinate system and the tool tip position while the machine is positioned to the zero point of the Z-axis of the machine coordinate system. This method is the same as method 1 in that the difference from the length of the reference tool is used as the tool length offset. Method 2 differs from method 1 in that the reference tool is a virtual tool whose tool tip is positioned to the origin of the workpiece coordinate system while the machine is positioned to the zero point of the Z-axis of the machine coordinate system.



In method 2, the reference measurement surface is the origin of the workpiece coordinate system. The tool tip position of the reference tool is also the origin of the workpiece coordinate system. Accordingly, distance L between the tool tip position of the reference tool and the reference measurement surface is zero. Therefore, zero is specified in parameter 6024 of distance L. The actual measurement surface is generally on the same position as the reference measurement surface, which is the origin of the workpiece coordinate system. If the measurement surface is set as the top surface of the reference block or if the top surface of the workpiece is not set to the origin of the workpiece coordinate system (for instance, when the origin of the workpiece coordinate system is shifted from the top surface of the workpiece by a distance equal to the cutting allowance), the distance between the reference measurement surface and the actual measurement surface is separately set as Hm. Then, tool length offset OFSL can be calculated in the same way as in method 1.



The reference tool in method 2 is a tool whose tip is positioned to the origin of the workpiece coordinate system while the machine is positioned to the machine zero point. Generally, the tool length offset must be adjusted according to the workpiece. The need to adjust the tool length offset for a new workpiece can be eliminated by setting the difference between the origin of the workpiece coordinate system of the new workpiece and the origin of the workpiece coordinate system used to measure the tool length offset as the workpiece origin offset of any of G54 to G59. To define the tool length offset using this method involves setting the workpiece origin offset as the tool length offset for each tool.

ETool replacement position

When the tool length offset is set in tool length measurement by pressing the [MEASURE] or [MEASUR+] soft key, the tool automatically moves to the tool replacement position. A reference position is selected beforehand as the tool replacement position in the TC3 bit (bit 1 of parameter 6003) and the TC2 bit (bit 0 of parameter 6003).

TC3	TC2	Description
0	0	The tool replacement position is the first reference position.
0	1	The tool replacement position is the second reference position.
1	0	The tool replacement position is the third reference position.
1	1	The tool replacement position is the fourth reference position.

Procedure for measuring the workpiece origin offset

The workpiece origin offset can be easily measured not only on the longitudinal axis of the tool, or the Z-axis but also on the X-axis and Y-axis in the plane orthogonal to the Z-axis. The workpiece origin offsets on the X-axis and Y-axis can be easily measured both when a surface of a workpiece is set as the workpiece origin and when the center of a cutting hole is set as the workpiece origin.

For details of the actual operation, refer to the manual supplied by the machine tool builder.

Procedure for measuring the workpiece origin offset on the Z-axis

- 1 **Set any tool on the spindle position. The selection of the tool is made by an MDI command or the like (see the procedure for measuring the tool length offset). A tool for which tool length offset has already been measured must be selected.**
- 2 **Press the HANDLE or JOG switch of the mode selection switches.**
- 3 **Turn on the workpiece origin offset measurement mode switch on the machine operator's panel. The screen is automatically replaced by the workpiece origin offset screen. Status indication WOFS blinks on the screen.**

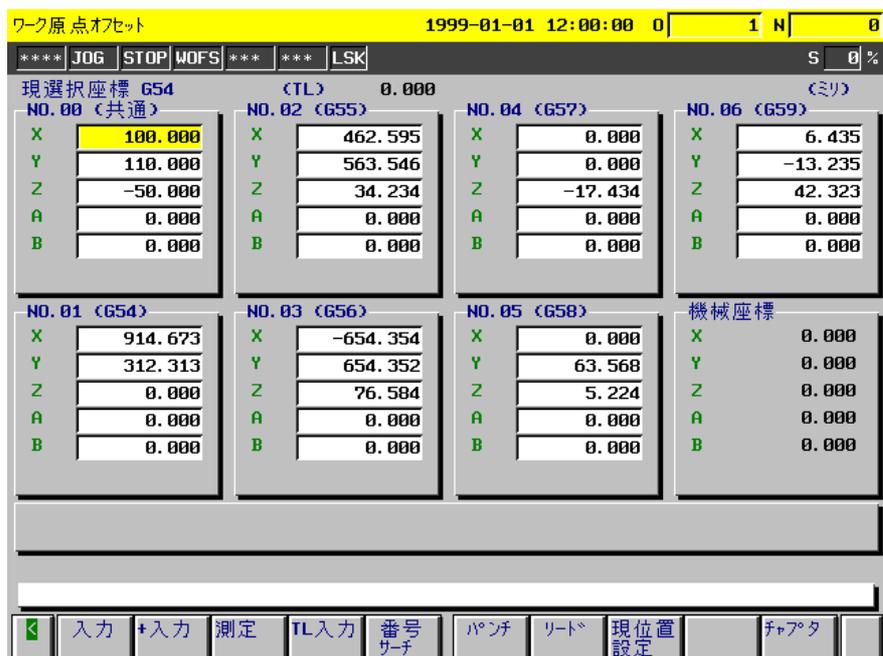
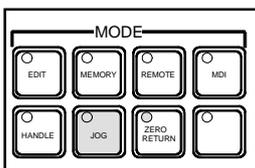
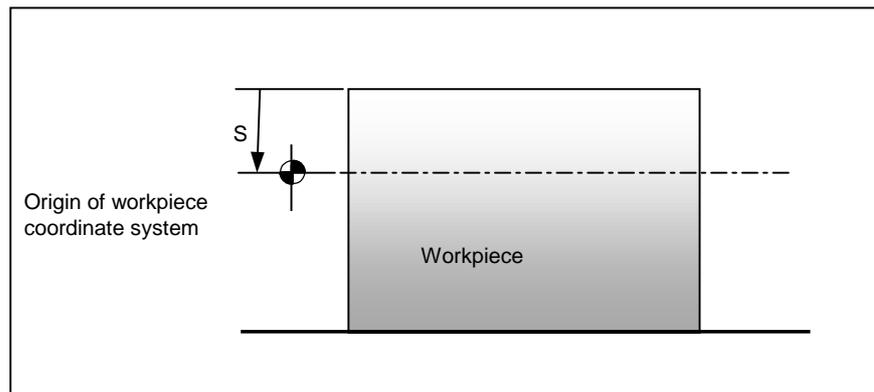


Fig. 1.1 (d) Workpiece Origin Offset Screen in Workpiece Origin Measurement

- 4 Enter the tool length offset for the selected tool on the workpiece origin offset screen. After entering the tool length offset by pressing numeric keys, press the [TL_INP] soft key.
- 5 Position the cursor to a desired workpiece origin offset of G54 to G59. The cursor may not be on the workpiece origin offset on the Z-axis.
- 6 Bring the tool into contact with the top surface of the workpiece by manual handle feed or jog feed.
- 7 Key in axis name Z. Then, press the [MEASURE] soft key and the [INPUT] soft key. Now, the workpiece origin offset on the Z-axis is specified. The cursor is positioned to the workpiece origin offset on the Z-axis. If a parameter setting is made such that the workpiece origin offset is measured only on the Z-axis (the WMA bit (bit 3 of parameter 6003) is set to 0), axis name Z need not be keyed in. If the top surface of the workpiece is not set as the origin of the workpiece coordinate system, for instance, when the origin of the workpiece coordinate system is shifted from the top surface of the workpiece by the distance corresponding to the cutting allowance, enter the corresponding value (S in the figure below) by pressing the numeric keys. Then, press the [MEASURE] soft key and the [INPUT] soft key.



- 8 To measure another workpiece origin offset, retract the tool from the workpiece. Then, repeat steps 5 to 7.

Procedure for measuring a workpiece origin offset on the X-axis or Y-axis, based on a reference surface

The measurement procedure in which the workpiece origin for the X-axis or Y-axis is set on a surface of a workpiece is basically the same as the procedure for measuring the workpiece origin offset on the Z-axis if the WMA bit (bit 3 of parameter 6003) is set to 1. The difference between the two procedures is found in step 4, in which the tool radius offset of the selected tool is input on the workpiece origin offset screen. After entering the tool radius offset with the numeric keys, press the [TL_INP] soft key.

CAUTION

When entering the tool radius offset, note the sign.

- When the measurement surface is in the positive direction of the tool, the offset must be entered with a minus sign.
- When the measurement surface is in the negative direction of the tool, the offset value must be entered with a plus sign.

Procedure for measuring the workpiece origin offset on the X-axis or Y-axis, based on a reference hole

When the WMH bit (bit 4 of parameter 6003) is set to 1, the workpiece origin offset on the X-axis or Y-axis can be measured, based on a reference hole.

- 1 Mount a measurement probe with a sensor on the spindle.**
- 2 Press the HANDLE or JOG switch of the mode selection switches.**
- 3 Set the workpiece origin offset measurement mode switch on the machine operator's panel to ON. The screen is automatically replaced by the workpiece origin offset screen. Status indication WOFS blinks on the screen to indicate that the preparations for measuring a workpiece origin offset have been made.**
- 4 Position the cursor to a desired workpiece origin offset of any of G54 to G59. The cursor need not be positioned to the workpiece origin offset on the X-axis or Y-axis.**
- 5 Move the measurement probe by manual handle feed or jog feed until it touches the lateral surface of the hole. The probe must be moved along one axis at a time.**
- 6 When the sensor detects that the tool has made contact, the machine inputs a skip signal to stop axial movement by manual**

handle feed or jog feed. At the same time, the position is stored as the first measurement point. The machine coordinates of the stored measurement point are displayed as HOLE MEASURED in the lower part of the screen, as shown below.

ワーク原点オフセット		1998-11-30 16:56:46 0		1 N		0	
*** JOG STOP WOF5 *** ** LSK S 0%							
現選択座標 G54		(TL) 10.000				(ミリ)	
NO.00 (共通)		NO.02 (G55)		NO.04 (G57)		NO.06 (G59)	
X	100.000	X	462.595	X	0.000	X	6.435
Y	110.000	Y	563.546	Y	0.000	Y	-13.235
Z	-50.000	Z	34.234	Z	-17.434	Z	42.323
A	0.000	A	0.000	A	0.000	A	0.000
B	0.000	B	0.000	B	0.000	B	0.000
NO.01 (G54)		NO.03 (G56)		NO.05 (G58)		機械座標	
X	914.673	X	-654.354	X	0.000	X	842.400
Y	312.313	Y	654.352	Y	63.568	Y	266.400
Z	0.000	Z	76.584	Z	5.224	Z	0.000
A	0.000	A	0.000	A	0.000	A	0.000
B	0.000	B	0.000	B	0.000	B	0.000
穴基準測定位置		#1 X	268.800	#2 X	634.800	#3 X	842.400
		Y	169.200	Y	-355.200	Y	266.400
<input type="button" value="←"/> <input type="button" value="入力"/> <input type="button" value="+入力"/> <input type="button" value="測定"/> <input type="button" value="TL入力"/> <input type="button" value="番号サーチ"/> <input type="button" value="パンチ"/> <input type="button" value="リート"/> <input type="button" value="現位置設定"/> <input type="button" value="チャータ"/>							

Fig. 1.1 (e) Workpiece Origin Offset Screen Displayed in Workpiece Origin Measurement

- 7 Move the measurement probe to a second measurement point. Meanwhile, the CNC locks the movement in the direction toward the current measurement point. For instance, if the probe is moved from a measurement point to which the probe was moved in the +X direction, the CNC allows movement in the X direction and locks the movement in the +X, +Y, and -Y directions until the skip signal goes to 0. The operation performed after contact is made at the second measurement point is the same as that after the first measurement point.
- 8 Bring the probe into contact with a third measurement point, then press the [MEASURE] soft key and the [CENTER] soft key. Then, the center is obtained from the coordinates of the three measurement points, and the workpiece origin offsets on the X-axis and Y-axis are specified. To cancel the measurement prior to its completion, press the key. When the key is pressed, the coordinates of the measurement points stored before that are all cleared.

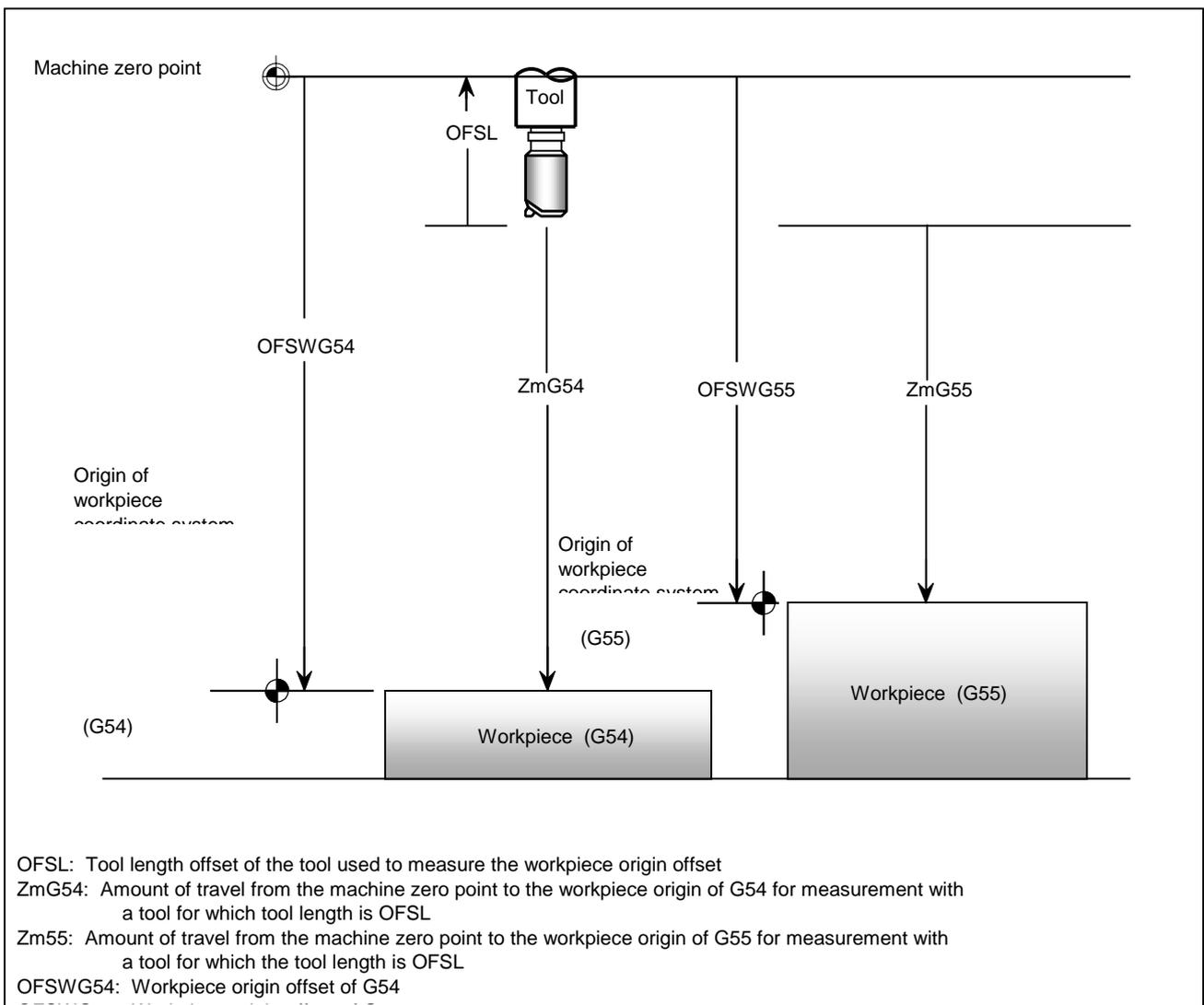
Description

Workpiece origin offset on the Z-axis

Methods 1 and 2 in "Definition of the Tool Length Offset" in the description of the procedure for measuring the tool length offset generally apply to the definition of workpiece origin offset on the Z-axis, as follows:

(1) Method 1

In method 1, the workpiece origin offset on the Z-axis is the distance between the origin of the machine coordinate system and the origin of the workpiece coordinate system, as shown below:



As shown in the figure, the workpiece origin offset on the Z-axis can be obtained as follows:

$$OFSW = Z_m - OFSL$$

where

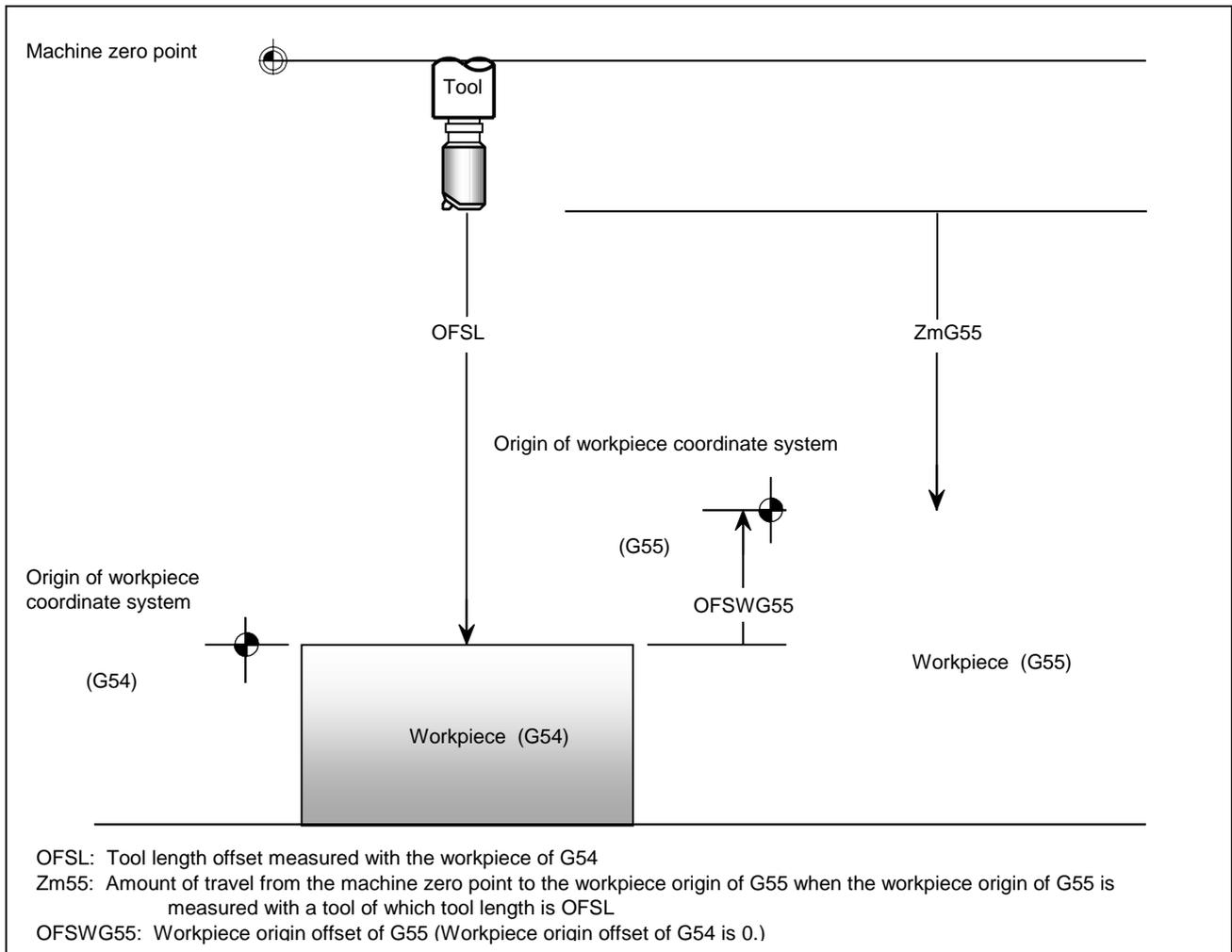
OFSW: Workpiece origin offset

OFSL: Tool length offset of a tool used to measure the workpiece origin offset

Z_m: Amount of travel from the machine zero point to the workpiece origin for measurement with a tool having a tool length of OFSL

(2) Method 2

The tool length offset in method 2 is the workpiece origin offset on the Z-axis, as described above. Therefore, the workpiece origin offset need not be set in most cases. If the workpiece origin offset is set as shown below, the need to measure the tool length offset for a new workpiece is eliminated even when the workpiece is changed after the tool length offset for the workpiece is measured or when multiple workpiece coordinate systems are assigned to G54 to G59 to cut multiple workpieces at the same time:



The workpiece origin offset can be obtained using method 2, in the same way as in method 1.

$$\text{OFSW} = \text{Zm} - \text{OFSL}$$

$$\text{OFSW} = \text{Zm} - \text{OFSL}$$

where

OFSW: Workpiece origin offset

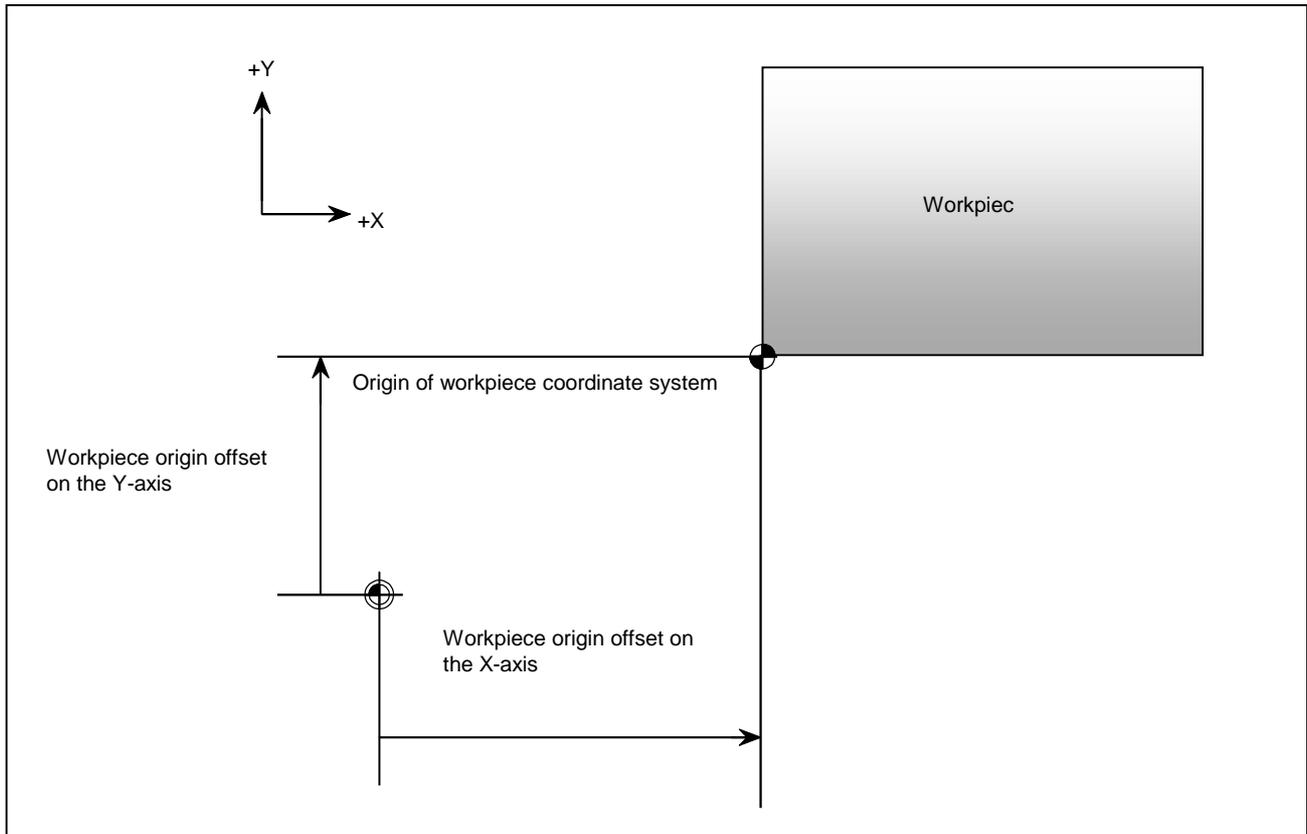
OFSL: Tool length offset of a tool with which the workpiece origin offset is measured

Zm: Amount of travel from the machine zero point to the workpiece origin when measurement is made with a tool having a tool length of OFSL

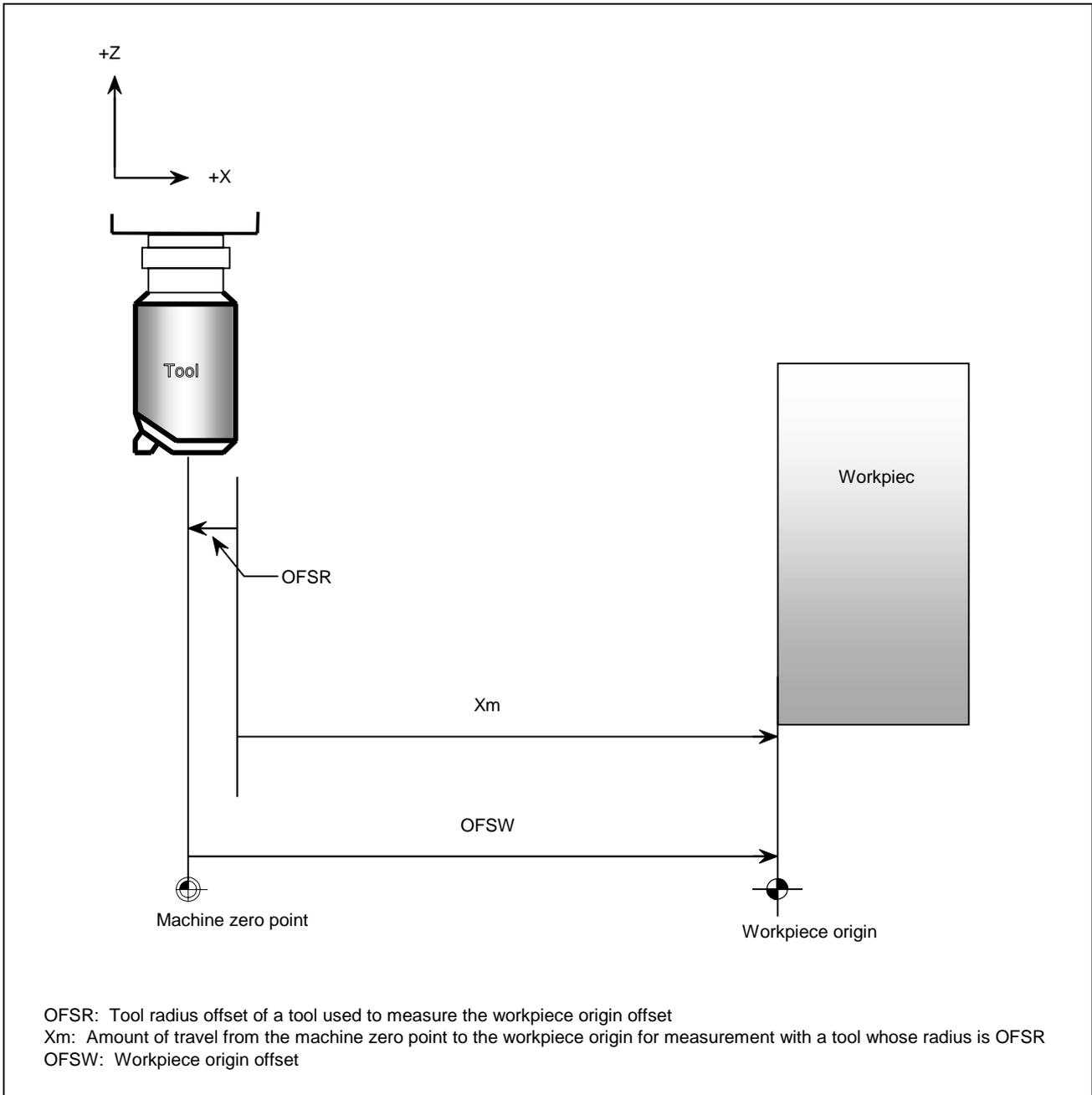
▣ Workpiece origin offset on the X-axis or Y-axis

The workpiece origin offset of the X-axis or Y-axis can easily be measured both when the surface of a workpiece is set as the workpiece origin and when the center of a cutting hole is set as the workpiece origin.

(1) When a surface is set as the workpiece origin



A side face of a workpiece is set as the workpiece origin, in the figure shown above. The offset measurement in which the workpiece origin on the X-axis or Y-axis is set on a face of a workpiece is basically the same as the measurement for the workpiece origin offset on the Z-axis. The two measurements differ in that the tool length of a tool used for measurement is considered to obtain the workpiece origin offset on the Z-axis while the tool radius offset is considered for the X-axis or Y-axis.



As shown above, the workpiece origin offset can be obtained as follows:

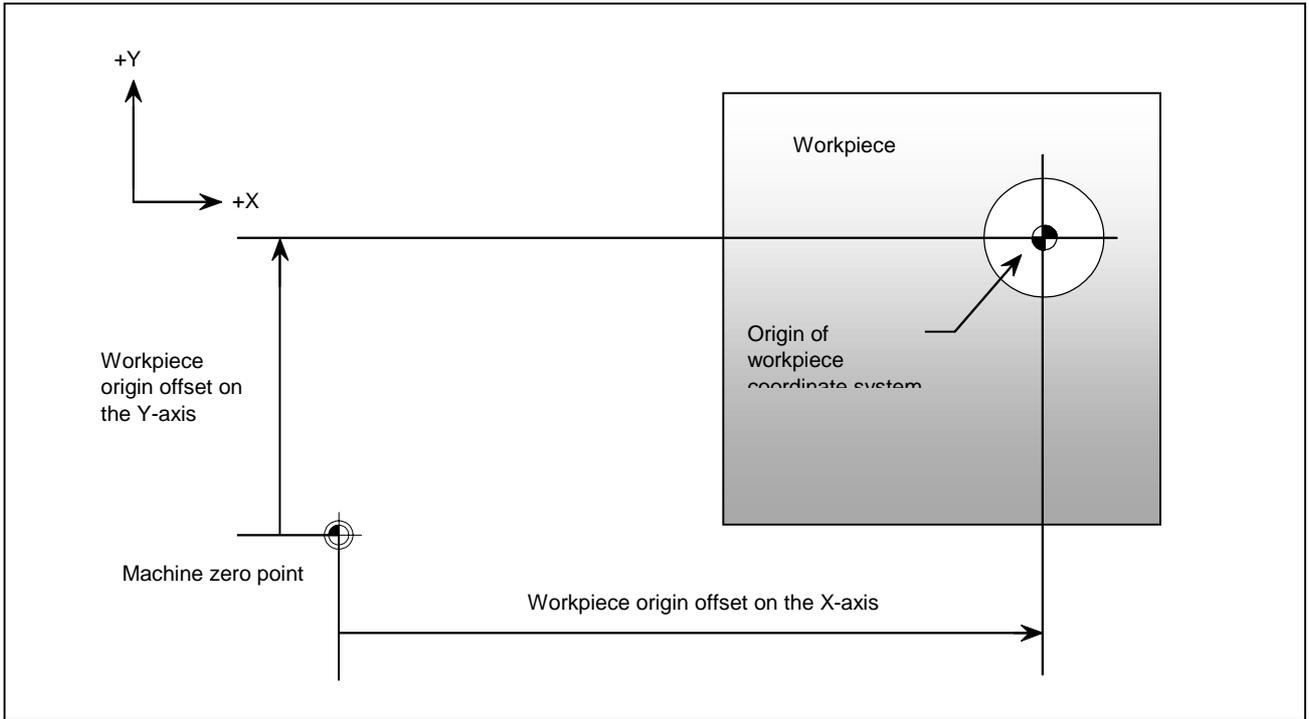
$$\text{OFSW} = \text{Xm} - \text{OFSR}$$

Note the sign of tool radius offset OFSR.

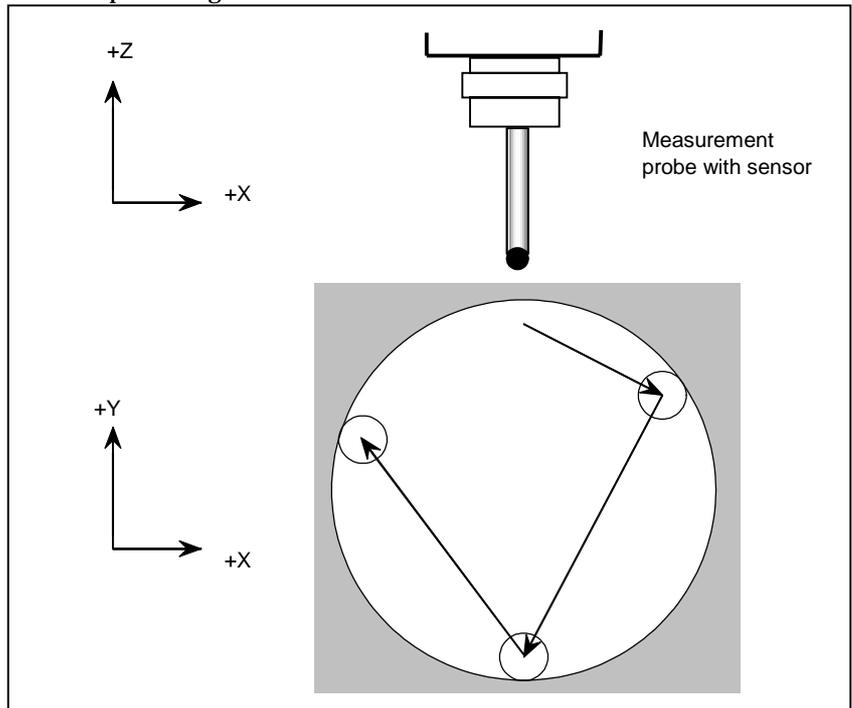
When the measurement surface is in the positive direction of the tool center, the sign of OFSR must be minus.

When the measurement surface is in the negative direction of the tool center, the sign of OFSR must be positive.

(2)When the center of a hole is set as the workpiece origin



In the example shown above, the center of a hole in a workpiece is set as the workpiece origin. In this case, the positions of any three points on the lateral surface of the hole are measured with a measurement probe having a sensor at its tip. Meanwhile, the skip signal (SKIP) is set to 1. A circle passing through three points is uniquely defined. The center of the circle is set as the workpiece origin on the X-axis and Y-axis.



9.11 MENU SWITCHES

With the menu switch function, some DI signals entered with the signal (BMI interface) from the machine can be set by CNC screen operation.

Screen configuration

To display the menu switch screen, follow the procedure described in Section III.x.x, "Offset Setting."

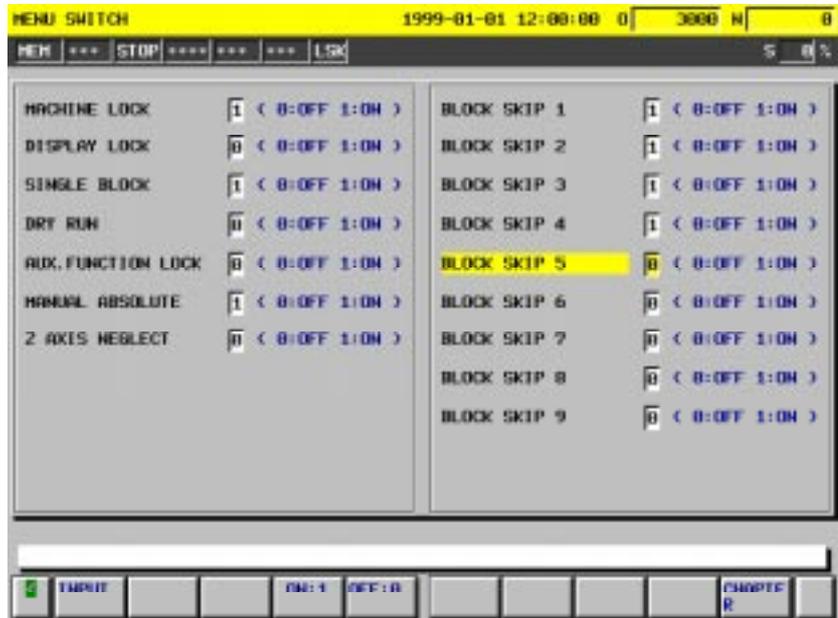


Fig.9.11 (a) Menu Switch Screen

£ Types of switches

The menu switch function can handle the following signals:

Switch name	Signal
Machine lock	MLK
Display lock	DLK
Single block	SBK
Dry run	DRN
Auxiliary function lock	AFL
Manual absolute	ABS
Z-axis ignore	Z-axis MLK(*1)
Block skip 1 to 9	BDT1 'BDT9

NOTE
 *1 This is MLK for the third axis, and is not always MLK for the Z-axis, depending on the machine configuration.

Operation

£ **Positioning the cursor**

Using the cursor keys  , position the cursor.

The cursor keys   cannot be used.

£ **Turning on a switch**

Method 1.Press the [ON:1] soft key.

Method 2.Press the numeric key then the key.

Method 3.Press the numeric key , then press the [INPUT] soft key.

£ **Turning off a switch**

Method 1.Press the [OFF:0] soft key.

Method 2.Press the numeric key then the key.

Method 3.Press the numeric key , then press the [INPUT] soft key.

CAUTION

If a menu switch and the corresponding machine signal are used at the same time, the input state is the logical OR of both switches.
 If a menu switch is turned off, but the corresponding machine signal is on, for example, the actual input state is on.

9.12 DISPLAYING AND SETTING TOOL LIFE MANAGEMENT DATA

Tool life management data can be displayed on the screen to check the current tool life management status. In addition, the screen can be used to edit tool life management data.

The screen consists of the following two types of data:

- . Tool life management (directory)
- . Tool life management (group editing)

Displaying tool life management data

Tool life management (directory)

To display the tool life management directory screen, follow the procedure described in Section III.x.x, "Offset Setting."



Fig.9.12 (a) Tool Life Management (Directory) Screen

£ Data (A)

Group numbers that belong to the groups listed below and override values are displayed. If there is no group number belonging to a group below, **** is displayed instead of a group number.

Next group used: Tool group number for which service life count is to be started with the next M06 command

Group currently used: Tool group number for which service life count is currently being performed

Group currently selected: Tool group number for which service life count is currently being performed, or tool group number for which life count operation was performed at the end

Count override: When the tool life count override signal is disabled (with bit 3 (LFV) of parameter No. 7401 set to 0), 1.0 times is indicated.

£ Data (B)

The set service life values, current tool life counter values, and registered tool numbers (in order of use) of each group are displayed. When the time-based service life count type is specified, the display and setting units used for set service life values and tool life counters is determined by bit 0 (FCO) of parameter No. 7403, as follows:

Bit 0 (FCO) of parameter No. 7403	0	1
Unit of display and setting of service life values and service life counters	1 minute	0.1 minute

The table below indicates the meaning of a mark prefixed to each tool number:

Tool state	Used	Not used
Service life remains.	@	No indication
Skip	■	
Service life has expired.	↑(*1)	

*1 When bit 2 (EMD) of parameter No. 7401 is set to 0, @ continues to be indicated until the next tool is selected, even if the service life has expired.

The service life counter indicates a count value for a tool marked with @. An asterisk (*) is displayed for a tool when the group to which the tool belongs is next specified.

One page displays information for multiple groups. The number of groups displayed on one page depends on the number of groups and the number of tools in each group.

£ Data (C)

Group numbers for which the tool change signal is output are displayed. If one screen cannot display all of those group numbers, -- -> is displayed. In other cases, **** is displayed at the end.

Tool life management (group editing)

Display the tool life management group editing screen by pressing the soft key on the tool life management directory screen.



Fig.9.12 (b) Tool Life Management (Group Editing) Screen

£ Data (A)

As with tool life management (directory), group numbers that belong to the groups listed below and override values are displayed. If there is no group number belonging to a group below, **** is displayed instead of a group number.

Next group used: Tool group number for which life count is to be started with the next M06 command

Group currently used: Tool group number for which life count is currently being performed

Group currently selected: Tool group number for which life count is currently being performed, or tool group number for which life count was performed at the end

Count override: When the tool life count override signal is disabled (with bit 3 (LFV) of parameter No. 7401 set to 0), 1.0 times is indicated.

£ Data (B)

The detailed tool life management data for a selected group is displayed as described below.

Type	1 Count-based life counting
	2 Time-based life counting
Life value	Tool life value
Counter	Tool life counter
Status	Indicates that the service life has expired.
	Indicates that a skip command was received.
	Indicates that the tool is currently being used.

The life counter indicates a count value for a tool marked with @. An asterisk (*) is displayed for a tool when the group to which the tool belongs is next specified.

T code	Tool number
H code	Tool length compensation value specification code
D code	Cutter compensation value specification code

Setting/editing tool life management data

Tool life management (directory)

£ Switching between pages

Using the page keys  , the previous or following groups can be displayed.

£ Selecting a group

Method 1

- (1) Press the [GROUP NUMBER] soft key.
- (2) Enter a group number.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter a group number.
- (2) Press the [GROUP NUMBER] soft key.

£ Presetting the tool life counter

The value of the tool life counter of a specified group can be updated.

Method 1

- (1) Select MDI mode.
- (2) Press the [PRESET COUNT] soft key.
- (3) Key in a desired counter value.

(4) Press the [EXEC] soft key.

Method 2

- (1) Select MDI mode.
- (2) Key in a desired counter value.
- (3) Press the [PRESET COUNT] soft key.

£ Clearing execution data

All execution data of a specified group up to the current point can be cleared.

Method 1

- (1) Press the [CANCEL] soft key.
- (2) Enter a group number.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter a group number.
- (2) Press the [CANCEL] soft key.

£ Skipping of tools

Among the tools registered in a specified group, the tool to be selected next is skipped.

Method 1

- (1) Press the [SKIP] soft key.
- (2) Enter a group number.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter a group number.
- (2) Press the [SKIP] soft key.

Tool life management (group editing)

Tool life management data can be edited when MDI mode is set, and the automatic operation stop state, automatic operation halt state, or reset state (not during a reset) is set. However, tool life management data cannot be edited while data is being set by a program. The following editing operations are possible:

- . Setting of service life count type, service life values, counter, and tool data (T code, H code, D code)
- . Addition of tool groups
- . Addition of tool numbers (T code)
- . Deletion of tool groups
- . Deletion of tool data (states, T code, H code, D code)

- . Setting for tool skip
- . Setting for tool clearing (restart of service life)

NOTE

Editing must not be performed for the group for which life count operation is to be started with the next M06 command, or for that group for which life count operation is being performed. Otherwise, the life management of the group is terminated forcibly.

2 When any of the following editing operations is performed, the tool change signal may be set to 1:

- £ Tool skip setting for the last tool
- £ Tool number deletion from a tool group resulting in the tool group containing only those tools whose lives have expired, as well as skipped tools

3 When any of the following editing operations is performed, the tool change signal may be set to 0:

- £ Tool number addition to a tool group resulting in the tools whose lives have not expired being set in the tool group
- £ Setting for tool clearing

£ **Switching between pages**



With the page keys , the previous or following groups can be displayed.

£ **Selecting a group**

Method 1

- (1) Press the [GROUP NUMBER] soft key.
- (2) Enter a group number.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter a group number.

- (2) Press the [GROUP NUMBER] soft key.

£ Switching between screens

Pressing the [SIGHT] soft key returns the screen display to the tool life management directory screen.

£ Setting of life count type, life values, counter, and tool data

Method 1

- (1) Press the [INPUT] soft key.
- (2) Enter a number you want to set.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Enter a number you want to set.
- (2) Press the [INPUT] soft key.

Method 3

- (1) Enter a number you want to set.
- (2) Press the  key.

NOTE

Even if a modification is made to a life value or life counter value, the tool state and tool change signal are not affected.

2

After making a life count type change, set a life value and life counter value again.

£ Adding a tool group**Method 1**

- (1) Select a group for which no data is set.
- (2) Position the cursor to the tool data (state, T code, H code, or D code).
- (3) Press the [ADD T CODE] soft key.
- (4) Enter a tool number.
- (5) Press the [EXEC] soft key.

Method 2

- (1) Select a group for which no data is set.
- (2) Position the cursor to the tool data (state, T code, H code, or D code).
- (3) Enter a tool number.
- (4) Press the [ADD T CODE] soft key.

A service life count type is set according to bit 3 (LTM) of parameter No. 7400. All of the service life values, service life counter value, H code, and D codes are set to 0.

£ Adding a tool number**Method 1**

- (1) Position the cursor to the tool data (state, T code, H code, or D code) immediately preceding the number to be added.
- (2) Press the [ADD T CODE] soft key.
- (3) Enter a tool number.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Position the cursor to the tool data (state, T code, H code, or D code) immediately preceding the number to be added.
- (2) Enter a tool number.
- (3) Press the [ADD T CODE] soft key.

(Example) Adding tool number 1500 between No. 1 and No.

- (1) Position the cursor to the data of displayed number 1. Then, enter 1500 and press [ADD T CODE].

	TOOL NO.	POT#	LENGTH	RADIUS
001	00000200	0002	6.260	1.341
002	00000112	0012	34.235	4.324
003	00000113	0013	100.240	24.523

- (2) The entered T code 1500 is added at the position of No. 2. H code and D code are set to 0.

	TOOL NO.	POT#	LENGTH	RADIUS
001	00000200	0002	6.260	1.341
002	00000600			
003	00000112	0012	34.235	4.324

£ Deleting a tool group

- (1) Select the group you want to delete.
- (2) Press the [DELETE] soft key.
- (3) Press the [GROUP] soft key.
- (4) Press the [EXEC] soft key.

£ Deleting tool data

- (1) Position the cursor to the tool data (state, T code, H code, or D code) of the tool you want to delete.
- (2) Press the [DELETE] soft key.
- (3) Press the [<CURSOR>] soft key.

NOTE

The deletion of all tools from a tool group has the same effect as deleting the tool group.
 2 If a tool marked with @ (currently used) is deleted, the @ mark moves to the preceding tool whose service life has expired, or moves to a skipped tool.

£ Setting tool skip

Method 1

- (1) Position the cursor to the tool data (state, T code, H code, or D code) of the tool you want to skip.

- (2) Press the [STATE] soft key.
- (3) Press the [SKIP] soft key.

Method 2

- (1) Position the cursor to the state of a tool you want to skip.
- (2) Press the [INPUT] soft key.
- (3) Press the address key.
- (4) Press the [EXEC] soft key.

Method 3

- (1) Position the cursor to the state of a tool you want to skip.
- (2) Press the address key.
- (3) Press the [INPUT] soft key.

Method 4

- (1) Position the cursor to the state of a tool you want to skip.
- (2) Press the address key.
- (3) Press the key.

£ Setting for tool clearing (restart of service life)

Method 1

- (1) Position the cursor to the tool data (state, T code, H code, or D code) of the tool you want to clear.
- (2) Press the [STATE] soft key.
- (3) Press the [CLEAR] soft key.

Method 2

- (1) Position the cursor to the state of the tool you want to clear.
- (2) Press the [INPUT] soft key.
- (3) Press the address key.
- (4) Press the [EXEC] soft key.

Method 3

- (1) Position the cursor to the state of the tool you want to clear.
- (2) Press the address key.
- (3) Press the [INPUT] soft key.

Method 4

- (1) Position the cursor to the state of the tool you want to clear.

(2) Press the  address key.

(3) Press the  key.

9.13 DISPLAYING AND SETTING THE SOFTWARE OPERATOR'S PANEL

MDI panel operations can substitute for the functions of switches on the machine operator's panel.

For example, mode selection and jog feed override selection can be made by operations on the MDI panel, thus allowing the corresponding switches to be eliminated from the machine operator's panel.

In addition, jog feed can be performed using the numeric keys.

Displaying the software operator's panel

Follow the procedure described in Section III.x.x, "Offset Setting" to display the software operator's panel screen.

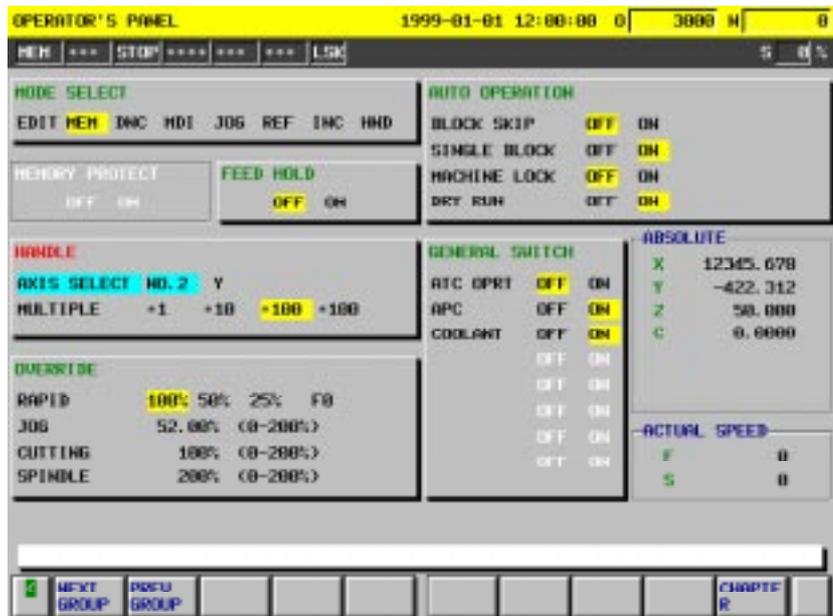


Fig.9.13 (a) Software Operator's Panel Screen

The switches are divided into seven groups. Those groups and switches whose names are displayed in white cannot be used.

£ **Mode selection**

Select operation mode.

£ **Memory protection**

Set the memory protection state.

£ Feed hold

Set feed hold state.

£ Manual handle

Select the axis on which a movement is to be made with the first handle, and also select a magnification.

£ Override

Set an override value for the rapid traverse rate, jog feedrate, cutting feedrate, and spindle speed.

£ Automatic operation

Set optional block skip, single block, machine lock, and dry run.

£ General-purpose switches

Eight general-purpose switches are available. A name not longer than eight characters can be assigned to each switch. A named switch is enabled. For the function of each switch, refer to the manual provided by the machine tool builder.

Operating and setting the software operator's panel

£ Selecting a group

Select a group with [NEXT GROUP], [PREV GROUP], or the cursor

keys  . The group name of the selected group is displayed in red.

£ Selecting data

Select data with the cursor keys  . A light blue cursor is displayed in the data name and data selected.

£ Setting data

Change data settings with the cursor keys  . The override values below can be set for the jog feedrate/cutting feedrate/spindle speed.

Jog feedrate override

25 overrides: 0, 0.1, 0.14, 0.2, 0.27, 0.37, 0.52, 0.72, 1.0, 1.4, 2.0, 2.7, 3.7, 5.2, 7.2, 10, 14, 20, 27, 37, 52, 72, 100, 140, and 200%

Cutting feedrate override

21 overrides: 0, 10, 20, 30, 40, 50, 60, 70, 80, 90, 100, 110, 120, 130, 140, 150, 160, 170, 180, 190, and 200%

Spindle speed

21 overrides: 0, 10, 20, 30, 40, 50, 60, 70, 80, 90, 100, 110, 120, 130, 140, 150, 160, 170, 180, 190, and 200%

Jog feed based on numeric keys

By parameter setting (No. 2020, No. 2021), jog feed can be specified using the MDI keys after assigning move axes to the MDI keys (1 to 4, 6 to 9). As move axes, the first axis to the sixth axis can be specified. No other axes can be specified.

When an open CNC is used, and a standard display/input unit for the CNC (MDI with arrow keys) is not installed or is unusable (with the CNC screen display function being used), jog feed based on numeric keys is unusable.

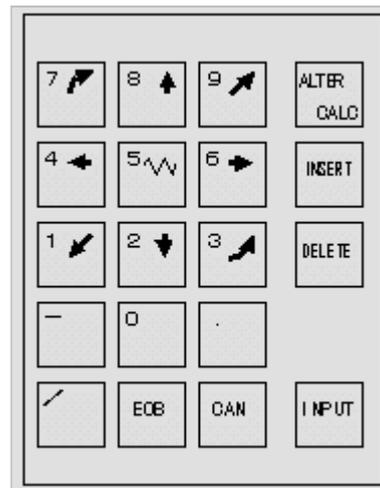


Fig.9.13 (b) Numeric Keys on a Display/Input

Procedure

- (1) To display the software operator's panel screen, follow the procedure described in Section III.x.x, "Offset Setting."
This function is available only when the software operator's panel screen is displayed.
- (2) Select JOG mode.
- (3) Pressing an MDI key enables jog feed.

- (4) Pressing the .5 key together with a numeric key from 1 to 4 or 6 to 9 performs jog rapid traverse.

9.14 DISPLAYING AND SETTING TOOL OFFSET DATA BASED ON TOOL NUMBERS

On the tool offset data screen, the tool length compensation values, cutter compensation values, and tool pot numbers for tool numbers (T code) can be displayed and set.

Displaying tool offset data

Follow the procedure described in Section III.x.x, "Offset Setting" to display the tool offset data screen based on tool numbers.

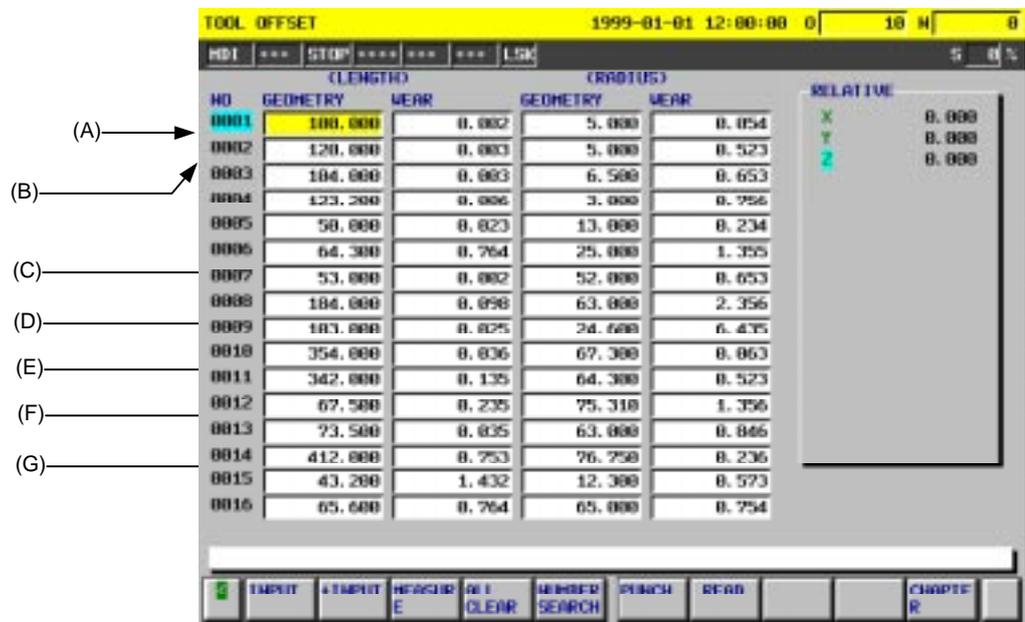


Fig.9.14 (c) Tool Offset Data Screen

£ (A) Display number

A display number is displayed at the leftmost position of the screen. A display number is used to edit and search for tool data (tool numbers, tool pot numbers, tool length compensation values, cutter compensation values). When tool data is deleted or inserted, the tool data corresponding to the display numbers is changed.

£ (B) Tool number

A tool number is used as a T code in an NC program.

£(C) Pot #

Pot # represents a pot number. When a tool number is specified, the pot number corresponding to the tool number is output as a code signal together with a strobe signal to the machine. For data for which the same pot number is set, an asterisk (*) is prefixed to the tool pot numbers.

£(D) Length

Length represents a tool length compensation value. When a tool number is specified, the tool compensation value corresponding to the tool number is used for tool length compensation.

£(E) Radius

Radius represents a cutter compensation value. When a tool number is specified, the cutter compensation value corresponding to the tool number is used for cutter compensation.

£(F) Relative coordinates

Relative coordinates are displayed.

**£(G) Tool offset increment system
/execution/selection**

The increment system of tool offset values, and the display number of the tool data that is currently being executed/selected are displayed. The tool data that is currently being executed is that for which compensation is enabled (with M06 specified after the T code). The currently selected tool data is that which is selected (with T code selected but M06 not specified). When no tool data is currently executed/selected, *** is displayed.

Selecting and setting tool offset data

**Entering and modifying
tool offset data****Method 1**

- (1) Position the cursor to the data you want to enter or modify.
- (2) Press the [INPUT] soft key.
- (3) Enter data.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Position the cursor to the data you want to enter or modify.
- (2) Enter data.

(3) Press the [INPUT] soft key.

Method 3

- (1) Position the cursor to the data you want to enter or modify.
- (2) Enter data.
- (3) Press the  key.

Offset data increment system

As an offset, a number with or without a decimal point can be entered. The increment system of an offset depends on the parameter settings, as indicated below.

Parameter				Offset value increment system (valid data range)	
OFE	OFD	OFC	OFA	Metric input	Inch input
0	0	0	1	0.01 (}9999.99)	0.001 (}999.999)
0	0	0	0	0.001 (}9999.999)	0.0001 (}999.9999)
0	0	1	0	0.0001 (}9999.9999)	0.00001 (}999.99999)
0	1	0	0	0.00001 (}9999.99999)	0.000001 (}999.999999)
1	0	0	0	0.000001 (}999.999999)	0.0000001 (}99.9999999)

- OFA : Bit 0 of parameter No. 6002
- OFC : Bit 1 of parameter No. 6002
- OFD : Bit 0 of parameter No. 6004
- OFE : Bit 0 of parameter No. 6007

Adding a tool number

Method 1

- (1) Position the cursor to the data immediately preceding the display number you want to add.
- (2) Press the [INSERT] soft key.
- (3) Press the [(T#)] soft key.
- (4) Enter a tool number.
- (5) Press the [EXEC] soft key.

Method 2

- (1) Position the cursor to the data immediately preceding the display number you want to add.
- (2) Press the [INSERT] soft key.
- (3) Press the  address key.
- (4) Enter a tool number.
- (5) Press the [EXEC] soft key.

Method 3

- (1) Position the cursor to the data immediately preceding the display number you want to add.
- (2) Press the  address key.
- (3) Enter a tool number.
- (4) Press the [INSERT] soft key.

(Example) Adding tool number 600 between display number 1 and display number 2

- (1) Position the cursor to the data of display number 1, then enter T600 and press [INSERT].

	TOOL NO.	POT#	LENGTH	RADIUS
001	00000200	0002	6.260	1.341
002	00000112	0012	34.235	4.324
003	00000113	0013	100.240	24.523

- (2) Tool number 600 is added as tool number 2. The pot number, length, and radius fields are blank.

	TOOL NO.	POT#	LENGTH	RADIUS
001	00000200	0002	6.260	1.341
002	00000600			
003	00000112	0012	34.235	4.324

NOTE

- 1 If 300 sets of data have already been registered, a warning (excessive data count) is issued, disabling data insertion.
- 2 If there are two or more identical tool numbers, a warning (duplicate data) is issued. In this case, the duplicate tool number or numbers are not set, and only fields are reserved.
- 3 If a T is entered without specifying a tool number (numeric value), no tool number is set, and only fields are reserved.

**Deleting tool offset data
(1 line)**

Method 1

- (1) Press the [DELETE] soft key.
- (2) Key in the display number of the data you want to delete.
- (3) Press the [EXEC] soft key.

Method 2

- (1) Key in the display number of the data you want to delete.
- (2) Press the [DELETE] soft key.

Method 3

- (1) Position the cursor to the data you want to delete.
- (2) Press the [DELETE] soft key.
- (3) Press the [<CURSOR>] soft key.

**Deleting tool offset data
(several lines)****Method 1**

- (1) Press the [DELETE] soft key.
- (2) Key in the display number of the first line of data you want to delete.
- (3) Press the [, (NUMBER)] soft key.
- (4) Key in the display number of the last line of data you want to delete.
- (5) Press the [EXEC] soft key.

Method 2

- (1) Press the [DELETE] soft key.
- (2) Key in the display number of the first line of data you want to delete.
- (3) Press the  address key.
- (4) Key in the display number of the last line of data you want to delete.
- (5) Press the [EXEC] soft key.

Method 3

- (1) Key in the display number of the first line of data you want to delete.
- (2) Press the  address key.
- (3) Key in the display number of the last line of data you want to delete.
- (4) Press the [DELETE] soft key.

Searching for tool numbers**Method 1**

- (1) Press the [SEARCH] soft key.
- (2) Press the [(T#)] soft key.

- (3) Enter a tool number.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Press the [SEARCH] soft key.
- (2) Press the  address key.
- (3) Enter a tool number.
- (4) Press the [EXEC] soft key.

Method 3

- (1) Press the  address key.
- (2) Enter a tool number.
- (3) Press the [SEARCH] soft key.

Searching for pot numbers**Method 1**

- (1) Press the [SEARCH] soft key.
- (2) Press the [(P#)] soft key.
- (3) Enter a pot number.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Press the [SEARCH] soft key.
- (2) Press the  address key.
- (3) Enter a pot number.
- (4) Press the [EXEC] soft key.

Method 3

- (1) Press the  address key.
- (2) Enter a pot number.
- (3) Press the [SEARCH] soft key.

To continue searching for the same pot number

Method 4

- (1) Press the [SEARCH] soft key.
- (2) Press the [(P#)] soft key.
- (3) Press the [EXEC] soft key.

Method 5

- (1) Press the [SEARCH] soft key.
- (2) Press the  address key.
- (3) Press the [EXEC] soft key.

Method 6

- (1) Press the  address key.
- (2) Press the [SEARCH] soft key.

NOTE

If a search is made based on the same pot number (method 4 to method 6) without performing even one search operation based on a pot number (method 1 to method 3), a warning indicating that the operation cannot be performed is issued.

Searching for the last tool offset data

- (1) Press the [SEARCH] soft key.
- (2) Press the [LAST] soft key.

Tool offset data input/output

Tool offset data input

Method 1 (Neither the input file name nor number need be specified.)

- (1) Select MDI mode.
- (2) Press the [READ] soft key.
- (3) Press the [NUMBER OFFSET] soft key.

Method 2 (The input file is specified with its file name.)

- (1) Select MDI mode.
- (2) Press the [READ] soft key.
- (3) Press the ["FILE NAME] soft key.
- (4) Enter a file name.
- (5) Press the [FILE NAME"] soft key.
- (6) Press the [NUMBER OFFSET] soft key.

Method 3. The input file is specified with its file number (one of three methods).

- (1) Select MDI mode.
- (2) Press the [READ] soft key.
- (3) Press the [FILE] soft key.
- (4) Enter a file number.
- (5) Press the [NUMBER OFFSET] soft key.

Method 4.The input file is specified with its file number (two of three methods).

- (1) Select MDI mode.
- (2) Press the [READ] soft key.
- (3) Press the address key.
- (4) Enter a file number.
- (5) Press the [NUMBER OFFSET] soft key.

Method 5.The input file is specified with its file number (three of three methods).

- (1) Select MDI mode.
- (2) Press the address key.
- (3) Enter a file number.
- (4) Press the [READ] soft key.

Tool offset data output

Method 1 (when neither an output file name nor a file number is specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [NUMBER OFFSET] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [NUMBER OFFSET] soft key.

Method 3 (The input file is specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE #)] soft key.
- (3) Enter a file number.
- (4) Press the the [NUMBER OFFSET] soft key.

Method 4 (The input file is specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the address key.
- (3) Enter a file number.
- (4) Press the [NUMBER OFFSET] soft key.

Method 5 (The input file is specified with its file number (three of three methods))

- (1) Press the  address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

- | | |
|---|--|
| 1 | When method 1 is used for tool offset data output (when neither an output file name nor a file number is specified) for output to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, or memory card, file name TOOL-DAT.TXT is selected |
| 2 | For input to or output from a FANUC Floppy Cassette, FANUC FA Card, or FANUC Handy File, an input/output file can be specified using a file name or file number. |
| 3 | For input to or output from a memory card, an input/output file can be specified using a file name. |

Input/output using other screens

Tool offset data input/output is possible on the floppy directory screen and memory card screen in addition to the tool offset data screen. For details, see Section III.x.x, "Floppy Directory Screen" and Section III.x.x, "Memory Card Screen."

Measurement

The [T CODEOFFSET] soft key is used for tool length measurement. For details of the function, see Section III.x, "Tool Length Measurement."

Relationships with other functions**£ Tool life management**

When tool offset based on a tool number is enabled (bit 5 (NOT) of parameter No. 0011 is set to 0), D/H code cannot be registered in the tool life management data.

£ External data input/output

Tool data can be entered or output using the external data input/output function.

£ Tool length automatic measurement

When tool length automatic measurement (G37) is specified, the tool length compensation value of the tool number currently being used is updated.

Custom macro system variables

With the system variables for tool compensation, no tool offset values can be read or updated based on tool numbers.

Warning

Message	Description
Excessive data count	An attempt was made to add a tool number when 300 sets of tool offset data have already been registered.
Duplicate data	An attempt was made to add a tool number that has already been registered.
The operation cannot be performed.	An attempt was made to make a search based on the same pot number without performing even one search operation based on a pot number.

9.15 FLOATING REFERENCE POSITION SETTING

The floating reference position is a position on the machine used as a reference for a machine operation.

The floating reference position is not always fixed, but can be moved, depending on the situation.

Before G30.1 can be specified to return to a floating reference position, the floating reference position must be set.

Setting a floating reference position

Procedure

£ When setting a floating reference position on all axes

- (1) Follow the procedure described in Section III.x.x, "Offset Setting" to display the overall position display screen, relative position display screen, or absolute position display screen.
- (2) Move the machine to the floating reference position by jog feed.
- (3) Press the [MEM_FRP] soft key.
- (4) Press the [ALL AXES] soft key.

£ When setting a floating reference position on particular axes

Method 1

- (1) Follow the procedure described in Section III.x.x, "Offset Setting" to display the overall position display screen, relative position display screen, or absolute position display screen.
- (2) Move the machine to the floating reference position by jog feed.
- (3) Press the [MEM_FRP] soft key.
- (4) Key in the desired axis names, then press the [EXEC] soft key.

Axis names can be entered in succession.

Example) [EXEC]

Method 2

- (1) Follow the procedure described in Section III.x.x, "Offset Setting" to display the overall position display screen, relative position display screen, or absolute position display screen.
- (2) Move the machine to the floating reference position by jog feed.
- (3) Key in the desired axis.

Axis names can be entered in succession.

9.SETTING AND DISPLAYING DATA

(4) Press the [MEM_FRP] soft key.

Example) [MEM_FRP]

Checking settings

Settings can be checked using parameter No. 1244.

Presetting the relative coordinate system

With bit 1 (FPC) of parameter No. 1200, a floating reference position can be memorized, and the relative position indication can be preset to 0 as well.

9.16 SERVO SPINDLE SCREEN

The servo spindle screen is used to set servo spindle parameters and monitor the operation status.

The user can set servo spindle parameters and monitor the operation status without being concerned with parameter and diagnosis data numbers.

For details, refer to "FANUC Series 15i/150i-MODEL A Maintenance Manual (B-63325EN)."

9.17 COMMUNICATION SETTING SCREEN

The communication setting screen is used to set RS-232C, RS-422, and input/output devices such as memory cards.

Display

Follow the procedure described in Section III.x.x, "Offset Setting" to display the communication setting screen.



Fig.9.17 (a) Communication Setting Screen

Basic setting

Set the general communication protocol requirements. The table below indicates the relationships between the setting items and parameters.

Setting item name	Parameter	Setting item display (parameter setting)
TV check	TVC iNo.0000#0 j	On (1)/Off (0)
TV (comment)	CTV iNo.0000#1 j	On (1)/Off (0)
Input/output code	ISP iNo.0000#2 j EIA iNo.0000#4 j	EIA (#2=0/1 ,#4 =1) ISO (#2=0 ,#4 =0) ASCII (#2=1 ,#4 =0)
EOB code	NCR iNo.0000#3 j	LF CR CR(0) ^LF(1)
F.G input device F.G output device B.G input device B.G output device	iNo.0020 j iNo.0021 j iNo.0022 j iNo.0023 j	RS232-C C1 (1) RS232-C C2 (2) RS232-C C3 (3) Memory card (8) Remote buffer (10) RS-422 C1 (13) Open CNC1 (15) (DNC operation interface) Open CNC2 (16) (Upload/download interface)

£ Setting RS-232C/RS-422

Set the communication conditions for each channel of RS-232C and RS-422.

A parameter set specifying a device specification, stop bit, and baud rate as indicated below is assigned to each channel according to the device number setting using parameter No. 5001 to No. 5003 and No. 5013.

Parameter set 1 : Parameters (Nos. 5110, 5111, 5112)

Parameter set 2 : Parameters (Nos. 5120, 5121, 5122)

⋮

Parameter set 9 : Parameters (Nos. 5190, 5191, 5192)

Setting item name	Parameter	Setting
Device specification	iNo.5110,5120,5130 j iNo.5140,5150,5160 j iNo.5170,5180,5190 j	1 8
Stop bit	iNo.5111,5121,5131 j iNo.5141,5151,5161 j iNo.5171,5181,5191 j	1 2

Baud rate	iNo.5112,5122,5132 j iNo.5142,5152,5162 j iNo.5172,5182,5192 j	1 √12 (50, 100, 110, 150, 200, 300, 600, 1200, 2400, 4800, 9600, 19200bps)
-----------	--	--

£ [Tip]

A brief description of the setting to which the cursor is positioned is provided.

Setting data

1) Select MDI mode.

2) With the cursor keys    , position the cursor to the item you want to set.

3) Select a setting with the [SELECT+] and [SELECT-] soft keys.

NOTE

1 For channels whose device numbers (parameter No. 5001 to No. 5003, No. 5013) are not set, the following default parameter sets are selected:

RS-232C channel 1: Parameter set 1 (Nos. 5110, 5111, 5112)

RS-232C channel 2: Parameter set 2 (Nos. 5120, 5121, 5122)

RS-232C channel 3: Parameter set 3 (Nos. 5130, 5131, 5132)

RS-422 channel 1: Parameter set 4 (Nos. 5140, 5141, 5142)

2 If a value that falls outside the specifiable range is specified in a stop bit or baud rate parameter, the following defaults are internally selected and displayed on the screen, even though the defaults are not reflected in the parameter:

Stop bit: 2

Baud rate: 9 (4800 bps)

3 If a value that falls outside the specifiable range is specified in a device specification parameter, "UNKNOWN" is indicated.

Floppy directory screen

Pressing the [RS232C] soft key switches the display screen to the floppy directory screen. The floppy directory screen enables data input/output through a serial interface channel set with the F.G input device item, F.G output device item, B.G input device item, or B.G output device item. For details, see Section III.8.x, "Floppy Directory Screen."

9.18 HIGH-SPEED HIGH-PRECISION MACHINING SETTING SCREEN

The high-speed high-precision machining setting screen displays parameters associated with high-speed high-precision machining. This screen can be used to save and switch between three types of parameter sets defining different finish levels.

This screen can also be used to set basic parameters for high-speed high-precision machining easily with the automatic setting function and automatic adjustment function.

Displaying the high-speed high-precision machining setting screen

Adjustment screen

To display the adjustment screen, follow the procedure described in Section III.x.x, "System."

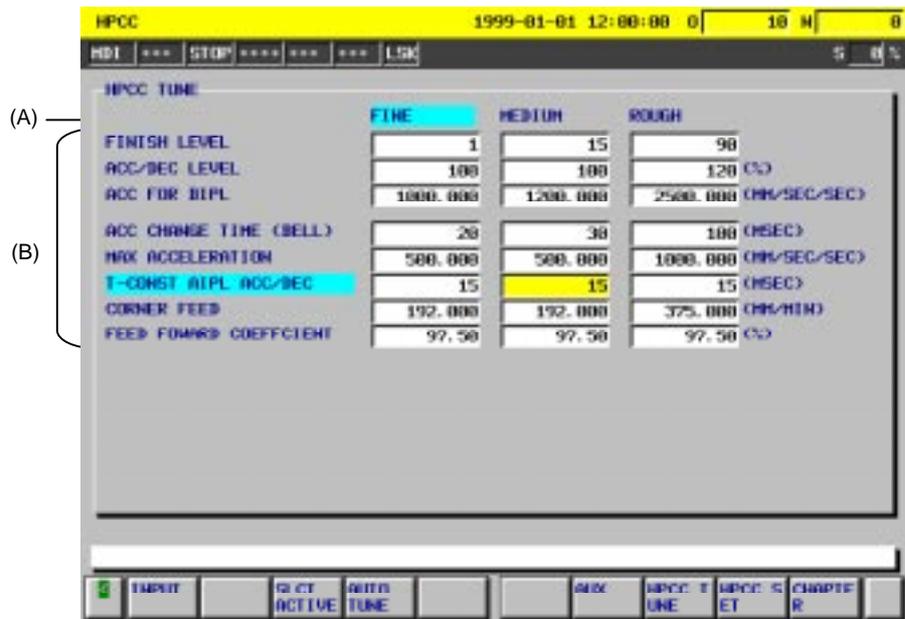


Fig.9.18 (a) Adjustment Screen

£ (A) Parameter set

The state of the parameter set used by the machining program is displayed. Three types of parameter sets are supported: the parameter set for finish machining, the parameter set for semifinish machining, and the parameter set for rough machining. A selected parameter set is displayed in reverse video.

£(B) Setting items

The settings of the items that can be set on this screen are reflected in the parameters.

A different parameter number is used to reflect the setting of the same item when a different parameter set is used.

Finish level (FINISH LEVEL)

This item specifies the finish level used for automatic adjustment calculation. The units have no dimensions. When finish level 1 is specified, a target precision of about 10 μ m is assumed for calculation. When finish level 10 is specified, a target precision of about 100 μ m is assumed for calculation. A value of between 1 and 100 can be set.

Acceleration/deceleration level (ACC/DEC LEVEL)

This item is used to make a fine adjustment of a set acceleration/deceleration value. The value of this item functions as an override value of a set acceleration/deceleration value. A value from 1 to 200(%) can be set. When 0 is specified, 100% is assumed.

Acceleration for acceleration/deceleration before interpolation (ACC FOR BIPL)

This item specifies an allowable acceleration value for acceleration/deceleration before interpolation on an axis-by-axis basis. The unit is mm/sec² or inch/sec², depending on the machine unit. A value of between 0 and 99999.999 can be set.

Acceleration change time (bell-shaped) (ACC CHANGE TIME(BELL))

This item specifies the acceleration time required to reach the maximum acceleration value of bell-shaped acceleration/deceleration before interpolation. The value set in this item is also used as a deceleration time required to reach an acceleration value of 0 from the maximum acceleration value. The unit is msec, and a value of between 0 and 200 can be set.

Maximum acceleration (MAX ACCELERATION)

This item specifies an allowable acceleration value to be used in a curvature area such as circular curves and very small straight lines. The unit is mm/sec² or inch/sec², depending on the machine unit. A value of between 0 and 99999.999 can be set.

Time constant for acceleration/deceleration after interpolation (T-CONST
AIPL ACC/DEC)

This item specifies an acceleration time required to reach a feedrate specified for acceleration/deceleration after interpolation. The unit is msec, and a value of between 0 and 4000 can be set.

Corner feedrate difference (CORNER FEED)

When the feedrate difference between two blocks exceeds the value set in this item, the tool is decelerated at the corner to prevent the value set in this item from being exceeded. The unit is mm/sec or inch/sec, depending on the machine unit. A value from 0 to 999999.999 can be set.

Feed-forward coefficient (FEED FORWARD COEFFICIENT)

This item specifies a coefficient used for feed-forward control. A value from 0.00 to 100.00(%) can be set.

Table 9.18 (a) Items and Parameter Numbers

	Finish machining	Semifinish machining	Rough machining
Finish level	1520	1540	1560
Acceleration/deceleration level	1533	1553	1573
Acceleration for acceleration/deceleration before interpolation	1534	1554	1574
Acceleration change time (bell-shaped)	1536	1556	1576
Maximum acceleration	1535	1555	1575
Time constant for acceleration/deceleration after interpolation	1522	1542	1562
Corner feedrate difference	1524	1544	1564
Feed-forward coefficient	1529	1549	1569

Setting screen

The setting screen can be displayed by pressing the [HPCC SET] soft key on the adjustment screen.

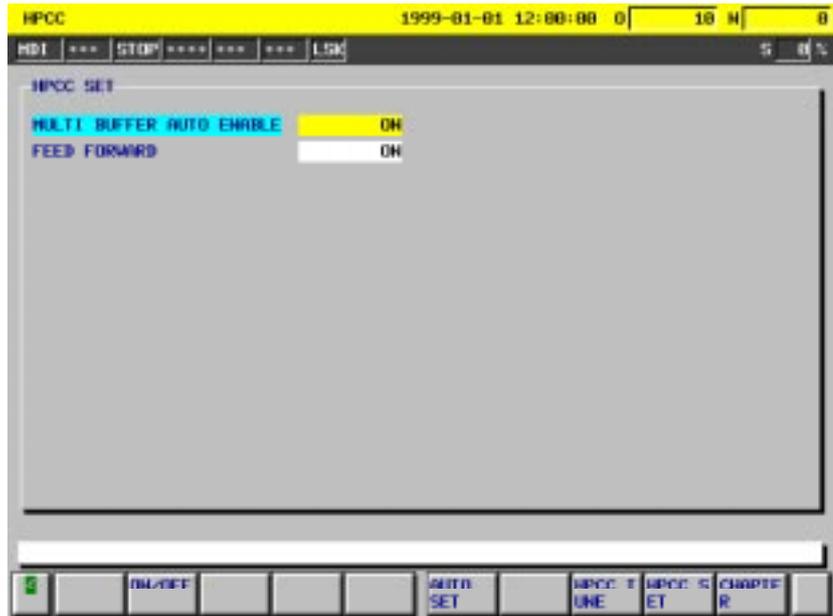


Fig.9.18 (b) Setting Screen

£ Setting items

The settings of the items that can be set on this screen are reflected in the parameters.

Modal multibuffer enable (MULTI BUFFER AUTO ENABLE)

This item specifies whether to make the multibuffer mode modal. The setting of this item is reflected in bit 6 (MBF) of parameter No. 2401.

Feed forward (FEED FORWARD)

This item specifies whether to enable or disable feed forward control. The setting of this item is reflected in bit 1 of parameter No. 1883.

Automatic setting

The minimum number of parameters required for high-speed high-precision machining is set automatically. This operation must be performed at least once before high-speed high-precision machining can be performed. Note, however, that only guideline values are set in automatic setting. Final settings must be determined based on the specifications of each machine and the results of automatic setting.

Procedure

- (1) Display the setting screen, then press the [AUTO SET] soft key.
- (2) The parameters listed in the table below are set. The previous parameter settings are lost because they are overwritten. When a parameter of axis type is used, the same value is set for all axes except an axis specified by bit 7 (NFF) of parameter No. 1009.

Parameter	Setting
1808#3 PI control	1
1883#1 Feed forward control enable	1
1884#4 High-speed velocity feedback	1
1962 Velocify feed-forward coefficient	50 (50%)
1518 Lower feed-forward coefficient limit for automatic calculation	9400 (94%)

- (3) Automatic setting can be disabled by setting bit 0 (SET) of parameter No. 1517 as follows:

Parameter SET.No.1517#0.

When the bit is set to 0: Automatic setting is enabled.

When the bit is set to 1: Automatic setting is disabled.

Automatic adjustment

Based on the cutting level (finish level) of a specified parameter set, the settings of the other items are automatically calculated, and set in the parameters. Note, however, that only guideline values are set in automatic adjustment. Final settings must be determined based on the specifications of each machine and the results of automatic adjustment.

Procedure

- (1) Set the following parameter and items of a parameter set to be automatically adjusted:
 - .Finish level
 - .Acceleration for acceleration/deceleration before interpolation
 - .Parameter No. 1825 for servo loop gain

If any of the items/parameters above is set to 0, a warning (insufficient automatic adjustment data) is issued, and no automatic adjustment is made.

- (2) Press the [AUTOMATIC ADJUSTMENT] soft key.
- (3) Select a parameter set with the [FINE]/[MEDIUM]/[ROUGH] soft key.
- (4) Each item is set. The parameters are overwritten, so that the previous settings are lost. When a parameter of axis type is used, the same value is set for all axes except the axis specified by bit 7 (NFF) of parameter No. 1009.
- (5) Automatic adjustment can be disabled using the parameters below.
 - Bit 1 (FIN) of parameter No. 1517
 - When this bit is set to 0: Automatic adjustment of finish machining is enabled.
 - When this bit is set to 1: Automatic adjustment of finish machining is disabled.
 - Bit 2 (MED) of parameter No. 1517
 - When this bit is set to 0: Automatic adjustment of semifinish machining is enabled.
 - When this bit is set to 1: Automatic adjustment of semifinish machining is disabled.
 - Bit 3 (ROU) of parameter No. 1517
 - When this bit is set to 0: Automatic adjustment of rough machining is enabled.
 - When this bit is set to 1: Automatic adjustment of rough machining is disabled.

Displaying parameter numbers

The parameter numbers in which the adjustment screen items are set can be checked.

Procedure

- (1) Display the adjustment screen, then press the [AUX] soft key. Parameter numbers are displayed instead of the setting made for each item.



Fig. 9.18 (c) Parameter Number Display

(2) Press the [AUX] soft key again.

The screen display returns to the original screen.

Setting of parameters of axis type on an axis-by-axis basis

When an item of a parameter of axis type is specified on the adjustment screen, the same value is set for all axes except an axis specified by bit 7 (NFF) of parameter No. 1009. The display indicates a setting for the first axis.

When a different value is to be set for each axis, enter data on the parameter screen. (A parameter number check can be made using the [AUX] soft key.) If a different value is set for each axis, an asterisk (*) is prefixed to the numeric value indication of each item.

	FINE	MEDIUM	ROUGH
FINISH LEVEL	1	15	96
ACC/BEC LEVEL	100	100	120 (%)
ACC FOR BIPL	1000.000 *	1200.000 *	2500.000 (MM/SEC/SEC)
ACC CHANGE TIME (BELL)	20	30	100 (MSEC)
MAX ACCELERATION	500.000 *	500.000 *	1000.000 (MM/SEC/SEC)
T-CONST AIPL ACC/BEC	15	15	15 (MSEC)
CORNER FEED	192.000 *	192.000 *	375.000 (MM/MIN)
FEED FORWARD COEFFICIENT	97.50	97.50	97.50 (%)

Fig. 9.18 (d) Example of Setting on an Axis-by-Axis Basis

Cautions

NOTE

- 1 Settings made in automatic setting or automatic adjustment are only guideline values. The actual error may differ from that expected due to causes such as a servo system delay.
- 2 If the gear reduction ratio of a rotation axis is very large, an overflow may occur in feed-forward calculation by the servo software, resulting in an alarm (rotation axis digital servo parameter error) immediately after automatic adjustment. In such a case, set bit 4 of parameter No. 1804 for the axis to 1.
- 3 If vibration occurs on an axis after automatic setting, set bit 4 of parameter No. 1884 for the axis to 0. In a very rare case, vibration may occur as a result of the machine characteristics when bit 4 of parameter No. 1884 is set to 1.
If parameter No. 1518 is set to specify a lower feed-forward coefficient limit in automatic adjustment, the lower limit value is always used when a finish above a certain level is specified. For this reason, values such as a time constant after interpolation calculated using the feed-forward coefficient become identical. Thus, automatic adjustment can produce the same results even though a different finish level is specified. In such a case, set parameter No. 1518 to 0, then make another automatic adjustment.

10

DISPLAY

10.1 PROGRAM DISPLAY

During operation in MEM or MDI mode, the program currently being executed can be displayed.

Displaying the contents of the program

On the program screen, the contents of the program currently being executed are displayed.

Follow the procedure described in Section III.x.x, "Programs" to display the program screen.

£ Program screen

Fig.10.1 (a) Program Screen

The cursor is positioned to the block currently executed.

Displaying a program list

On the program directory screen, a list of programs registered in the program memory appears.

Follow the procedure described in Section III.x.x, "Programs" to display the program directory screen.

The program directory screen contains a screen displaying a list of program numbers and another displaying a list of program numbers and names.

To switch between the two screens,   use the page keys.

£ Program directory screen (number display)

Fig.10.1 (b) Program Directory Screen (Number Display)

The numbers of the registered programs are listed.

At the bottom of the screen, the free space in program memory and the number of registerable programs are displayed.

£ **Program**
(number and name display)

directory

screen

**Fig.10.1 (c) Program Directory Screen
(Number and Name Display)**

The numbers and names of the registered programs are listed together with the program sizes.

At the bottom of the screen, the free space in program memory and the number of registerable programs are displayed.

Checking programs

The program check screen contains three screens.

Follow the procedure described in Section III.x.x, "Programs" to display the program check screen.

£ **Program check screen**

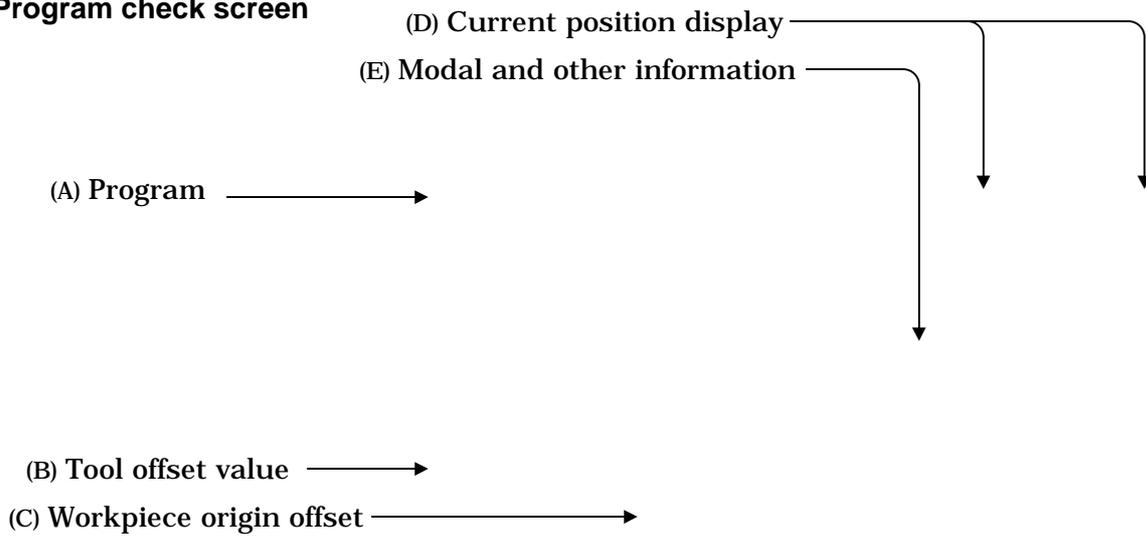


Fig.10.1 (d) Program Check Screen

On the program check screen, the following items of information, related to the block currently being executed, are displayed:

(A) Program

The currently executing block of the program is displayed, as are the preceding and succeeding blocks. The block currently being executed is displayed in reverse video.

(B) Tool offset value

The offset numbers and values for tool length compensation and cutter compensation are displayed. If the tool offset memory B/C options exist, the amounts of geometric compensation and wear compensation are displayed.

(C) Workpiece origin offset

The workpiece origin offset of the currently selected workpiece coordinate system is displayed.

(D) Current position

The current positions in the relative, absolute, and machine coordinate systems can be displayed, as well as the remaining amount of travel. To display the current positions in the relative and machine coordinate systems, press the [RELATIVE] soft key. To display the current positions in the absolute and machine coordinate systems, press the [ABSOLUTE] soft key. To display the current position in the machine coordinate system and the remaining amount of travel, press the [MACHINE] soft key.

If the number of controlled axes is seven or greater, use

the page keys to switch the displayed axes.



(E) Modal and other information

The following items of information are displayed:

- Modal G code
The modal G code in the block currently executed is displayed.
- Remaining number of subprogram calls (L)
The remaining number of subprogram calls is displayed.
- Specified F/actual F
The specified feedrate and the actual feedrate are displayed.
- Specified S/actual S
The specified spindle speed and the actual spindle speed are displayed.

By setting parameter NOS (bit 0 of No. 2204) to 1, the display is suppressed.

- Operating time/parts count
Two display items can be selected from among automatic operation time, cutting time, cycle time, general-purpose timer time, and machined parts count, with parameters Nos. 2240 and 2241.
- Additional information displayed when an NC window option exists
The M code (five types) currently being executed, spindle tool number, next machining tool number, and spindle speed range are displayed. Refer to the manual supplied by the machine tool builder for details.

£ Previous block screen

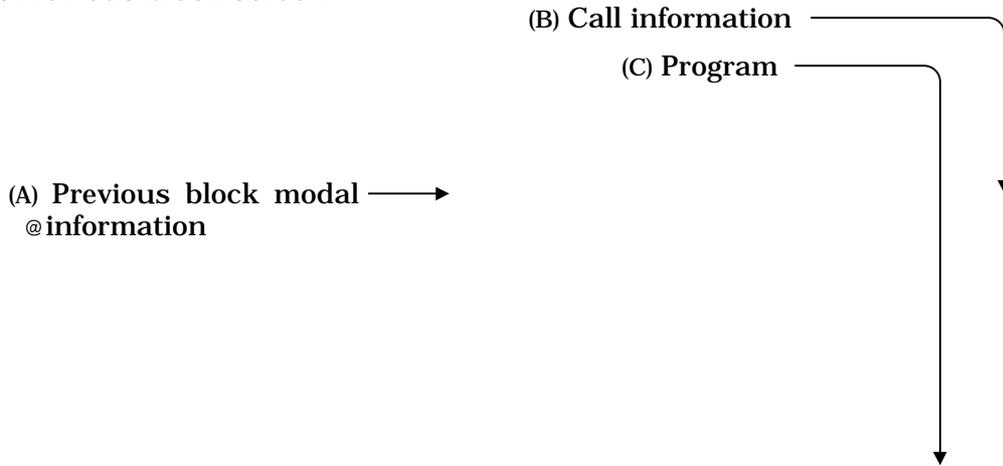


Fig. 10.1 (e) Previous Block Screen

On the previous block screen, the following items of information, related to the block executed last, are displayed:

- (A) Previous block modal information
The specified value in the block executed last (block preceding that currently being executed) is displayed, as well as the modal value in that block.
- (B) Call information
The numbers of the programs called with subprogram calls, macro calls, macro interrupts, and graphic copying are displayed sequentially, starting with that of the main program.
- (C) Program
The currently executing block of the program is displayed, as well as the preceding and succeeding blocks. The block currently being executed is displayed in reverse video.

£ Active block screen

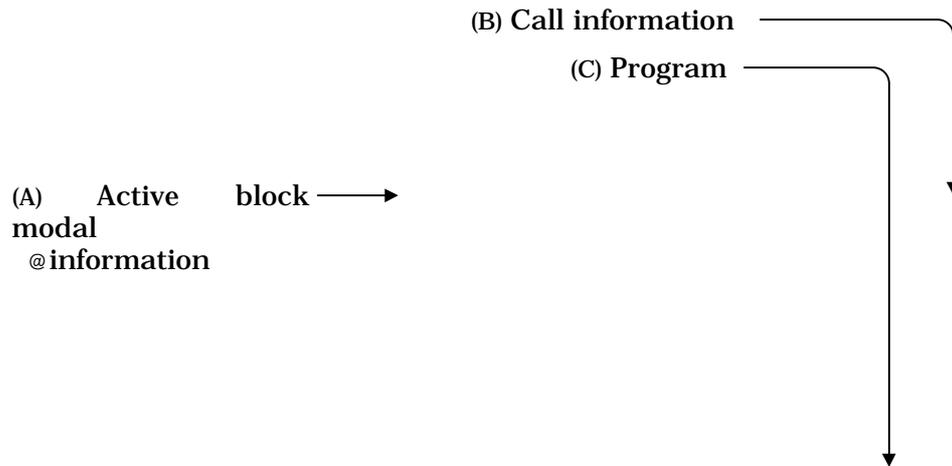


Fig. 10.1 (f) Active Block Screen

On the active block screen, the following items of information, related to the block currently being executed, are displayed:

- (A) Active block modal information
The specified value in the block currently being executed is displayed, as well as the modal value in that block.
- (B) Call information
The numbers of the programs called with subprogram calls, macro calls, macro interrupts, and graphic copying are displayed sequentially, starting with that of the main program.
- (C) Program
The currently executing block of the program is displayed, as well as the preceding and succeeding blocks. The block currently being executed is displayed in reverse video.

10.2 PROGRAM AND SEQUENCE NUMBER DISPLAY, STATUS DISPLAY, AND DATA SETTING AND INPUT/OUTPUT OPERATION WARNING DISPLAY

The program number, sequence number, and the current CNC status are constantly displayed on the screen (except immediately after the power is turned on, when the PMC screen is displayed, and when a system alarm has been issued).

If an error is found when an attempt is made to set data or perform an input/output operation, the CNC indicates the error and does not accept the attempt.

This section explains program and sequence number display, CNC status display, and data setting and input/output operation warning display.

10.3 PROGRAM AND SEQUENCE NUMBER DISPLAY

The program and sequence numbers of the program currently executing are displayed at the upper right corner of the screen.

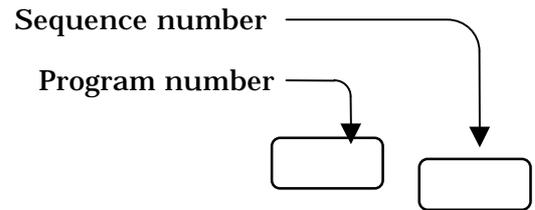


Fig. 10.3 (g) Program and Sequence Number Display

10.4 STATUS DISPLAY AND DATA SETTING AND INPUT/OUTPUT OPERATION WARNING DISPLAY

The currently selected mode, automatic operation status, alarm status, program editing status, and other information are displayed on the second line from the top of the screen.

This allows the user to determine the system status.

The current date and time is displayed on the first line.

If an error is found when an attempt is made to set data or perform an input/output operation, the CNC indicates the error as a warning at the bottom of the screen and does not accept the attempt. This prevents invalid data from being set and an invalid input/output operation from being performed.

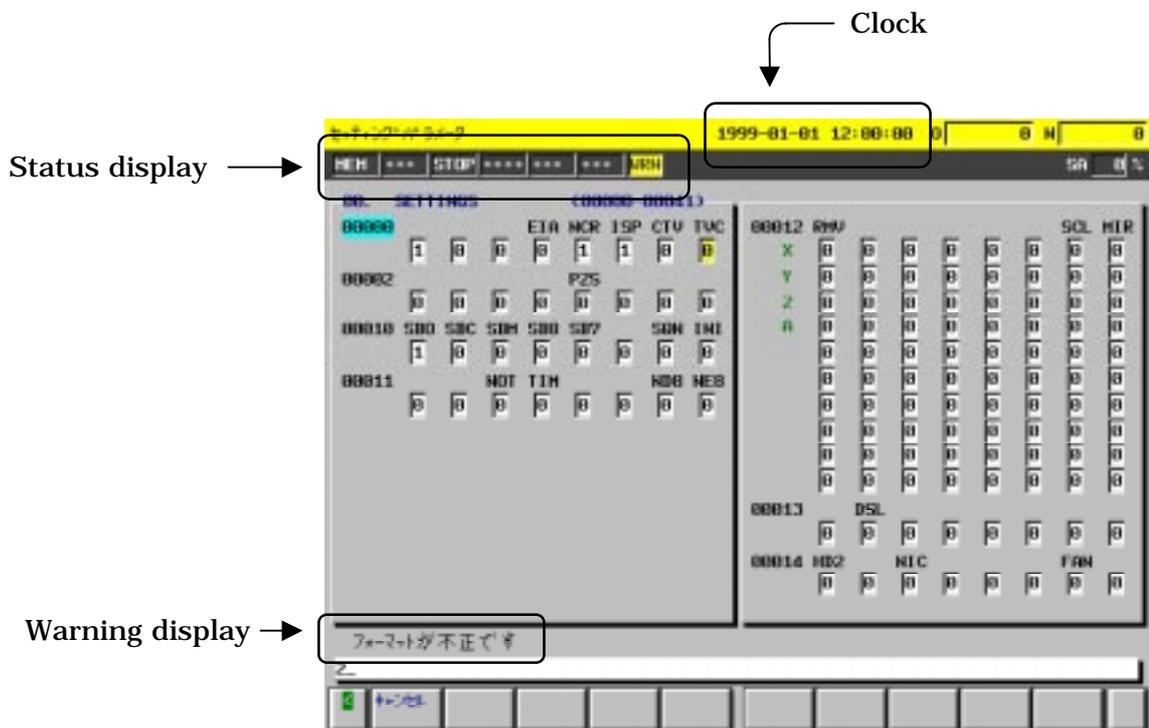


Fig.10.4 (h) Status/Clock/Warning Display

Status display

Display contents

1 Automatic operation mode selection

This field displays the automatic operation mode currently selected.

MDI : Operation
 MEM : Memory operation
 DNC : Operation
 EDIT : Memory operation
 **** : Automatic operation is not selected.

2 Manual operation mode selection

This field displays the manual operation mode currently selected.

JOG : Jog feed
 REF : Manual reference
 HND : Manual handle feed
 INC : Incremental feed
 AGJ : Manualfeed in spacified direction
 J+H : Jof and manual handle feed
 I+. : Incremetal and manual handle feed
 **** : Manual operation is not selected.

3 Automatic operation status

This field displays the status of automatic operation.

RSET : Reset is in progress.
 STOP : Automatic operation is stopped.
 HOLD : Automatic operation is suspended.
 STRT : Automatic operation is started.
 MSTR : Execution of a manual numeric command is started.
 NSRC : Sequence number search is in progress.

4 Program editing status

This field displays the status of program editing.

READ : Registration is in progress.
 PNCH : Punch-out is in progress.
 VRFY : Verification is in progress.
 SRCH : Search is in progress.
 COND : Memory arrangement is in progress.
 EDIT : An editing operation (such as insertion or alteration) is in progress.
 **** : No editing operation is in progress,

5 Axis movement/dwell/emergency stop status

This field indicates the axis movement, dwell, or emergency stop status.

MTN	: Movement along an axis is in progress.
DWL	: The machine is dwelling.
EMG	: An emergency stop is in progress.
***	: None of the above

6 M/S/T/B status

This field indicates whether a miscellaneous function such as M, S, T, or B is being performed.

FIN	: A miscellaneous function such as M, S, T, or B is being performed. (The system is waiting for a completion signal from the PMC.)
***	: A miscellaneous function such as M, S, T, or B is not being performed.

7 Alarm or label skip status

This field indicates the alarm, warning, battery, or label skip status.

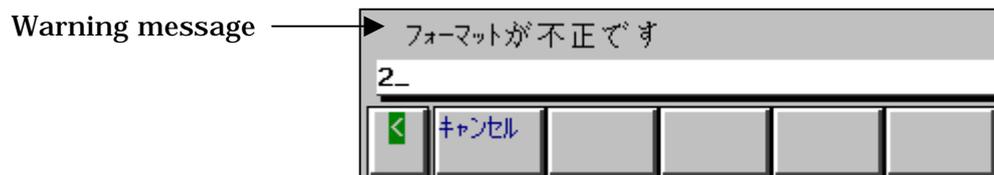
ALM	An alarm has occurred. (Blinking in reverse video)
WRN	A warning message is displayed. (Blinking in reverse video)
BAT	The battery is running down. (Blinking in reverse video)
LSK	Label skip status at the time of data input
Blank	None of the above

Clock

The clock displays the current date and time.

Data setting and input/output operation warning display

When an attempt is made to set data, but the keyed-in data contains an error (such as an invalid format or the valid range being executed) or it cannot be input (because of, for example, an invalid mode or write-protection), a warning message is displayed according to the cause of the error.



If this occurs, the CNC does not accept the attempt to set the data or perform the input/output operation. Remove the cause of the error, and retry the data setting or input/output operation.

10.5 CURRENT POSITION DISPLAY

Four current-position screens are used.

- Overall position display screen
- Current-position screen based on the relative coordinate system
- Current-position screen based on the workpiece coordinate system
- Current-position screen based on the machine coordinate system

On each screen, the actual feedrate and the actual spindle speed are also displayed, in addition to the current position.

Overall position display

Overall position display screen

On the overall position display screen, the current positions in the relative, workpiece, and machine coordinate systems are displayed, as well as the remaining amount of travel.

Fig.10.5 (i) Overall position display screen

Display contents

£ Current position

The current positions in the following coordinate systems are displayed simultaneously:

- RELATIV : Displays the current position in the relative coordinate system
- ABSOLUT : Displays the current position in the workpiece coordinate system.
- MACHINE : Displays the current position in the machine coordinate system.

£ Remaining amount of travel

The remaining amount of travel for the commands in a single block is displayed.

£ Actual feedrate

The actual feedrate and the actual spindle speed are displayed.
By setting parameter NOS (bit 0 of No. 2204) to 1, the display of the actual spindle speed is suppressed.

Display of the current position in the relative coordinate system

Relative-position display screen

On this screen, the current position in the relative coordinate system that is relative to the operator-specified coordinates is displayed.

Fig.10.5 (j) Relative-Position Display Screen

Display contents£ **Current position**

The current position in the relative coordinate system is displayed.

£ **Actual feedrate**

The actual feedrate and the actual spindle speed are displayed.

By setting parameter NOS (bit 0 of No. 2204) to 1, the display of the actual spindle speed is suppressed.

Display of the current position in the workpiece coordinate system

Absolute-position display screen

On this screen, the current (absolute) position in the workpiece coordinate system is displayed.

Fig.10.5 (k) Absolute-Position Display Screen

Display contents£ **Current position**

The current position in the workpiece coordinate system is displayed.

- **Actual feedrate**

The actual feedrate and the actual spindle speed are displayed.

By setting parameter NOS (bit 0 of No. 2204) to 1, the display of the actual spindle speed is suppressed.

Display of the current position in the machine coordinate system

Machine position display screen

The current position in the machine coordinate system is displayed.

Fig.10.5 (l) Machine Position Display Screen

Display contents

£ Current position

The current position in the machine coordinate system is displayed.

£ Actual feedrate

The actual feedrate and the actual spindle speed are displayed.

By setting parameter NOS (bit 0 of No. 2204) to 1, the display of the actual spindle speed is suppressed.

10.5.1 Presetting the current position

The current positions in the relative and workpiece coordinate systems can be preset.

The current position in the workpiece coordinate system can be preset using program command G92. For details of presetting the current position in the workpiece coordinate system, see Section II.x.x.x, "Setting workpiece coordinates (G92)."

Setting the current position to specified coordinates

To set the current position to specified coordinates in the relative coordinate system, use either of the following methods on the overall position display screen or the relative-position display screen. To set the current position to specified coordinates in the workpiece coordinate system, use either of the following methods on the absolute-position display screen.

£ ata protection key

To set the current position to specified coordinates in the workpiece coordinate system, enable the data protection key (KEY2).

£ Procedure

Method 1

(1) Press the [PRESET] soft key.
 (2) Press the [AXIS] soft key.
 (3) Key in the desired axis names and values.
 (4) Press the [EXEC] soft key.

Method 2

(1) Press the [PRESET] soft key.
 (2) Key in the desired axis names and values.
 (3) Press the [EXEC] soft key.

Example) X100.0Y200.0Z300.0 [EXEC]

This example sets the position to 100.0 on the X-axis, 200.0 on the Y-axis, and 300.0 on the Z-axis.

In the workpiece coordinate system, this example is equivalent to the following command executed by a program:
 G92X100.0Y200.0Z300.0;

The workpiece coordinate system is shifted so that the current position is located at the specified coordinates.

10.5.2 Presetting a workpiece coordinate system

A workpiece coordinate system that has been shifted because of, for example, manual intervention, can be preset to the previous position of the workpiece coordinate system, set off from the machine zero point by the workpiece origin offset through MDI operation. The offset from the previous position of the workpiece coordinate system on each axis can be specified.

Workpiece coordinate system preset can also be performed with the program command G92.1. For details of how to preset a workpiece

coordinate system, see Section II.x.x.x, "Presetting a workpiece coordinate system."

Presetting procedure

Use either of the following methods on the overall position display screen, relative-position display screen, or absolute-position display screen.

£ Mode

Place the system in the reset state or the automatic operation stopped state.

£ Presetting all axes

All axes of a workpiece coordinate system can be preset to the previous position of the workpiece coordinate system.

- (1) Press the [WORKPICE COORDINATE SYSTEM] soft key.
- (2) Press the [ALL AXIS] soft key.

£ Presetting a specific axis

A specific axis of a workpiece coordinate system can be preset to a position, offset by a specified value from the previous position of the workpiece coordinate system.

Method 1

- (2) Press the [WORKPICE COORD] soft key.
- (5) Press the [AXIS] soft key.
- (6) Key in the desired axis name (X, Y, Z, ...) and offset.
Multiple axes can be specified.
- (7) Press the [EXEC] soft key.

Method 2

- (4) Press the [WORKPICE COORD] soft key.
- (5) Key in the desired axis name (X, Y, Z, ...) and offset.
Multiple axes can be specified.
- (6) Press the [EXEC] soft key.

Examples)

XOY0Z0

[EXEC]

This example presets the X-, Y-, and Z-axes of a workpiece coordinate system to the previous position of the workpiece coordinate system.

(The offset from the previous position of the workpiece coordinate system is 0 for each axis.)

X100.0Y200.0Z200.0 [EXEC]

This example presets a workpiece coordinate system to a position, offset from the previous position of the workpiece coordinate system by 100.0 for the X-axis, 200.0 for the Y-axis, and 300.0 for the Z-axis.

The cutter compensation, tool length compensation, and tool offset are cancelled, and the origin of the currently selected workpiece coordinate system (G54 to G59) is preset to a position, offset from the origin of the machine coordinate system by the workpiece origin offset. As a result, the current position (absolute coordinates) in the workpiece coordinate system is equal to the machine coordinates minus the workpiece origin offset.

Whether presetting is to enabled in relative position display, as in absolute position display, can be specified with parameter DSE (bit 1 of No. 2202).

10.5.3 » ° u v Z b g

.....
(G92).....x.x.x.....(G92).....

» ° u **0** £

.....0.....0.....

E f [^ L [

.....0.....(KEY2).....

E S † **0** £

- (1) † "
- (2) S †

E ` £ † **0** £

- ..1
- (1) † "
- (2) †
- (3) ...(X,Y,Z...).....
-
- (4) s

- ..2
- (1) † "

(2) ...(X,Y,Z...).....

.....

(3) s

) XYZ s

XYZ.....0.....

.....
G92X0Y0Z0;

.....

» ° u w Ł W l Ł

E f [^ L [

.....

.....(KEY2).....

E Ł

..1

(1) ° ...>^.....

(2) †

(3)

(4) s

..2

(1) ° ...>^.....

(2)

(3) s

) X100.0Y200.0Z300.0 s
X.100.0.Y.200.0.Z.300.0.....

.....
G92X100.0Y200.0Z300.0;
.....

10.5.4 [N W n v Z b g

.....MDI.....
.....(G92.1).....X.X.X.....

v Z b g Ł

.....

E [h Ł

.....

E S † v Z b g

.....

(1) fi W n

(2) s t

E ` E t v Z b g

.....

..1

(2) fi W

(5) t

(6) ...(X,Y,Z...).....

.....

(7) s

..2

(4) fi W

(5) ...(X,Y,Z...).....

.....

(6) s

) X0Y0Z0 S
XYZ.....
.....0.

X100.0Y200.0Z200.0 S
.....X.100.0, Y.200.0, Z.300.0.....

.....(G54.G59).....
.....

.....DSE.No.2202#1.....

10.6 OPERATOR MESSAGE DISPLAY

Operator message display commands cause operator messages to be displayed on the operator message screen.

There are two types of operator messages:

1. External operator messages
2. Operator messages generated by custom macro commands

An operator message display command causes the system to automatically switch to the operator message screen. (There are cases where the system does not switch to the screen.)

To view operator messages, follow the procedure described in Section III.x.x.x, "Messages" to display the operator message screen.

£ Operator message screen

Fig. 10.6 (a) Operator Message Screen

£ Explanation of operator messages

For an explanation of external operator messages, refer to the manual supplied by the machine tool builder.

For an explanation of the operator messages generated by custom macro commands, refer to the manual supplied by the creator of the macros.

£ Clearing operator messages

External operator messages can be cleared with external commands. For an explanation of how to clear them, refer to the manual supplied by the machine tool builder.

When an operator message generated by a custom macro command is displayed, the CNC performs a single-block stop. At the start of the next cycle, the operator message generated by the custom macro command is cleared.

10.7 ALARM MESSAGE DISPLAY

When an alarm occurs, an alarm message is displayed on the alarm message screen.

There are three types of alarm messages:

1. Alarm messages generated by CNC self-diagnosis
2. External alarm messages
3. Alarm messages generated by custom macro commands

When an alarm occurs, the system automatically switches to the alarm message screen. (Note that there are some cases where the system does not switch the screen.)

To view alarm messages, follow the procedure described in Section III.x.x.x, "Messages" to display the alarm message screen.

£ Alarm message screen

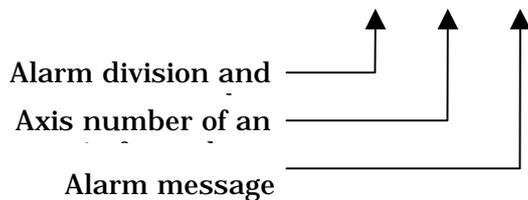


Fig.10.7 (a) Alarm Message Screen

An alarm message is displayed with its alarm division and number. If the alarm is an axis-form alarm, the axis name is also displayed. Up to 18 alarm messages can be displayed. Even if more alarms occur, the 19th and subsequent messages cannot be displayed.

£ Explanation of alarm messages

For an explanation of the alarm messages reported by CNC self-diagnosis, refer to the FANUC Series 15i/150i-MODEL A Maintenance Manual (B-63325JA).

For an explanation of external alarm messages, refer to the manual supplied by the machine tool builder.

For an explanation of the alarm messages generated by custom macro commands, refer to the manual supplied by the creator of the macros.

£ Clearing alarm messages

The alarm messages reported by CNC self-diagnosis and those generated by custom macro commands are cleared when the CNC is reset or when the [CANCEL] soft key is pressed. Note, however, that some of the alarm messages reported by CNC self-diagnosis cannot be cleared unless the cause of the alarm is removed.

External operator messages are cleared with external commands. For an explanation of how to clear them, refer to the manual supplied by the machine tool builder.

10.8 TOOL PATH DRAWING

The tool path of the program currently being used for machining can be drawn. This allows checking of the progress of machining and the tool's current position.

The following functions are supported:

- Display the current position of the tool in the workpiece coordinate system.
- Set any graphic coordinate system.
- Draw rapid traverse and cutting feed in different colors.
- Display the F, S, and T values of the program whose tool path is currently drawn, as well as the machining time.
- Enlarge/reduce the drawing.

Tool path drawing screen

Follow the procedure described in Section III.x.x, "Graphics" to display the tool path drawing screen.

£ Tool path drawing screen

The tool path drawing screen is divided into three portions:

- Drawing area in which a tool path is drawn.
- Portion in which machining information such as the tool position is displayed
- Portion in which the graphic coordinate system is displayed

Fig10.8 (a) Tool Path Drawing Screen

£ Tool path

In the graphic coordinate system, which is set with the appropriate graphic parameter, described later, the tool path in the workpiece coordinate system is drawn.

The position drawn depends on the setting of parameter DTL (bit 2 of No. 2202), as follows:

- If parameter DTL is set to 0
The position drawn is equal to the current position in the workpiece coordinate system plus the distance traveled because of tool length compensation and tool axial tool length compensation.
- If parameter DTL is set to 1
The position drawn is equal to the current position in the workpiece coordinate system, excluding the distance traveled because of the tool length compensation and tool axial tool length compensation. The position drawn is that of the tool tip, and matches the position specified by the program.

Even if the tool position changes discontinuously because of the setting of the origin or switching between workpiece coordinate systems, drawing is performed as if the tool moved.

Tool path drawing continues even after the system is switched to another screen. During drawing, "DRAW" is displayed in the drawing status display portion. This makes it possible to determine whether drawing is in progress even after the system is switched to another screen.

Fig10.8 (b) Drawing Status Display

£ Machining information

In the right-hand portion of the screen, the position in the workpiece coordinate system (on only those three axes on which drawing is being performed), feedrate (F), spindle speed (S), tool number (T), and machining time are displayed.

£ Graphic coordinate system

In the lower right portion of the screen, the coordinate axes of the graphic coordinate system are displayed together with the axis names. If the graphic coordinate system is not three-dimensional, the coordinate axis graduation is displayed in the tool path drawing area.

Graphic parameter screen

Follow the procedure described in Section III.x.x, "Graphics" to display the graphic parameter screen.

The parameters required to draw a tool path are specified on the graphic parameter screen. The graphic parameter screen contains two pages.

£ Page 1 of the graphic parameter screen

Fig10.8 (c) Page 1 of the Graphic Parameter Screen

On page 1 of the graphic parameter screen, set the graphic coordinate system, graphic colors, and graphic range.

In the setting of the graphic coordinate system, the coordinate axes of the coordinate system are graphically represented, together with the axis names. If the coordinate system is three-dimensional, it is displayed with the rotation angle.

The graphic range can be set in either of two ways: with the scale factor and the graphic center or with the graphic maximum and minimum.

£ Page 2 of the graphic parameter screen**Fig. }10.8 (d) Page 2 of the Graphic Parameter Screen**

On page 2 of the graphic parameter screen, set the graphic coordinate axes.

Graphic parameters

To draw a tool path, set the graphic coordinate system, tool path graphic colors, and graphic range on the graphic parameter screen.

The following explains the graphic parameters to be specified on the graphic parameter screen.

Merely specifying the graphic parameters does not make their settings effective. The [ERASE] soft key on the tool path drawing screen must be pressed to make them effective.

£ **Graphic coordinate system**

Select the graphic coordinate system in which to draw a tool path from among the following, and set its number.

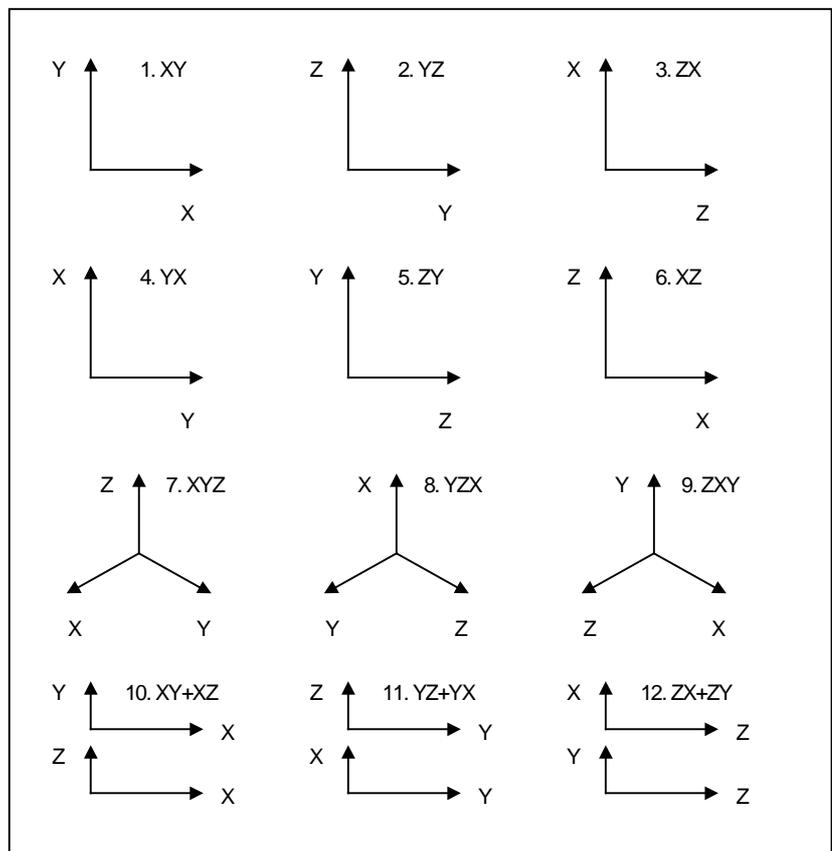


Fig. }10.8 (e) Graphic Coordinate Systems

£ **Horizontal rotation angle**

When the graphic coordinate system selected is a three-dimensional one, such as 7 (XYZ), 8 (YZX), or 9 (ZXY), the coordinate system can be rotated about a horizontal plane of rotation. The rotation angle can be set in the range of -360° to +360°.

The initial rotation angle can be set for parameter No. 4831. The actual rotation angle is equal to the initial rotation angle plus the specified rotation angle.

In Fig. 1.1 (e), shown below, graphic coordinate system XYZ is converted into XY''Z'' because of the following settings:

Initial rotation angle: 180°

Horizontal rotation angle: 30°

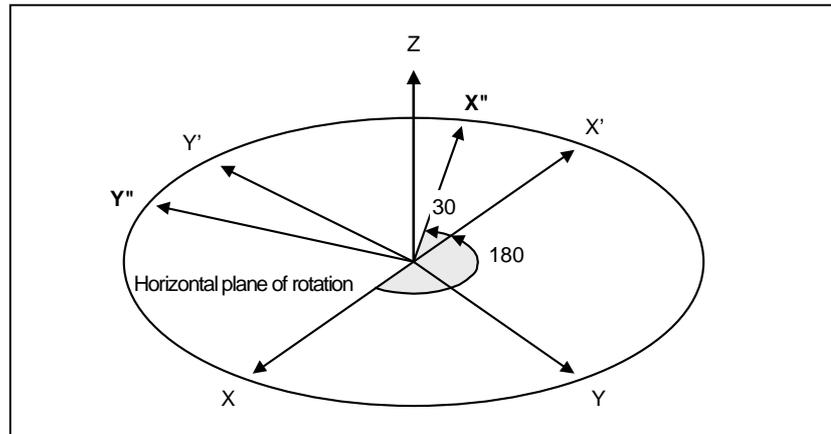


Fig.10.8 (f) Horizontal Rotation of a Coordinate System

£ Vertical rotation angle

When the selected graphic coordinate system is a three-dimensional one, such as 7 (XYZ), 8 (YZX), or 9 (ZXY), the coordinate system can be rotated about a specified vertical rotation axis on a horizontal plane. The rotation angle can be set in the range of -360° to $+360^\circ$.

The vertical rotation axis on a horizontal plane can be specified with the angle formed with the horizontal axis on the screen. The angle can be set for parameter No. 4832.

In Fig. 1.1 (f), shown below, graphic coordinate system XYZ is converted into X'Y'Z' because of the following settings:

Rotation angle of the vertical rotation axis: 65°

Vertical rotation angle: 20°

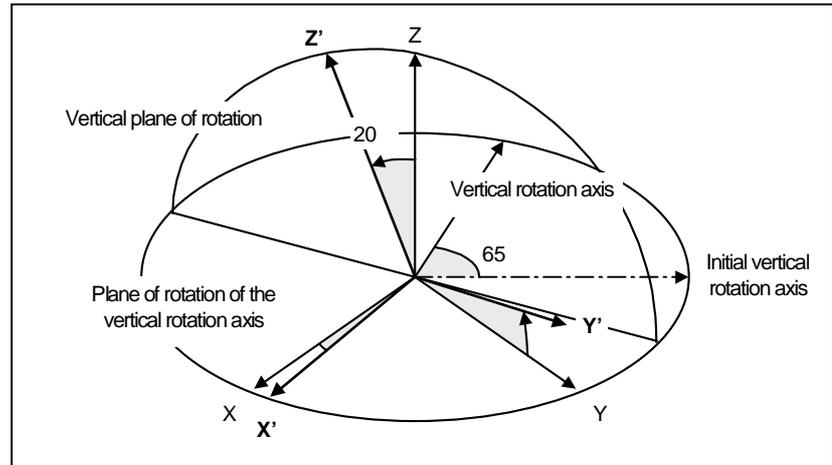


Fig. }10.8 (g) Vertical Rotation of a Coordinate System

£ Graphic colors

Set the number of the graphic color representing cutting feed, rapid traverse, and reverse in the tool path.

.....Red....Green....Yellow

..Blue....Purple....Light blue....White

£ Setting the graphic range

To draw a tool path within the tool path graphic area, set the graphic range. There are two ways of doing this:

1. Setting the graphic center and the scale factor
2. Setting the graphic range maximum and minimum

Whether 1 or 2 is effective depends on which parameter was specified last. Once set, the graphic range is retained even after the power is cut off.

1. Setting the graphic center and the scale factor

Set the coordinates of the center of the tool path graphic area, as coordinates in the workpiece coordinate system. Then, set the appropriate scale factor so that the graphic range is contained within the tool path graphic area.

The scale factor can be 0.01 to 100.

With a small scale factor, a tool path can be drawn within a wide range.

With a large scale factor, a tool path can be drawn in the vicinity of the graphic center being enlarged.

2. Setting the graphic range maximum and minimum

Specifying the graphic parameters

£ Moving the cursor

Use the   page keys and the    

cursor keys to position the cursor to the desired parameter.
It is not possible to move between pages with the cursor keys.

£ Inputting (absolute input)

settings

METHOD1

Press the [INPUT] soft key.
Enter a number you want to set.
Press the [EXEC] soft key.

METHOD2

Enter a number you want to set.
Press the [INPUT] soft key.

METHOD3

Enter a number you want to set.
Press the...  key.

For details of inputting data, see Section X.X.X, "Inputting Data."

£ Inputting (incremental input)

settings

METHOD1

(1) Press the [+INPUT] soft key.
(2)
(3) Press the [EXEC] soft key.

METHOD2

(1)
(2) Press the [+INPUT] soft key.

For details of inputting data, see Section X.X.X, "Inputting Data."

Executing tool path drawing

£ Starting drawing

- (1) Display the tool path drawing screen.
- (2) Press the [START] soft key. "DRAW" is displayed in the drawing status display portion, and the system enters that state in which tool movement during automatic or manual operation can be drawn. Subsequently, drawing continues even after another screen is displayed.
- (3) Start automatic or manual operation.

£ Ending drawing

- (1) Display the tool path drawing screen.
- (2) Press the [STOP] soft key. Tool path drawing ends, and "DRAW" is cleared from the drawing status display portion.

£ Erasing the drawn tool path

- (1) Press the [STOP] soft key to stop drawing.
- (2) Press the [ERASE] soft key. The drawn tool path is erased. When changing the settings of the graphic parameters, press [ERASE] to make the new settings effective.

NOTE

- 1 To perform drawing only, without moving the tool, place the system in the machine locked state.
- 2 If the feedrate is high, the tool path may not be drawn correctly. If this occurs, reduce the feedrate using an appropriate function such as dry run.

Graphic shift/enlarge

After a tool path has been drawn on the tool path drawing screen, it is possible to shift the center of drawing and enlarge part of the drawing. The shift and enlarge operations cannot be performed while a tool path is being drawn.

Graphic shift

The center of drawing can be shifted. In addition, the scale factor can be changed. Thus, the drawing can be enlarged/reduced, with any position as the center.

£ Procedure

- (1) Press the [SHIFT] soft key. A red cursor is displayed at the center of the screen, and the soft key display changes.
- (2) Use the [↑][↓][←][→] soft keys to move the red cursor to a new center of drawing.
- (3) To change the scale factor, set the relative scale factor. Key in a value within the range of 0.01 to 100 and press the [INPUT] soft key. The entered value is displayed in the "Relative" field located in the center right portion of the screen.
- (4) Press the [SET END] soft key to end the procedure. Then, press the [ERASE] soft key to make the graphic shift settings effective. Drawing is now possible with the new settings.

£ Relative scale factor

The relative scale factor is relative to the scale factor currently set for the appropriate graphic parameter.

For example, if a tool path that has been drawn is to be redrawn twice as large, the relative scale factor must be set to 2.00.

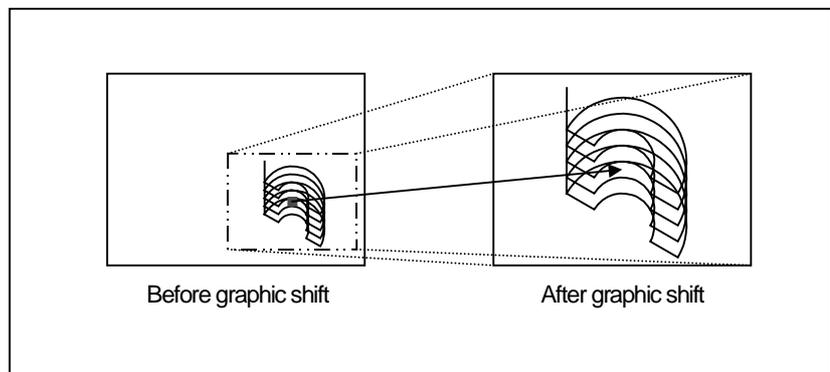


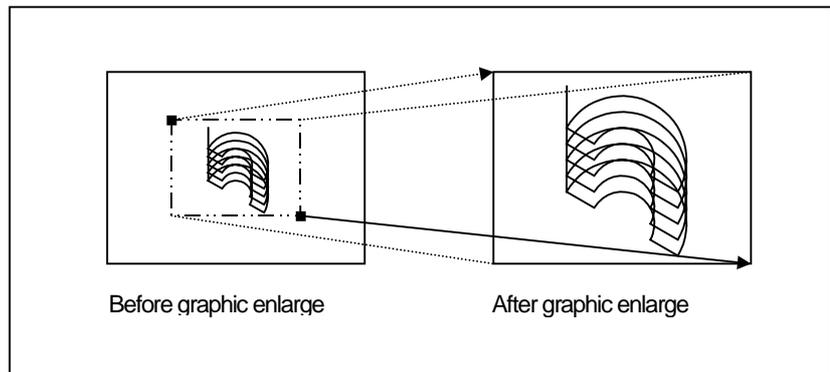
Fig.10.8 (h) Graphic Shift (Relative Scale Factor = 2.00)

Graphic enlarge

A specified rectangular area can be drawn enlarged.

£ Operation procedure

- (1) Press the [ENLARGE] soft key.
A red and a yellow cursor are displayed in the middle of the screen, and the soft key display changes.
- (2) Use the [↶] [↷] [↵] [↶] soft keys to move the two cursors to diagonal points of a new rectangular graphic range. Subsequently, tool path drawing is performed so that this rectangular range is contained within the graphic area.
- (3) Press the [SET END] soft key to end the procedure. Then, press the [ERASE] soft key to make the graphic enlarge settings effective. Drawing can now be performed with the new settings.



NOTES

1. To cancel the graphic shift/enlarge procedure, press the [CANCEL] soft key.
2. Even after graphic shift/enlarge has been set up, the previously drawn tool path will not be shifted or enlarged. The settings of graphic shift/enlarge take effect after the [ERASE] soft key is pressed.
3. The settings of graphic shift/enlarge are not automatically stored in the appropriate graphic parameters. For an explanation of the procedure for storing the settings in the graphic parameters, see the next section, "Storing and canceling the settings of graphic shift/enlarge."

Storing and canceling the settings of graphic shift/enlarge

The settings of graphic shift/enlarge can be stored in the appropriate graphic parameters. (The settings of graphic shift/enlarge are not automatically stored in the graphic parameters, but are cleared when the power to the CNC is turned off.)

It is possible to cancel the settings of graphic shift/enlarge to redraw the tool path with the previous settings.

£ AFTER SHIFT

Pressing the [AFTER SHIFT] soft key on the tool path drawing screen or the graphic parameter screen causes the settings of the graphic shift/enlarge to be stored in the appropriate graphic parameters.

£ BEFORE SHIFT

Pressing the [BEFORE SHIFT] soft key on the tool path drawing screen or the graphic parameter screen causes the settings of the graphic shift/enlarge to be canceled, making the current settings of the graphic parameters effective.

NOTE

When the graphic parameter screen is displayed after graphic shift/enlarge is set up, an @ may appear next to some settings. The @ indicates that the setting of the graphic parameter differs from the corresponding setting made for graphic shift/enlarge. When the settings of graphic shift/enlarge are stored or canceled, the @ disappears.

10.9 Operating monitor screen

The load value of the servo motor and the spindle motor is in real time displayed in the bar chart. Moreover, the load value of selected servo motor or only one spindle motor displays the shape of waves as a change situation of one minute.

An absolute position besides the loading meter and the override value, T code, the number of processing parts, and instruction speed/real speed/operation time are displayed on this screen.

Display

. Operating monitor " is selected according to the procedure of . -2.3 and . -2.5 and the operating screen is displayed.

}10.9 (a) Oprtating monitor screen

- 1 Absolute coordinates
It is the same display on the position display screen.
- 2 Sending speed/the number of main axis rotation/tool
An instruction speed/a real speed is the one displayed on the program check screen and is the same. The Override displays the state of the input signal. T code displays the instruction value.

10.DISPLAY

- . Loading meter
The load value of one spindle motor or less +3 servo motor is in real time displayed in the bar chart.
The spindle motor displays the one selected by the parameter (No.5850) or the main axis change function. The servo motor selects an arbitrary axis by the parameter (No.2231-2233).

- . Timer
It is the same display on the timer screen

- . Waveform display
The load value is displayed as a change situation of one minute though selected from among the motor displayed with the loading meter. The display is updated every minute.
The name of the selected motor is displayed on the left of a wavy display.

Cursor key/page key

- 1 By using cursor key   , the motor displayed waveform display can be selected from displayed with the loading meter.

- 2 By using pageing key   , the control axis can do the paging of the control axis displayed in the absolute value position page key by six axes in the machine with more than seven axes.

10.10 DISPLAY AND SETTING OF THE PARTS COUNTS, OPERATING TIME, AND CLOCK

The total machined parts count, required parts count, and machined parts count are displayed.

Also displayed are the cumulative power-on time, cumulative automatic operation time, cumulative cutting time, cumulative cycle time, and cumulative time of the timer that starts and stops with input signals from the PMC, and that can be used for any purpose (referred to as the general-purpose timer in this manual). For items other than the total machined parts count, cumulative power-on time, and cumulative cycle time, the operator can set the desired values through MDI operation.

The clock is always displayed at the top of the screen. It allows the setting of the date and time.

Displaying the parts counts/operating time/clock screen

Follow the procedure described in Section III.x.x, "Setting the Offset" to display the parts counts/operating time/clock screen.

£ Parts counts/operating time/clock screen

Fig.10.10 (a) Parts

Fig.10.10 (a) counts/Operating Time/Clock Screen

£(A) Parts counts**Total machined parts count**

A cumulative value, which is incremented by 1 when M02, M30, or the M code specified for parameter No. 4026 is executed. The value cannot be set from this screen. It can be set for parameter No. 508.

Required parts count

Set the number of machined parts that are required. "0" is regarded as being an infinite number of parts. The number can also be set for parameter No. 509.

Machined parts count

A cumulative value, which is incremented by 1 when M02, M30, or the M code specified for parameter No. 4026 is executed. The value can also be set for parameter No. 597. The count may be set so that it is reset when it reaches the required parts count. Refer to the manual supplied by the machine tool builder.

£(B) Operating time**Power-on time**

The cumulative power-on time is displayed. The time cannot be set from this screen. It can be set for parameter No. 500.

Automatic operation time

Cumulative automatic operation time (excluding the time when automatic operation is stopped or paused). The time can be set from this screen. It can also be set for parameters Nos. 501 and 502.

Cutting time

Cumulative time of cutting feed due to linear interpolation (G01), circular interpolation (G02, G03), and so on. The time can also be set for parameters Nos. 503 and 504.

Cycle time

Cumulative automatic operation time (excluding the time when automatic operation is stopped or paused), which is preset at the start of a cycle from a reset state. The time can be set from this screen. It is preset to 0 when the power is cut off.

General-purpose timer time
 Cumulative time of the general-purpose timer that starts and stops with input signals from the PMC. The time can be set from this screen. It can also be set for parameters Nos. 505 and 506. The timer can be used to show the cumulative coolant-on time, for example. Refer to the manual supplied by the machine tool builder for details.

£(C) Clock

Date/time
 The current date and time is displayed. The date and time can be set from this screen.

Usage

Counting the number of parts

When the M02 or M30 command is executed, the total machined parts count and the machined parts count are incremented by 1. Thus, by creating a program that executes M02 or M30 each time a single part is machined, the number of machined parts can be automatically counted. The counts are also incremented by executing the M code set for parameter No. 4026. Refer to the manual supplied by the machine tool builder for details.

Valid ranges of the operating time and the clock

Each value to be set must not be negative or exceed the maximum set for it.

Item	Maximum
Year	2094
Month	12
Day	31

Item	Maximum
Hours	23
Minutes	59
Seconds	59

Custom macro variables

Using the system variables of custom macros, the machined parts count, the required parts count, and the date and time can be obtained and changed.

Variable number	Description
#3901	Machined parts count
#3902	Required parts count
#3011	Date (year, month, and day)
#3012	Time (hours, minutes, and seconds)

(Example) If the date and time is 20 May, 1999, 4:17:05 p.m.,

#3011 = 19990520

#3012 = 161705

10.11 BACKGROUND DRAWING

Background drawing allows the tool path of a program to be drawn without machining. It also enables the tool path of another program to be drawn during machining.

Background drawing is faster than ordinary tool path drawing, thereby allowing the program to be checked promptly.

Background drawing screen

Follow the procedure described in Section III.x.x, "Graphics" to display the background drawing screen.

£ Background drawing screen

Background drawing screen is divided into three portions:

- Drawing area in which a tool path is drawn.
- Portion in which machining information such as the tool position is displayed
- Portion in which the graphic coordinate system is displayed

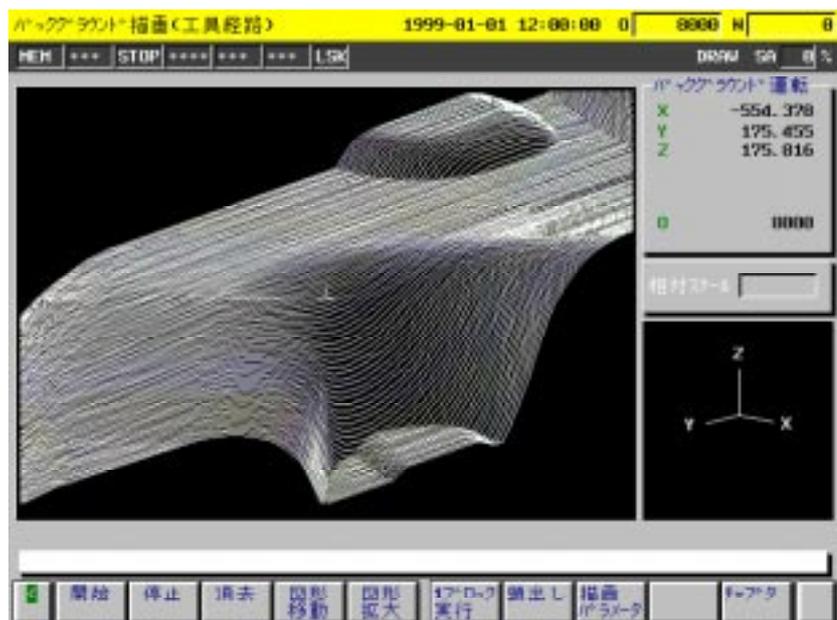


Fig.10.11 (a) Background drawing screen

£ Tool path

In the graphic coordinate system, which is set with the appropriate graphic parameter, described later, the tool path in the workpiece or machine coordinate system is drawn.

Use parameter No. 2209 to select either the workpiece or machine coordinate system.

Tool path drawing continues even after the system is switched to another screen. During drawing, "DRAW" is displayed in the drawing status display portion. This makes it possible to determine whether drawing is in progress even after the system is switched to another screen.

Fig.10.11 (b) Drawing Status Display

£ Machining information

In the right-hand part of the screen, the current position of the tool (on only the three axes on which drawing is being performed) and the number of the program currently being executed are displayed.

£ Graphic coordinate system

In the lower right portion of the screen, the coordinate axes of the graphic coordinate system are displayed together with the axis names. If the graphic coordinate system is not three-dimensional, the coordinate axis graduation is displayed in the tool path drawing area.

Graphic parameter screen

Follow the procedure described in Section III.x.x, "Graphics" to display the graphic parameter screen.

The parameters required to draw a tool path are specified on the graphic parameter screen. The graphic parameter screen contains two pages.

£ Page 1 of the graphic parameter screen



Fig. 10.11 (c) Page 1 of the Graphic Parameter Screen

On page 1 of the graphic parameter screen, set the graphic coordinate system, graphic colors, graphic range, number of the program for which background drawing is to be performed, and other information.

In the setting of the graphic coordinate system, the coordinate axes of the coordinate system are graphically represented, together with the axis names. If the coordinate system is three-dimensional, it is displayed with the rotation angle.

The graphic range can be set in either of two ways: with the scale factor and the graphic center or with the graphic maximum and minimum.

£ Page 2 of the graphic parameter screen



Fig.10.11 (d) Page 2 of the Graphic Parameter Screen

On page 2 of the graphic parameter screen, set the graphic coordinate axes.

Graphic parameters

To draw a tool path, set the graphic coordinate system, tool path graphic colors, and graphic range on the graphic parameter screen.

The following explains the graphic parameters to be specified on the graphic parameter screen.

Merely specifying the graphic parameters does not make their settings effective. The [ERASE] soft key on the tool path drawing screen must be pressed to make them effective.

£ **Graphic coordinate system**

Select the graphic coordinate system in which to draw a tool path from among the following, and set its number.

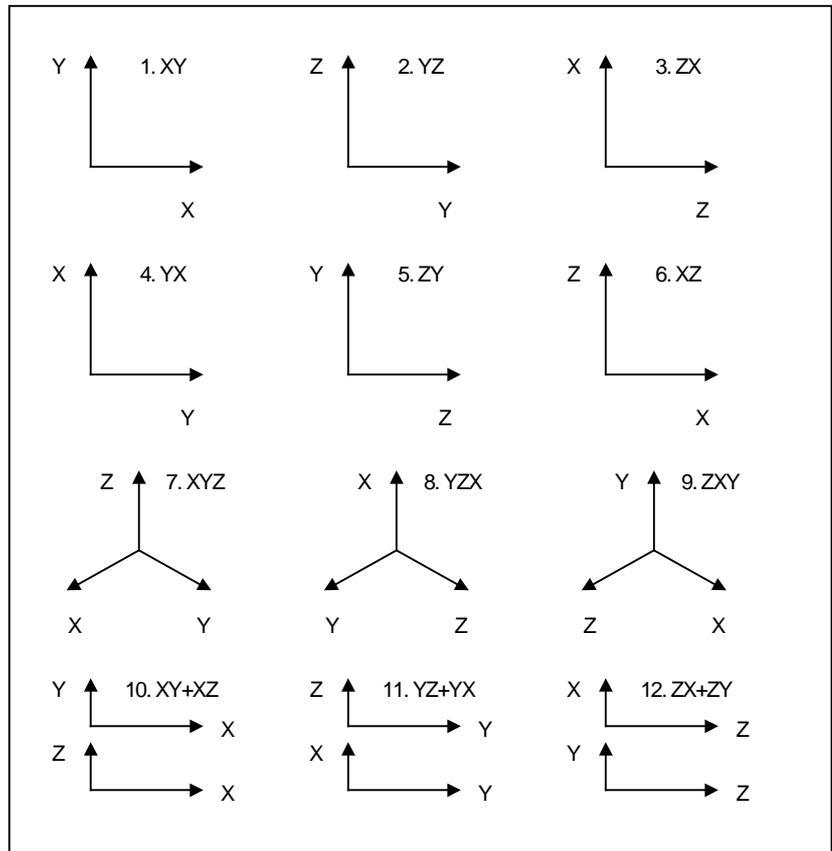


Fig.10.11 (e) Graphic Coordinate Systems

£ Horizontal rotation angle

When the graphic coordinate system selected is a three-dimensional one, such as 7 (XYZ), 8 (YZX), or 9 (ZXY), the coordinate system can be rotated about a horizontal plane of rotation. The rotation angle can be set in the range of -360° to $+360^\circ$.

The initial rotation angle can be set for parameter No. 4831. The actual rotation angle is equal to the initial rotation angle plus the specified rotation angle.

In Fig. 1.1 (e), shown below, graphic coordinate system XYZ is converted into $XY''Z''$ because of the following settings:

Initial rotation angle: 180°

Horizontal rotation angle: 30°

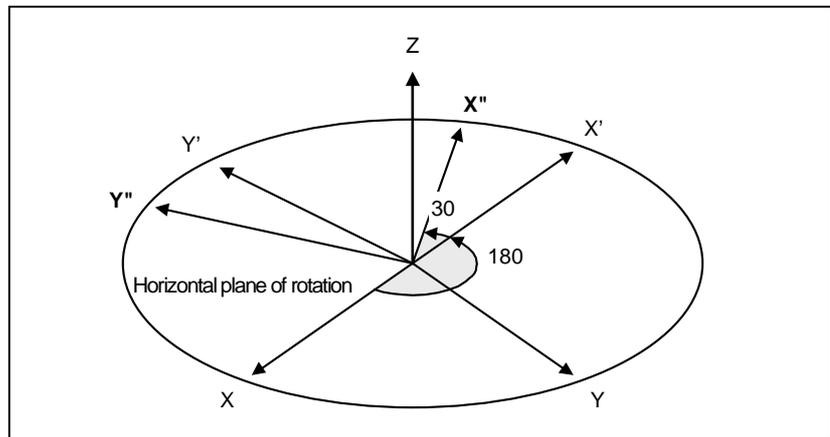


Fig.10.11 (f) Horizontal Rotation of a Coordinate System

£ Vertical rotation angle

When the selected graphic coordinate system is a three-dimensional one, such as 7 (XYZ), 8 (YZX), or 9 (ZXY), the coordinate system can be rotated about a specified vertical rotation axis on a horizontal plane. The rotation angle can be set in the range of -360° to $+360^{\circ}$.

The vertical rotation axis on a horizontal plane can be specified with the angle formed with the horizontal axis on the screen. The angle can be set for parameter No. 4832.

In Fig. 1.1 (f), shown below, graphic coordinate system XYZ is converted into X'Y'Z' because of the following settings:

Rotation angle of the vertical rotation axis: 65°

Vertical rotation angle: 20°

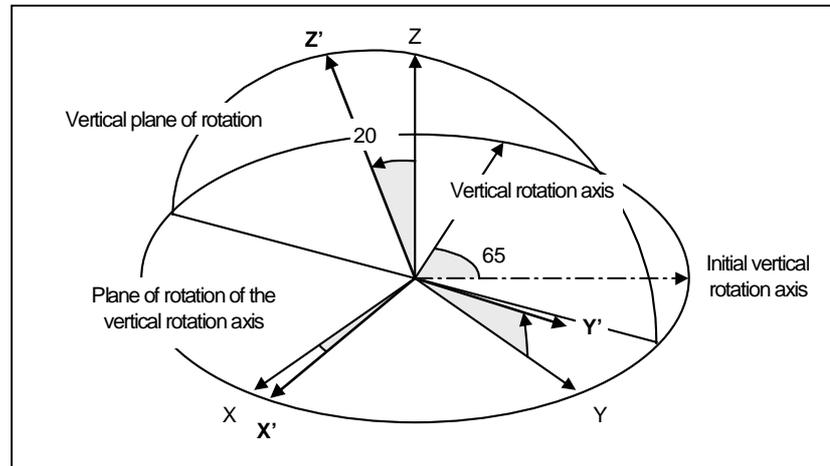


Fig.10.11 (g) Vertical Rotation of a Coordinate System

£ Graphic colors

Set the number of the graphic color representing cutting feed, rapid traverse, and reverse in the tool path.

.....Red.....Green.....Yellow

..Blue.....Purple.....Light blue.....White

£ Setting the graphic range

To draw a tool path within the tool path graphic area, set the graphic range. There are two ways of doing this:

1. Setting the graphic center and the scale factor
2. Setting the graphic range maximum and minimum

Whether 1 or 2 is effective depends on which parameter was specified last. Once set, the graphic range is retained even after the power is cut off.

1. Setting the graphic center and the scale factor

Set the coordinates of the center of the tool path graphic area, as coordinates in the workpiece coordinate system. Then, set the appropriate scale factor so that the graphic range is contained within the tool path graphic area.

The scale factor can be 0.01 to 100.

With a small scale factor, a tool path can be drawn within a wide range.

With a large scale factor, a tool path can be drawn in the vicinity of the graphic center being enlarged.

2. Setting the graphic range maximum and minimum

Set the maximum and minimum for the desired graphic range, as coordinates in the workpiece coordinate system. Drawing is performed so that the entire specified range is contained within the tool path graphic area.

From the maximum and minimum, the graphic center and the scale factor are automatically calculated, and the graphic center and the scale factor displayed on the graphic parameter screen are updated.

The automatically calculated scale factor is clamped within the range of 0.01 to 100 if it exceeds the range. The maximum must be greater than the minimum.

If the graphic coordinate system selected is 10 (XY+XZ), 11 (YZ+YX), or 12 (ZX+ZY), the horizontal ratio of the upper and lower coordinate systems will be clamped within the range of 1:2 to 2:1.

Specifying the graphic parameters

£ **Moving the cursor**

Use the   page keys and the    

cursor keys to position the cursor to the desired parameter.
It is not possible to move between pages with the cursor keys.

£ **Inputting settings
(absolute input)**

Method 1
Press the [INPUT] soft key.
Enter a number you want to set.
Press the [EXEC] soft key.

Method 2
Enter a number you want to set.
Press the [INPUT] soft key.

Method 3
Enter a number you want to set.
Press the [INPUT] key.

For details of inputting data, see Section X.X.X, "Inputting Data."

- **Inputting settings
(incremental input)**

Method 1
(1) Press the [+ INPUT] soft key.
(2) Enter the value to be added to the current setting.
(3) Press the [EXEC] soft key.

Method 2
(1) Enter the value to be added to the current setting.
(2) Press the [+ INPUT] soft key.

For details of inputting data, see Section X.X.X, "Inputting Data."

Executing background drawing

£ Registering the program

Register the desired program in memory. Unless it is registered in memory, its tool path cannot be drawn. M02 or M30 must be specified at the end of the program.

£ Calling the program

Press the [REWIND] soft key to call the program whose tool path is to be drawn.

£ Starting drawing

Press the [START] soft key. This starts drawing.

If "STOP" is not displayed in the drawing status display portion located in the upper right corner of the screen, drawing starts from the top of the program. While drawing is in progress, "DRAW" is displayed in the drawing status display portion.

To resume drawing after it has stopped, press the [START] soft key.

£ Stopping drawing

To pause drawing, insert M00 into the program beforehand or press the [STOP] soft key. While drawing is stopped, "STOP" is displayed in the drawing status display portion.

To resume drawing, press the [START] soft key. To redraw the tool path from the beginning of the program, press the [REWIND] soft key, then the [START] soft key.

£ Ending drawing

Drawing ends when the end of the program (O2 or M30) is executed, and "DRAW" disappears.

£ Erasing the drawn tool path

Press the [ERASE] soft key. The drawn tool path is erased. When changing the settings of the graphic parameters, press [ERASE] to make the new settings effective.

Graphic shift/enlarge

After a tool path has been drawn on the tool path drawing screen, it is possible to shift the center of drawing and enlarge part of the drawing. The shift and enlarge operations cannot be performed while a tool path is being drawn.

Graphic shift

The center of drawing can be shifted. In addition, the scale factor can be changed. Thus, the drawing can be enlarged/reduced, with any position as the center.

£ Procedure

- (1) Press the [SHIFT] soft key. A red cursor is displayed at the center of the screen, and the soft key display changes.
- (2) Use the [] [] [] [] soft keys to move the red cursor to a new center of drawing.
- (3) To change the scale factor, set the relative scale factor. Key in a value within the range of 0.01 to 100 and press the [INPUT] soft key. The entered value is displayed in the "Relative" field located in the center right portion of the screen.
- (4) Press the [SET END] soft key to end the procedure. Then, press the [ERASE] soft key to make the graphic shift settings effective. Drawing is now possible with the new settings.

£ Relative scale factor

The relative scale factor is relative to the scale factor currently set for the appropriate graphic parameter.

For example, if a tool path that has been drawn is to be redrawn twice as large, the relative scale factor must be set to 2.00.

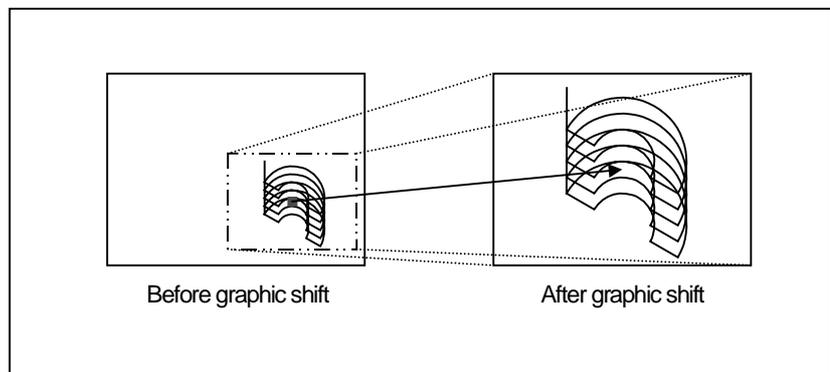


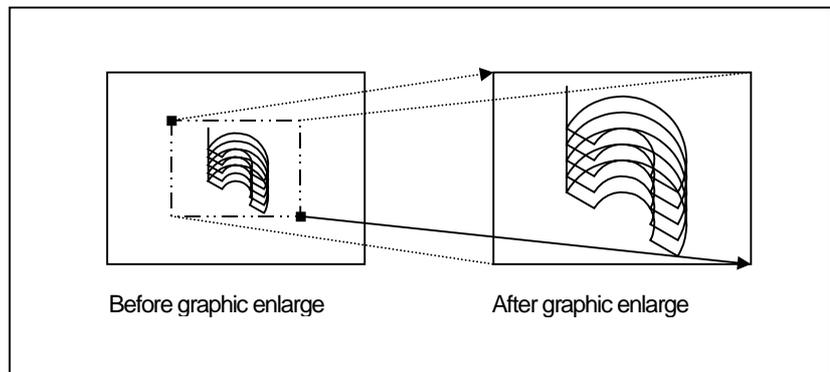
Fig.10.11 (h) Graphic Shift (Relative Scale Factor = 2.00)

Graphic enlarge

A specified rectangular area can be drawn enlarged.

£ Operation procedure

- (1) Press the [ENLARGE] soft key.
A red and a yellow cursor are displayed in the middle of the screen, and the soft key display changes.
- (2) Use the [↖] [↘] [↙] [↗] soft keys to move the two cursors to diagonal points of a new rectangular graphic range. Subsequently, tool path drawing is performed so that this rectangular range is contained within the graphic area.
- (3) Press the [SET END] soft key to end the procedure. Then, press the [ERASE] soft key to make the graphic enlarge settings effective. Drawing can now be performed with the new settings.



NOTES

1. To cancel the graphic shift/enlarge procedure, press the [CANCEL] soft key.
2. Even after graphic shift/enlarge has been set up, the previously drawn tool path will not be shifted or enlarged. The settings of graphic shift/enlarge take effect after the [ERASE] soft key is pressed.
3. The settings of graphic shift/enlarge are not automatically stored in the appropriate graphic parameters. For an explanation of the procedure for storing the settings in the graphic parameters, see the next section, "Storing and canceling the settings of graphic shift/enlarge."

Storing and canceling the settings of graphic shift/enlarge

The settings of graphic shift/enlarge can be stored in the appropriate graphic parameters. (The settings of graphic shift/enlarge are not automatically stored in the graphic parameters, but are cleared when the power to the CNC is turned off.)

It is possible to cancel the settings of graphic shift/enlarge to redraw the tool path with the previous settings.

⌘ AFTER SHIFT

Pressing the [AFTER SHIFT] soft key on the tool path drawing screen or the graphic parameter screen causes the settings of the graphic shift/enlarge to be stored in the appropriate graphic parameters.

⌘ BEFORE SHIFT

Pressing the [BEFORE SHIFT] soft key on the tool path drawing screen or the graphic parameter screen causes the settings of the graphic shift/enlarge to be canceled, making the current settings of the graphic parameters effective.

NOTE

When the graphic parameter screen is displayed after graphic shift/enlarge is set up, an @ may appear next to some settings. The @ indicates that the setting of the graphic parameter differs from the corresponding setting made for graphic shift/enlarge. When the settings of graphic shift/enlarge are stored or canceled, the @ disappears.

Data handled during background drawing

The following items of data are handled during background drawing:

1. Parameters

The same parameter settings as those used during ordinary automatic operation are used. During background drawing, parameters cannot be changed. If an attempt is made to change parameters with programmable parameter input (G10L50), a warning is displayed and drawing stops.

2. Tool offset value/workpiece origin offset/extended workpiece origin offset/macro variables/fixture offset data.

The data items used during background drawing differ from those used during automatic operation. When background drawing starts, a copy of the data for automatic operation is made to create the data for background drawing. Subsequently, the data for background drawing is handled separately from that used for automatic operation. If, therefore, the data for background drawing is changed with G10 or other codes, automatic operation will not be affected. Similarly, if the data for automatic operation is changed, background drawing will not be affected, either.

3. Tool life management data/tool number offset data/volumetric compensation data These items of data are not used during background drawing.

If an attempt is made to change the items with G10, a warning is displayed and drawing stops.

Functions that behave differently during background drawing

The following functions behave differently during background drawing:

1. Custom macros

- 1 Interface signals
#1000 to #1035 are always regarded as being 0.
- 2 Alarm message output
The alarm messages generated by #3000 are not displayed.
- 3 Message output
The messages generated by #3006 are not displayed but ignored.
- 4 Clock
#3001 and #3002 are ignored. Thus, if the following commands are specified, drawing cannot proceed any further.

```
#3001=1;
      WHILE      [#3001      LE      100]      DO1;
      END1;
```
- 5 Mirror image
#3007 is always regarded as being 0.
- 6 State in which the program is being resumed
#3008 is always regarded as being 0.
- 7 External output commands
DBRNT, DPRNT, POPEN, and PCLOS are ignored.

2. Ignored functions

- 1 G04 (dwell)
- 2 G20/G21 (inch/metric switching)
- 3 Miscellaneous functions (M, S, T, and B)
- 4 G22/G23 (stored stroke limit on/off)
- 5 G10.1 (PMC data setting)
- 6 G10.6 (Tool retreat/escape data setting)
- 7 G10.9 (programmable diameter/radius switching)
- 8 G81.1 (chopping)
- 9 G25/G26 (spindle fluctuation detection on/off)

3. Functions that partially behave differently

- 1 G28 (automatic reference position return)
Drawing is performed up to the mid point.
- 2 G29 (automatic return from the reference position)
Drawing is performed, starting from the mid point.
- 3 G27 (Reference position return check)
The reference position return check is not performed.

- 4 The stored stroke limit check is not performed
 - 5 G31 (skip function) and G31.1/G31.2/G31.3 (multistage skip function)
Drawing is performed up to the specified position, regardless of the skip signal.
 - 6 G60 (unidirectional positioning)
Drawing is always performed directly up to the specified position, even if the positioning direction is opposite.
4. Functions that behave differently
- 1 G02.2/G03.2 (involute interpolation)
Circular interpolation is performed if parameter GST (bit 1 of No. 2207) is set to 0. A warning is displayed and drawing stops if parameter GST is set to 1.
 - 2 G06.1 (spline interpolation)
Linear interpolation is performed if parameter GST (bit 1 of No. 2207) is set to 0. A warning is displayed and drawing stops if parameter GST is set to 1.
 - 3 G02.1/G03.1 (circular threading B)
Circular interpolation is performed. Drawing is not possible with a rotation axis.
 - 4 G02.3/G03.3 (exponential interpolation)
Linear interpolation is performed with a linear axis only.
 - 5 G07 (hypothetical axis interpolation)
Circular interpolation is performed.
 - 6 G07.1 (cylindrical interpolation)
Drawing is performed on a cylindrical expanded plane.
 - 7 G12.1 (polar coordinate interpolation)
Drawing is performed in the Cartesian coordinate system on a polar coordinate plane.

Functions that are not available during background drawing

If an attempt is made to execute the following functions, a warning is displayed and drawing stops:

M198 External subprogram call

M10 Programmable data input

NOTE

Parameters/tool life management data data/tool number offset data/volumetric compensation data cannot be changed with G10. The tool offset value/workpiece origin offset/extended workpiece origin offset/macro variables/fixture offset data can be changed. Only those items of data used for background drawing can be changed; those items of data used for ordinary automatic operation cannot be changed.

Functions available during background drawing

G00	Positioning
G01	Linear interpolation
G02/G03	Circular interpolation, Helical interpolation
G15/G16	Polar coordinate
G17/G18/G19	Plane selection
G33	Thread cutting (drawn as linear interpolation)
G38	Cutter compensation vector keeping
G39	Cutter compensation corner arc
G40/G41/G42	Cutter compensation/cancel
G40.1/G41.1/G42.2	Normal direction control
G43/G44/G49	Tool length compensation/cancel
G45/G46/G47/G48	Tool offset
G50/G51	Scaling/cancel
G50.1/G51.1	Programmable mirror image/cancel
G52	Local coordinate system
G53	Machine coordinate system selecting
G54.G59	Workpiece coordinate system selecting
G54.1	Extention orkpiece coordinate system selecting
G61	Exact stop mode
G62	Automatic corner override mode
G63	Tapping mode

G64	Cutting mode
G65	Macro call
G66	Macro call A
G66.1	Macro call B
G67	Macro call A/B cancel
G68/G69	Coordinate system rotation/cancel
G72.1/G72.2	Figure copy
G90/G91	Absoluy
G92	Changing the workpiece coordinate system
G92.1	Workpiece coordinate system preset
G93	Inverse time feed
G94	Feed per minute
G95	Feed per revolution
G96/G97	Constant surface speed control/cancel
M98	Sub program control
Arbtrary angle chopping, Corner R	
G73/G74/G76/G80/G81/G82/G83/G84/G85/G86/G87/G88/G89/ G98/G99	Canned cycle/cancel

Drawing start position

The drawing start position is determined as follows:

1. If G92, G52, or G92.1 is specified first
If G92, G52, G92.1 is followed by the first move command, the point specified with G92, G52, or G92.1 will be the drawing start position.
2. If G92, G52, or G92.1 is not specified first
If the first move command is not preceded by G92, G52, or G92.1, the end point of the first move command will be the drawing start position.

10.12 FLOPPY DIRECTORY SCREEN

The floppy directory screen provides functions for using the FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and FANUC PROGRAM FILE Mate connected to the serial interface (RS-232C or RS-422) and a variety of utilities.

- 1 Output data such as part programs and parameters to an external unit
- 2 Input data such as part programs and parameters from an external unit
- 3 List the files on an external unit
- 4 Delete files from an external unit

Display

Follow the procedure described in Section III.x.x, "Setting the Offset" to display the floppy directory screen.

Fig.10.12 (a) Floppy Directory Screen

£ Listing the files on a floppy

The files contained on an inserted floppy or FA card can be listed. Up to 12 files can be listed per page.

Number : File number
File name : File name
Size : File size (in bytes)

Volume: Volume number of a multivolume file

A multivolume file is a file stored on multiple floppies.
The volume numbers indicate the order of these floppies.

£ Page Select

Page information for the file list is displayed.

If the list cannot be displayed on a single page, "Prev Page" and "Next Page" are displayed to indicate that there are multiple pages.

£ DNC Select

The names of the files to be loaded during DNC operation are displayed.

£ Message

Messages are displayed as required.

£ In/Out Data

The types of data to be input and output are displayed.

£ Output File Name

The names of the files to be output at data output time are displayed.

Displaying a file list

- 1 Press the [DIR. VIEW] soft key.
A file list is displayed.
- 2 If the file list cannot be displayed on a single page, "Prev Page" and "Next Page" are displayed in Page Select to indicate that there are multiple pages. Press the
or   keys to display the specified page.
- 3 The display is updated when the [DIR. VIEW] soft key is pressed.
If the floppy or FA card is replaced with another, press the [DIR. VIEW] soft key again to update the display.

Setting input/output data

- 1) Press the [DATA SELECT] soft key.
The items of data that can be input/output are displayed on the soft keys. If the desired item is not displayed, press the [NEXT DATA] soft key to update the soft key display.

- 2) Press the soft key corresponding to the item to be input/output.

Soft key, indicating An item of input/output data	Data type
Program	NC program
Parameter	System parameter
Pitch error	Pitch error compensation data
Tool compensation	Tool offset data
Work offset	Workpiece origin offset data
Macro variable	Custom macro variable
System log	System log data
PERIODMAINTE1	Periodic maintenance data
PERIODMAINTE2	Item selection menu (machine system) data
Volumetric error	Volumetric compensation data
Number offset	Tool offset data with a tool number
Fixture offset	Fixture offset data
Dynamic compensation	Rotary head dynamic tool compensation data
System configuration	System configuration data

Setting the output file name

Entering the file name with the MDI keys

- 1) Press the [NAME INPUT] soft key.
File Name> is displayed in the key-in buffer.
- 2) Press the ["FILE NAME] soft key.
" is added to the key-in buffer.
- 3) Enter the file name with the MDI keys.
- 4) Press the [FILE NAME"] soft key.
" is added after the file name.
- 5) Press the [EXEC] soft key.
The output file name is set.

Selecting a file from a file list

- 1) Display a file list.
See "Displaying a file list," described earlier.

- 2) Position the cursor to the desired output file name.
- 3) Press the [NAME GET] soft key.
The output file name is set.

Searching for a file

- 1) Press the [NO. SEARCH] soft key.
NO. SEARCH is displayed in the key-in buffer.
- 2) Press the [(FILE#)] soft key.
N is added to the key-in buffer.
- 3) Enter a file number.
- 4) Press the [EXEC] soft key.
File search is executed, and the cursor is positioned to the file having the specified number.

Inputting/outputting data

Inputting data

- 1) Set input/output data.
See "Setting input/output data," described earlier.
- 2) On the file list, position the cursor to the desired input file.
See "Displaying a file list," described earlier.

3) Perform the necessary operation, according to the table below:

Input data	Input operation
Program	<p>To input all the programs in the file</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG READ] soft key. 3) Press the [ALL] soft key. <p>To input a single program as is</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG READ] soft key. 3) Press the [1 PROGRAM] soft key. 4) Press the [NEW] soft key. <p>To input a program with its program number changed</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG READ] soft key. 3) Press the [(PROGRAM#)] soft key. 4) Enter a program number. 5) Press the [EXEC] soft key.
Parameter	<ol style="list-style-type: none"> 1) Select MDI mode. 2) Place the system in an emergency stop state. 3) Press the [DATA READ] soft key. 4) Press the [EXEC] soft key.
Pitch error Tool compensation Work offset Macro variable Number offset Fixture offset Dynamic compensation	<ol style="list-style-type: none"> 1) Select MDI mode. 2) Press the [DATA READ] soft key. 3) Press the [EXEC] soft key.
Volumetric error PERIODMAINT1 PERIODMAINT2	<ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [DATA READ] soft key. 3) Press the [EXEC] soft key.
System log System configuration	The data cannot be input.

Outputting data

- 1) Set **input/output** data.
See "Setting input/output data," described earlier.
- 2) Set **the output file** name.
See "Setting the output file name," described earlier.

3) Perform the necessary operation, according to the table below:

Input/output data	Output operation
Program	<p>To output all programs</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [All] soft key. <p>To output the selected program</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [THIS PROGRAM] soft key. <p>To specify and output a single file</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [(PROGRAM#)] soft key. 4) Enter a program number. 5) Press the [EXEC] soft key. <p>To output multiple programs specified with a range</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [(PROGRAM#)] soft key. 4) Key in the number of the first program in the range. 5) Press the [(PROGRAM#)] soft key. 6) Key in the number of the last program in the range. 7) Press the [EXEC] soft key.
Parameter Pitch error Tool compensation Workpiece zero compensation Macro variable System log Number offset Fixture offset Dynamic compensation System configuration	<ol style="list-style-type: none"> 1) Select MDI mode. 2) Press the [DATA PUNCH] soft key. 3) Press the [EXEC] soft key.
Volumetric error PERIODMAINTE1 PERIODMAINTE2	<ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [EXEC] soft key.

Deleting a file

- 1) On the file list, position the cursor to the file to be deleted.
See "Displaying a file list," described earlier.
- 2) Press the DELETE FILE] soft key.
- 3) Press the [EXEC] soft key.
The file is deleted, and the file list is updated.

Setting a DNC operation file

- 1) On the file list, position the cursor to the file to be used during
DNC operation.
See "Displaying a file list," described earlier.
- 2) Press the [DNC SET] soft key.
The file name is set in DNC Select.

Communication setting screen

Pressing the [COM. SETING] soft key causes the system to switch to the communication setting screen. On the communication setting screen, it is possible to set the serial interface channel to be used on the floppy directory screen, as well as the communication conditions. See Section III.x.x, "Communication Setting Screen" for details.

10.13 MEMORY CARD SCREEN

The memory card screen provides functions for using a memory card as an external memory unit and a variety of utilities.

- 1 Output data such as part programs and parameters to an external unit
- 2 Input data such as part programs and parameters from external unit
- 3 List the files on a memory card
- 4 Delete files from a memory card
- 5 Format a memory card

Display

Follow the procedure described in Section III.x.x, "Setting the Offset" to display the memory card screen.

Fig.10.13 (a) Memory Card Screen

£ Listing the files on a memory card

The files contained on the inserted memory card can be listed. Up to 12 files can be listed per page.

Number : File number
File name : File name
Size : File size (in bytes)
Date : Date on which the file was created
Time : Time at which the file was created

£ Free size

The amount of free space on the inserted memory card is displayed (in bytes). If the size exceeds 99999999 bytes, ***** is displayed.

£ Page Select

Page information for the file list is displayed.
If the list cannot be displayed on a single page, "Prev Page" and "Next Page" are displayed to indicate that there are multiple pages.

£ Status

The status of the memory card slot is displayed.

£ Message

Messages are displayed as required.

£ In/Out Data

The types of data to be input and output are displayed.

£ Output File Name

The names of the files to be output at data output time are displayed.

Displaying a file list

1 Press the [DIR. VIEW] soft key.
A file list is displayed.

2 If the file list cannot be displayed on a single page, "Prev Page" and "Next Page" are displayed in Page Select to indicate that there are multiple pages. Press the



..or

keys to display the specified page.

3 The display is updated when the [DIR. VIEW] soft key is pressed.
If the memory card is replaced with another, press the [DIR. VIEW] soft key again to update the display.

Setting input/output data

1) Press the [DATA SELECT] soft key.
The items of data that can be input/output are displayed on the soft keys. If the desired item is not displayed, press the [NEXT DATA] soft key to update the soft key display.

2) Press the soft key corresponding to the item to be input/output.

Soft key, indicating an item of input/output data	Data type
Program	NC program
Parameter	System parameter
Pitch error	Pitch error compensation data
Tool compensation	Tool offset data
Work offset	Workpiece origin offset data
Macro variable	Custom macro variable
System log	System log data
PERIODMAINTE1	Periodic maintenance data
PERIODMAINTE2	Item selection menu (machine system) data
Volumetric error	Volumetric compensation data
Number offset	Tool offset data with a tool number
Fixture offset	Fixture offset data
Dynamic compensation	Rotary head dynamic tool compensation data
System configuration	System configuration data

Setting the output file name

Entering the file name with the MDI keys

- 1) Press the [NAME INPUT] soft key.
File Name> is displayed in the key-in buffer.
- 2) Press the ["FILE NAME] soft key.
" is added to the key-in buffer.
- 3) Enter the file name with the MDI keys.
- 4) Press the [File Name"] soft key.
" is added after the file name.
- 5) Press the [EXEC] soft key.
The output file name is set.

Selecting a file from a file list

- 1) Display a file list.
See "Displaying a file list," described earlier.

- 2) Position the cursor to the desired output file name.
- 3) Press the [NAME GET] soft key.
The output file name is set.

Searching for a file

- 1) Press the [NO. SEARCH] soft key.
NO. SEARCH is displayed in the key-in buffer.
- 2) Press the [(FILE#)] soft key.
N is added to the key-in buffer.
- 3) Enter a file number.
- 4) Press the [EXEC] soft key.
File search is executed, and the cursor is positioned to the file having the specified number.

Inputting/outputting data

Inputting data

- 1) Set input/output data.
See "Setting input/output data," described earlier.
- 2) On the file list, position the cursor to the desired input file.
See "Displaying a file list," described earlier.

3) Perform the necessary operation, according to the table below:

Input data	Input operation
Program	<p>To input all the programs in the file</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG READ] soft key. 3) Press the [ALL] soft key. <p>To input a single program as is</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG READ] soft key. 3) Press the [1_PRGRAM] soft key. 4) Press the [NEW] soft key. <p>To input a program with its program number changed</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG READ] soft key. 3) Press the [(PROG#)] soft key. 4) Enter a program number. 5) Press the [EXEC] soft key.
Parameter	<ol style="list-style-type: none"> 1) Select MDI mode. 2) Place the system in an emergency stop state. 3) Press the [DATA READ] soft key. 4) Press the [EXEC] soft key.
Pitch error Tool compensation Work offset Macro variable Number offset Fixture offset Dynamic compensation	<ol style="list-style-type: none"> 1) Select MDI mode. 2) Press the [DATA READ] soft key. 3) Press the [EXEC] soft key.
Volumetric error PERIODMAINTE1 PERIODMAINTE2	<ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [DATA READ] soft key. 3) Press the [EXEC] soft key.
System log System configuration	The data cannot be input.

Outputting data

- 1) Set **input/output** **data**.
See "Setting input/output data," described earlier.
- 2) Set **the** **output** **file** **name**.
See "Setting the output file name," described earlier.

3) Perform the necessary operation, according to the table below:

Input/output data	Output operation
Program	<p>To output all programs</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [ALL] soft key. <p>To output the selected program</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [THIS] soft key. <p>To specify and output a single file</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [(PROG#)] soft key. 4) Enter a program number. 5) Press the [EXEC] soft key. <p>To output multiple programs specified with a range</p> <ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [PROG PUNCH] soft key. 3) Press the [(PROG#)] soft key. 4) Key in the number of the first program in the range. 5) Press the [(PROG#)] soft key. 6) Key in the number of the last program in the range. 7) Press the [EXEC] soft key.
Parameter Pitch error Tool compensation Workpiece zero compensation Macro variable System log Number offset Fixture offset Dynamic compensation System configuration	<ol style="list-style-type: none"> 1) Select MDI mode. 2) Press the [DATA PUNCH] soft key. 3) Press the [EXEC] soft key.
Volumetric error PERIODMAINTE1 PERIODMAINTE2	<ol style="list-style-type: none"> 1) Select EDIT mode. 2) Press the [DATA PUNCH] soft key. 3) Press the [EXEC] soft key.

Deleting a file

- 1) On the file list, position the cursor to the file to be deleted.
See "Displaying a file list," described earlier.
- 2) Press the [DELETE FILE] soft key.
- 3) Press the [EXEC] soft key.
The file is deleted, and the file list is updated.

Formatting a memory card

- 1) Press the [FORMAT] soft key.
- 2) Press the [EXEC] soft key.
The memory card is formatted. While formatting is in progress, a message is displayed with the estimated end time. The estimated end time is only a rough guide. The actual end time is affected by the CNC status and the characteristics of the memory card.

11 CLEARING THE SCREEN

11.1 CLEARING THE SCREEN

If the same character is displayed at the same position on the screen for a long time, the life of the LCD will be shortened.

To prevent this from occurring, the CNC screen can be cleared. There are two ways to do this: using the screen clear function, executed by pressing a key, and automatic screen clear function, which automatically clears the screen if there has been no key activity for the time set for a parameter.

11.1.1 Screen clear function

11.1.2 Automatic screen clear function

.....(.).....NC.....NC.....

12 INPUTTING/OUTPUTTING DATA

Information written on floppy disks, NC tape, and memory cards can be loaded into the CNC, using the memory card slot or an external input/output unit such as the Handy File. Information can also be written to these media.

The following types of information can be input and output:

1. NC program
2. System parameter
3. Pitch error data
4. Tool offset data
5. Workpiece origin offset data
6. Custom macro variable
7. System log data
8. Periodic maintenance data
9. Item selection menu (machine system) data
10. Volumetric compensation data
11. Tool offset data with a tool number
12. Fixture offset data
13. Rotary head dynamic tool compensation data
14. System configuration data

Before an input/output unit can be used, the parameters related to input/output must be specified.

For an explanation of specifying the parameters, see Section III-2, "Operation Units."

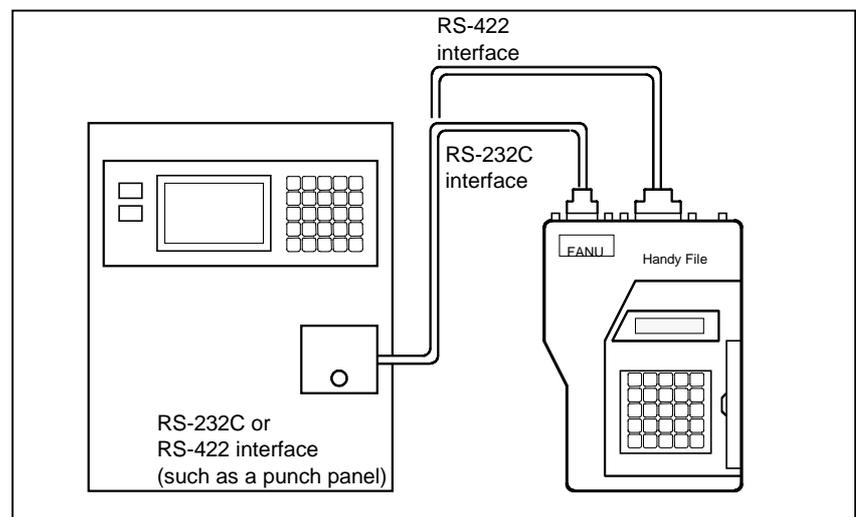


Fig.12 (a) Example of Connecting an External Input/Output Unit

(Connection of the Handy File)

12.1 FILES

Of the external input/output devices, the FANUC Handy File and FANUC Floppy

Cassette use floppy disks as their input/output medium, and the FANUC FA Card uses an FA card as its input/output medium.

In this manual, these input/output medium is generally referred to as a floppy.

However, when the description of one input/output medium varies from the description of another, the name of the input/output medium is used. In the text below, a floppy represents a floppy disk or FA card.

Unlike an NC tape, a floppy allows the user to freely choose from several types of data stored on one medium on a file-by-file basis. Input/output is possible with data extending over more than one floppy disk.

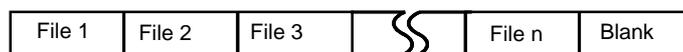
Explanations

- What is a File

The unit of data, which is input/output between the floppy and the CNC by one input/output operation (pressing the VREADW or VPUNCHW key), is called a HfileI.

When inputting CNC programs from, or outputting them to the floppy, for example, one or all programs within the CNC memory are handled as one file.

Files are assigned automatically file numbers 1,2,3,4 and so on, with the lead file as 1.

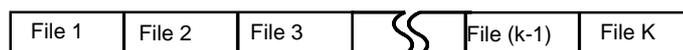


- Request for floppy replacement

When one file has been entered over two floppies, LEDs on the adaptor flash alternately on completion of data input/output between the first floppy and the CNC, prompting floppy replacement. In this case, take the first floppy out of the adaptor and insert a second floppy in its place. Then, data input/output will continue automatically.

Floppy replacement is prompted when the second floppy and later is required during file search-out, data input/output between the CNC and the floppy, or file deletion.

Floppy 1



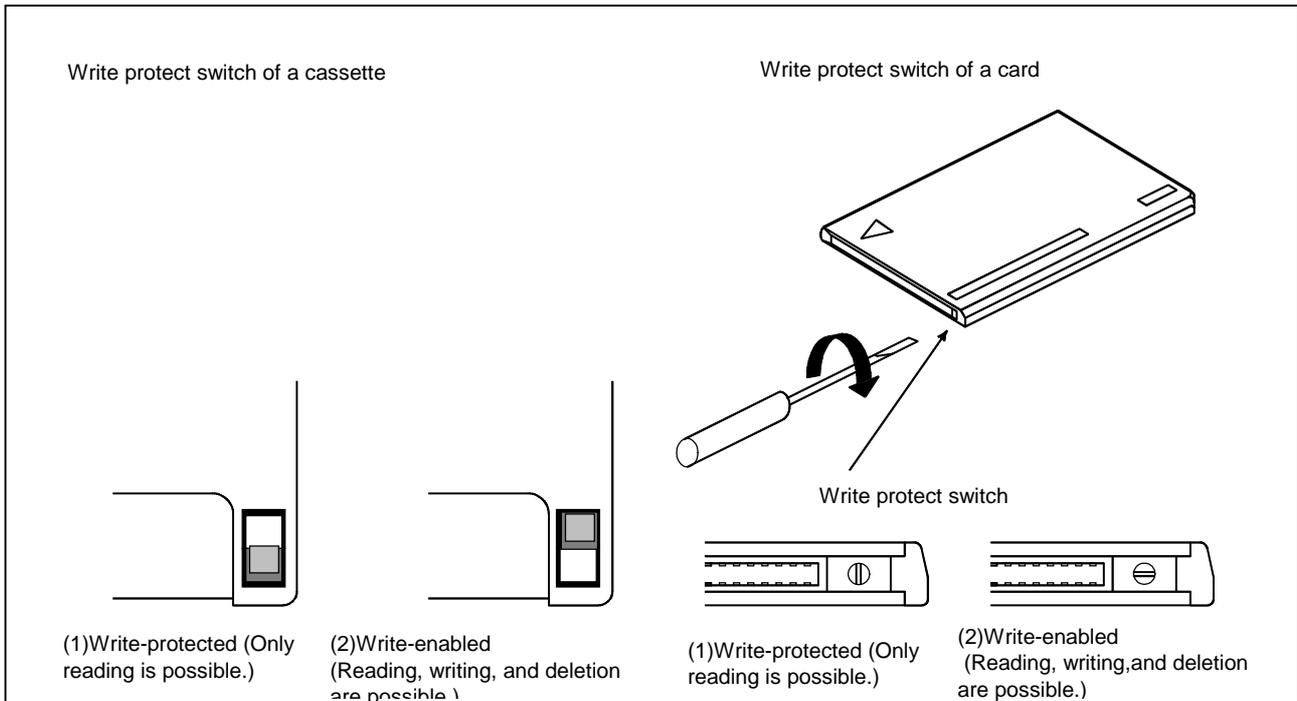
Floppy 2



Since floppy replacement is processed by the input/output device, no special operation is required. The CNC will interrupt data input/output operation until the next floppy is inserted into the adaptor. When reset operation is applied to the CNC during a request for floppy replacement, the CNC is not reset at once, but reset after the floppy has been replaced.

- Protect switch

The floppy is provided with the write protect switch. Set the switch to the write enable state. Then, start output operation.



i12.1 (a) Protect switch

- Writing memo

Once written in the cassette or card, data can subsequently be read out by correspondence between the data contents and file numbers. This correspondence are output to the CNC and displayed. The data contents can be displayed with display function for directory of floppy disk (See Section II-10.11 To display the contents, write the file numbers and the contents on the memo column which is the back of floppy.

(Entry example on MEMO)

File 1 C parameters

File 2 fset data

File 3 C program O0100

 : :

File (n-1)NC program O0500

File n C program O0600

12.2 SPECIFYING PARAMETERS REQUIRED FOR INPUT/OUTPUT

Follow the procedure described below to specify the parameters related to communication:

- (1) On the setting parameter screen, enable parameter writing.
- (2) On the communication setting screen, specify the parameters related to communication.

12.2.1 Setting parameter screen

Follow the procedure described below to enable parameter writing:

- (1) Place the system in MDI mode or in the emergency stop state.
- (2) Follow either of the procedures described below to display the setting parameter screen.

Method 1

Press the  function key several times, until the setting parameter screen appears.

Method 2

- (1) Press the  function key.
- (2) Click the [SETTINGPARAM] soft key.

Fig. 12.2.1 (b) Setting Parameter Screen

- (3) Set PWE (bit 0 of parameter No. 8000) and PRA (bit 2 of parameter No. 8000).

12.2.2 Communication setting screen

On this screen, specify the parameters related to communication.

Display

Follow either of the procedures described below to display the communication setting screen:

Method 1

Press the  function key several times, until the communication setting screen appears.

Method 2

(1) Press the  function key.

(2) Click the [COM. SETTING] soft key.

Fig. 12.2.2 (a) Communication Setting Screen

- BASIC SETTING

Specify the general items for the communication protocol. The table below lists the items, along with corresponding parameters.

Item name	Parameter	Option (parameter setting)
TV CHECK	TVC iNo.0000#0 j	ON (1)/OFF (0)
TV (COMMENT)	CTV iNo.0000#1 j	ON (0)/OFF (1)
IN/OUT CODE	ISP iNo.0000#2 j EIA iNo.0000#4 j	EIA (#2=0/1 ,#4 =1) ISO (#2=0 ,#4 =0) ASCII (#2=1 ,#4 =0)
EOB CODE	NCR iNo.0000#3 j	LF CR CR(0) ^LF(1)
F.G. INPUT F.G. OUTPUT B.G. INPUT B.G. OUTPUT	iNo.0020 j iNo.0021 j iNo.0022 j iNo.0023 j	RS232-C C1 (1) RS232-C C2 (2) RS232-C C3 (3) MEMCARD (8) RMTBUF (10) RS422 C1 (13) OPEN CNC1 (15) (DNC operational interface) OPEN CNC2 (16) (UP LOAD/DOWN LOAD interface)

- RS-232C, RS422 SETTING

Set the communication conditions for each of RS-232C and RS-422 channels. The device type, number of stop bits, and baud rate are specified using the following parameter sets and are allocated to channels for which device numbers are set using parameters Nos. 5001 to 5003 and 5013.

Parameter set 1: parameters Nos. 5110, 5111, and 5112

Parameter set 2: parameters Nos. 5120, 5121, and 5122

:

Parameter set 9: parameters Nos. 5190, 5191, and 5192

Item name	Parameter	Setting
DEVICE TYPE	(No.5110,5120,5130) (No.5140,5150,5160) (No.5170,5180,5190)	1 to 8
STOP BIT	(No.5111,5121,5131) (No.5141,5151,5161) (No.5171,5181,5191)	1/2
BAUD RATE	(No.5112,5122,5132) (No.5142,5152,5162) (No.5172,5182,5192)	1 to 12 (50, 100, 110, 150, 200, 300, 600, 1200, 2400, 4800,

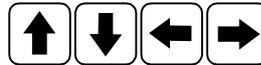
		9600, 19200bps)
--	--	-----------------

- [HINT]

Brief description of the item to which the cursor is positioned, such as the possible settings.

Data setting

1) Enter MDI mode.



2) Use the cursor keys,    , to position the cursor on the desired item.

3) Use the [SELECT +] and [SELECT -] soft keys to select the desired setting.

NOTE

1 For channels for which device numbers (parameters Nos. 5001 to 5003 and 5013) are not set, the following default parameter sets are selected:

RS232C CHANEL 1: Parameter set 1 (Nos. 5110, 5111, and 5112)

RS232C CHANEL 2: Parameter set 2 (Nos. 5120, 5121, and 5122)

RS323C CHANEL 3: Parameter set 3 (Nos. 5130, 5131, and 5132)

RS422 CHANEL 1: Parameter set 4 (Nos. 5140, 5141, and 5142)

2 If the values set for the STOP BIT and BAUD RATE parameters are out of range, the following default values are selected internally and displayed on the setting screen. These values are not, however, reflected in the parameters.

STOP BIT: 2

BAUD RATE: 9 (4800 bps)

3 If the value set for a DEVICE TYPE parameter is out of range, "UNKNOWN" appears.

12.3 INPUTTING AND OUTPUTTING PROGRAMS

12.3.1 Outputting Programs

The following explains the operation for outputting the programs registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select EDIT mode.
- (2) Follow either of the procedures described below to display the Program Text screen.

Method 1

Press the  function key several times, until the Program Text screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [TEXT] soft key.

Outputting the currently displayed program

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [THIS PROGRAM] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [THIS PROGRAM] soft key.

Method 3 (Specified with its file number (one of two methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [THIS PROGRAM] soft key.

Method 4 (Specified with its file number (one of two methods))

- (1) Press the [PUNCH] soft key.
- (2) Press  the address key.

- (3) Enter a file number.
- (4) Press the [THIS PROGRAM] soft key.

Outputting a specified single program

Neither the input file name nor number need be specified

Method 1

- (1) Press the [PUNCH] soft key.
- (2) Press the [(PROGRAM#)] soft key.
- (3) Enter a program number.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Press the [PUNCH] soft key.
- (2) Press  the address key.
- (3) Enter a program number.
- (4) Press the [EXEC] soft key.

Method 3

- (1) Press the  address key.
- (2) Enter a program number.
- (3) Press the [PUNCH] soft key.

- The output file is specified with its file name

Method 1

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [(PROGRAM#)] soft key.
- (6) Enter a program number.
- (7) Press the [EXEC] soft key.

Method 2

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.

- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the address key.
- (6) Enter a program number.
- (7) Press the [EXEC] soft key.

£ Specified with its file number

Method 1

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [(PROGRAM#)] soft key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 2

- (1) Press the [PUNCH] soft key.
- (2) Press the address key.
- (3) Enter a file number.
- (4) Press the address key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 3

- (1) Press the address key.
- (2) Enter a file number.
- (3) Press the address key.
- (4) Enter a program number.
- (5) Press the [PUNCH] soft key.

Outputting specified multiple programs

£ Neither the output file name nor number need be specified

Method 1

- (1) Press the [PUNCH] soft key.
- (2) Press the [(PROGRAM#)] soft key.
- (3) Enter the number of the first program to be output.
- (4) Press the [, (PROGRAM#)] soft key.
- (5) Enter the number of the last program to be output.
- (6) Press the [EXEC] soft key.

Method 2

- (1) Press the [PUNCH] soft key.
- (2) Press the address key.
- (3) Enter the number of the first program to be output.
- (4) Press the address key.
- (5) Press the address key.
- (6) Enter the number of the last program to be output.
- (7) Press the [EXEC] soft key.

Method 3

- (1) Press the address key.
- (2) Enter the number of the first program to be output.
- (3) Press the address key.
- (4) Press the address key.
- (5) Enter the number of the last program to be output.
- (6) Press the [PUNCH] soft key.

£ The output file is specified with its file name

Method 1

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [(PROGRAM#)] soft key.

- (6) Enter the number of the first program to be output.
- (7) Press the [, (PROGRAM#)] soft key.
- (8) Enter the number of the last program to be output.
- (9) Press the [EXEC] soft key.

Method 2

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.

(5) Press the address key.

(6) Enter the number of the first program to be output.

(7) Press the address key.

(8) Press the address key.

(9) Enter the number of the last program to be output.

(10) Press the [EXEC] soft key.

£ Specified with its file number

Method 1

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [(PROGRAM#)] soft key.
- (5) Enter the number of the first program to be output.
- (6) Press the [, (PROGRAM#)] soft key.
- (7) Enter the number of the last program to be output.
- (8) Press the [EXEC] soft key.

Method 2

(1) Press the [PUNCH] soft key.

(2) Press the address key.

(3) Enter a file number.

(4) Press the address key.

(5) Enter the number of the first program to be output.



(6) Press the address key.

(7) Press the address key.

(8) Enter the number of the last program to be output.

(9) Press the [EXEC] soft key.

Method 3

(1) Press the address key.

(2) Enter a file number.

(3) Press ...the address key.

(4) Enter the number of the first program to be output.

(5) Press ...the address key.

(6) Press ...the address key.

(7) Enter the number of the last program to be output.

(8) Press the [PUNCH] soft key.

Outputting all programs

Method 1 (Neither the output file name nor number need be specified)

(1) Press the [PUNCH] soft key.

(2) Press the [ALL] soft key.

Method 2 (The output file is specified with its file name)

(1) Press the [PUNCH] soft key.

(2) Press the ["FILE NAME] soft key.

(3) Enter a file name.

(4) Press the [FILE NAME"] soft key.

(5) Press the [ALL] soft key.

Method 3 (Specified with its file number (one of two methods))

(1) Press the [PUNCH] soft key.

(2) Press the [(FILE#)] soft key.

(3) Enter a file number.

(4) Press the [ALL] soft key.

Method 4 (Specified with its file number (two of two methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the address key.
- (3) Enter a file number.
- (4) Press the [ALL] soft key.

Outputting a portion of the currently displayed program

Method 1 (Neither the output file name nor number need be specified)

- (1) Position the cursor to the beginning of the portion you want to punch out.
- (2) Press the [PUNCH] soft key.
- (3) Press the [HERE] soft key.

Method 2 (The output file is specified with its file name)

- (1) Position the cursor to the beginning of the portion you want to punch out.
- (2) Press the [PUNCH] soft key.
- (3) Press the ["FILE NAME] soft key.
- (4) Enter a file name.
- (5) Press the [FILE NAME"] soft key.
- (6) Press the [HERE] soft key.

Method 3 (Specified with its file number (one of two methods))

- (1) Position the cursor to the beginning of the portion you want to punch out.
- (2) Press the [PUNCH] soft key.
- (3) Press the [(FILE#)] soft key.
- (4) Enter a file number.
- (5) Press the [HERE] soft key.

Method 4 (Specified with its file number (two of two methods))

- (1) Position the cursor to the beginning of the portion you want to punch out.
- (2) Press the [PUNCH] soft key.
- (3) Press the address key.
- (4) Enter a file number.
- (5) Press the [HERE] soft key.

12.3.2 Program Inputting

The following explains the operation for inputting the programs to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select EDIT mode.
- (2) Follow either of the procedures described below to display the Program Text screen.

Method 1

Press the  function key several times, until the Program Text screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [TEXT] soft key.

Registering a single program

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [1_PROGRAM] soft key.
- (3) Press the [NEW] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [1_PROGRAM] soft key.
- (6) Press the [NEW] soft key.

Method 3 (The input file is specified with its file number (one of two methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [1_PROGRAM] soft key.
- (5) Press the [NEW] soft key.

Method 4 (The input file is specified with its file number (two of two methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.

- (3) Enter a file number.
- (4) Press the [1_PROGRAM] soft key.
- (5) Press the [NEW] soft key.

Registering a program with a new program number

£ Methods in which neither the input file name nor number need be specified

Method 1

- (1) Press the [READ] soft key.
- (2) Press the [(PROGRAM#)] soft key.
- (3) Enter a program number.
- (4) Press the [EXEC] soft key.

Method 2

- (1) Press the [READ] soft key.
- (2) Press the address key.
- (3) Enter a program number.
- (4) Press the [EXEC] soft key.

Method 3

- (1) Press the address key.
- (2) Enter a program number.
- (3) Press the [READ] soft key.

£ Methods in which the input file is specified with its file name

Method 1

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [(PROGRAM#)] soft key.
- (6) Enter a program number.
- (7) Press the [EXEC] soft key.

Method 2

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the address key.

- (8) Enter a program number.
- (9) Press the [EXEC] soft key.

£ Methods in which the input file is specified with its file number

Method 1

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [(PROGRAM#)] soft key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 2

- (1) Press the [READ] soft key.
- (2) Press the address key.
- (3) Enter a file number.
- (4) Press the address key.
- (5) Enter a program number.
- (6) Press the [EXEC] soft key.

Method 3

- (1) Press the address key.
- (2) Enter a file number.
- (3) Press the address key.
- (4) Enter a program number.
- (5) Press the [READ] soft key.

Registering multiple NC programs

Method 1 (Neither the input file names nor numbers need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [ALL] soft key.

Method 2 (The input files are specified with their file names)

- (1) Press the [READ] soft key.

- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [ALL] soft key.

Method 3 (The input files are specified with their file numbers (one of two methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [ALL] soft key.

Method 4 (The input files are specified with their file numbers (two of two methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [ALL] soft key.

Registering additional programs

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [1_PROGRAM] soft key.
- (3) Press the [ADD] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [1_PROGRAM] soft key.
- (6) Press the [ADD] soft key.

Method 3 (The input file is specified with its file number (one of two methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [1_PROGRAM] soft key.
- (5) Press the [ADD] soft key.

Method 4 (The input file is specified with its file number (two of two methods))

- (1) Press the [READ] soft key.
- (2) Press the address key.
- (3) Enter a file number.
- (4) Press the [1_PROGRAM] soft key.
- (5) Press the [ADD] soft key.

12.4 INPUTTING AND OUTPUTTING SYSTEM PARAMETERS

12.4.1 Outputting System Parameters

The following explains the operation for outputting the system parameters registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Parameter screen.

Method 1

Press the  function key several times, until the Parameter screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [PARAMETER] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [PARAMETER] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [PARAMETER] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [PARAMETER] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press  the address key.

- (3) Enter a file number.
- (4) Press the [PARAMETER] soft key.

Method 5 (Specified with its file number (three of three methods))

- (2) Press the address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of system parameters," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be CNC-PARA.TXT.

12.4.2 Inputting System Parameters

The following explains the operation for inputting the system parameters to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Enter the emergency stop status.
- (3) Follow either of the procedures described below to display the Parameter screen.

Method 1

Press the  function key several times, until the Parameter screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [PARAMETER] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [PARAMETER] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [PARAMETER] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [PARAMETER] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [PARAMETER] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.5 INPUTTING AND OUTPUTTING TOOL OFFSET DATA

12.5.1 Outputting Tool Offset Data

The following explains the operation for outputting the tool offset data registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Tool offset data screen.

Method 1

Press the  function key several times, until the Tool offset data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [TOOL OFFSET] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [TOOL OFFSET] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [TOOL OFFSET] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [TOOL OFFSET] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the  address key.

- (3) Enter a file number.
- (4) Press the [TOOL OFFSET] soft key.

Method 5 (Specified with its file number (three of three methods))

- (2) Press the address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of tool offset data," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be TOOL-OFS.TXT.

12.5.2 Inputting Tool Offset Data

The following explains the operation for inputting the tool offset data to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Tool offset data screen.

Method 1

Press the  function key several times, until the Tool offset data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [TOOL OFFSET] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [TOOL OFFSET] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [TOOL OFFSET] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [TOOL OFFSET] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [TOOL OFFSET] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.6 INPUTTING AND OUTPUTTING WORKPIECE ZERO POINT OFFSET DATA

12.6.1 Outputting Workpiece zero point offset data

The following explains the operation for outputting the Workpiece zero point offset data registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Workpiece zero point offset data screen.

Method 1

Press the  function key several times, until the Workpiece zero point offset data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [WORK OFFSET] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [WORK OFFSET] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [WORK OFFSET] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [WORK OFFSET] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press  the address key.

- (3) Enter a file number.
- (4) Press the [WORK OFFSET] soft key.

Method 5 (Specified with its file number (three of three methods))

- (2) Press the address key.

- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of Workpiece zero point offset data," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be EXT-WKZ.TXT.

12.6.2 Inputting System Parameters

The following explains the operation for inputting the Workpiece zero point offset data to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Workpiece zero point offset data screen.

Method 1

Press the  function key several times, until the Workpiece zero point offset data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [WORK OFFSET] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [WORK OFFSET] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [WORK OFFSET] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [WORK OFFSET] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [WORK OFFSET] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.7 INPUTTING AND OUTPUTTING CUSTOM MACRO VARIABLE

12.7.1 Outputting Custom Macro Variable

The following explains the operation for outputting the Custom macro variable registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Custom macro variable screen.

Method 1

Press the  function key several times, until the Custom macro variable screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [MACRO VAL.] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [MACRO VAL.] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [MACRO VAL.] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [MACRO VAL.] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the  [PUNCH] soft key.
- (2) Press the _____ address key.

- (3) Enter a file number.
- (4) Press the [MACRO VAL.] soft key.

Method 5 (Specified with its file number (three of three methods))

- (2) Press the address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of Custom macro variable," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be MACRO.TXT.

12.7.2 Inputting Cuctom Macro Variable

The following explains the operation for inputting the Cuctom macro variable to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Cuctom macro variable screen.

Method 1

Press the  function key several times, until the Cuctom macro variable screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [MACRO VAL.] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [MACRO VAL.] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [MACRO VAL.] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [MACRO VAL.] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [MACRO VAL.] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.8 INPUTTING AND OUTPUTTING THREE-DIMENSIONAL ERROR COMPENSATION DATA

12.8.1 Outputting Three-dimensional Error Compensation Data

The following explains the operation for outputting the Three-dimensional error compensation data registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Three-dimensional error compensation data screen.

Method 1

Press the  function key several times, until the Three-dimensional error compensation data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [VOLUMETRIC] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [VOLUMETRIC] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [VOLUMETRIC] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [VOLUMETRIC] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.

- (2) Press the address key.
- (3) Enter a file number.
- (4) Press the [VOLUMETRIC] soft key.

Method 5 (Specified with its file number (three of three methods))

- (2) Press the address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of Three-dimensional error compensation data," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be VALCOMP.TXT.

12.8.2 Inputting Three-dimensional Error Compensation Data

The following explains the operation for inputting the Three-dimensional error compensation data to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Three-dimensional error compensation data screen.

Method 1

Press the  function key several times, until the Three-dimensional error compensation data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [VOLUMETRIC] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [VOLUMETRIC] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [VOLUMETRIC] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [VOLUMETRIC] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [VOLUMETRIC] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.9 INPUTTING AND OUTPUTTING TOOL OFFSET DATA BY TOOL NUMBER

12.9.1 Outputting Tool Offset Data by Tool Number

The following explains the operation for outputting the Tool offset data by tool number registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Tool offset data by tool number screen.

Method 1

Press the  function key several times, until the Tool offset data by tool number screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [T CODE OFFSET] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [T CODE OFFSET] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [T CODE OFFSET] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [T CODE OFFSET] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press  the address key.

- (3) Enter a file number.
- (4) Press the [T CODE OFFSET] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of Tool offset data by tool number," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be TOOL-DAT.TXT.

12.9.2 Inputting Tool Offset Data by Tool Number

The following explains the operation for inputting the Tool offset data by tool number to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Tool offset data by tool number screen.

Method 1

Press the  function key several times, until the Tool offset data by tool number screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [T CODE OFFSET] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [T CODE OFFSET] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [T CODE OFFSET] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [T CODE OFFSET] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [TOOL OFFSET] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.10 INPUTTING AND OUTPUTTING FIXTURE OFFSET DATA

12.10.1 Outputting Fixture Offset Data

The following explains the operation for outputting the fixture offset data registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Fixture offset data screen.

Method 1

Press the  function key several times, until the Fixture offset data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [FOFS] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [FOFS] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [FOFS] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [FOFS] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the  address key.

- (3) Enter a file number.
- (4) Press the [FOFS] soft key.

Method 5 (Specified with its file number (three of three methods))

- (2) Press  the address key.
- (2) Enter a file number.
- (3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of fixture offset data," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be FOFS.TXT.

12.10.2 Inputting Fixture Offset Data

The following explains the operation for inputting the fixture offset data to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Fixture offset data screen.

Method 1

Press the  function key several times, until the Fixture offset data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [FOFS] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [FOFS] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [FOFS] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [FOFS] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [FOFS] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.11 INPUTTING AND OUTPUTTING ROTARY HEAD DYNAMIC TOOL OFFSET DATA

12.11.1 Outputting Rotary Head Dynamic Tool Offset Data

The following explains the operation for outputting the Rotary head dynamic tool offset data registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Rotary head dynamic tool offset data screen.

Method 1

Press the  function key several times, until the Rotary head dynamic tool offset data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [DOFS] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [DOFS] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [DOFS] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [DOFS] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.

(2) Press the address key.

(3) Enter a file number.

(4) Press the [DOFS] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of Rotary head dynamic tool offset data," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be DOFS.TXT.

12.11.2 Inputting Rotary Head Dynamic Tool Offset Data

The following explains the operation for inputting the Rotary head dynamic tool offset data to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the Rotary head dynamic tool offset data screen.

Method 1

Press the  function key several times, until the Rotary head dynamic tool offset data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [DOFS] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [DOFS] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [DOFS] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [DOFS] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [DOFS] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press  the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.12 INPUTTING AND OUTPUTTING SYSTEM CONFIGURATION DATA

12.12.1 Outputting System Configuration Data

The following explains the operation for outputting the system configuration data registered in the memory of the CNC to a floppy disk.

Setup for output

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the System configuration data screen.

Method 1

Press the  function key several times, until the System configuration data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [SYSTEM CONFIG] soft key.

Execution of output

Method 1 (Neither the output file name nor number need be specified)

- (1) Press the [PUNCH] soft key.
- (2) Press the [SYSTEM CONFIG] soft key.

Method 2 (The output file is specified with its file name)

- (1) Press the [PUNCH] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [SYSTEM CONFIG] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [PUNCH] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [SYSTEM CONFIG] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [PUNCH] soft key.

(2) Press the address key.

(3) Enter a file number.

(4) Press the [SYSTEM CONFIG] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [PUNCH] soft key.

NOTE

When method 1 (in which neither the output file name nor number need be specified), described in "Output of system configuration data," is used to output system parameters to a FANUC Floppy Cassette, FANUC FA Card, FANUC Handy File, and memory card, the file name will be SYS_CONF.TXT.

12.12.2 Inputting System Configuration Data

The following explains the operation for inputting the system configuration data to the memory of the CNC to a floppy disk.

Setup for input

- (1) Select MDI mode.
- (2) Follow either of the procedures described below to display the System configuration data screen.

Method 1

Press the  function key several times, until the System configuration data screen appears.

Method 2

- (1) Press the  function key.
- (2) Press the [SYSTEM CONFIG] soft key.

Execution of input

Method 1 (Neither the input file name nor number need be specified)

- (1) Press the [READ] soft key.
- (2) Press the [SYSTEM CONFIG] soft key.

Method 2 (The input file is specified with its file name)

- (1) Press the [READ] soft key.
- (2) Press the ["FILE NAME] soft key.
- (3) Enter a file name.
- (4) Press the [FILE NAME"] soft key.
- (5) Press the [SYSTEM CONFIG] soft key.

Method 3 (Specified with its file number (one of three methods))

- (1) Press the [READ] soft key.
- (2) Press the [(FILE#)] soft key.
- (3) Enter a file number.
- (4) Press the [SYSTEM CONFIG] soft key.

Method 4 (Specified with its file number (two of three methods))

- (1) Press the [READ] soft key.
- (2) Press  the address key.
- (3) Enter a file number.
- (4) Press the [SYSTEM CONFIG] soft key.

Method 5 (Specified with its file number (three of three methods))

(2) Press the address key.

(2) Enter a file number.

(3) Press the [READ] soft key.

12.13 INPUTTING AND OUTPUTTING DATA USING THE FLOPPY DIRECTORY SCREEN

Usually, it is necessary to switch to the appropriate screen depending on the type of data to be input/output (for example, the Parameter screen to output and input parameters to and from an external device and the Program screen to output and input programs). By using the Floppy Directory screen, however, various types of data such as program and offset can be input and output on this single screen. The screen allows the input/output of the following types of data:

- NC program
- System parameter
- Pitch error compensation data
- Tool offset data
- Workpiece origin offset data
- Custom macro variable
- System log data
- Periodic maintenance data
- Item selection menu (machine system) data
- Volumetric compensation data
- Tool offset data by tool number
- Fixture offset data
- Rotary head dynamic tool compensation data
- System configuration data
- Operation/alarm history data

For details, see II-10.11, "Floppy Directory Screen."

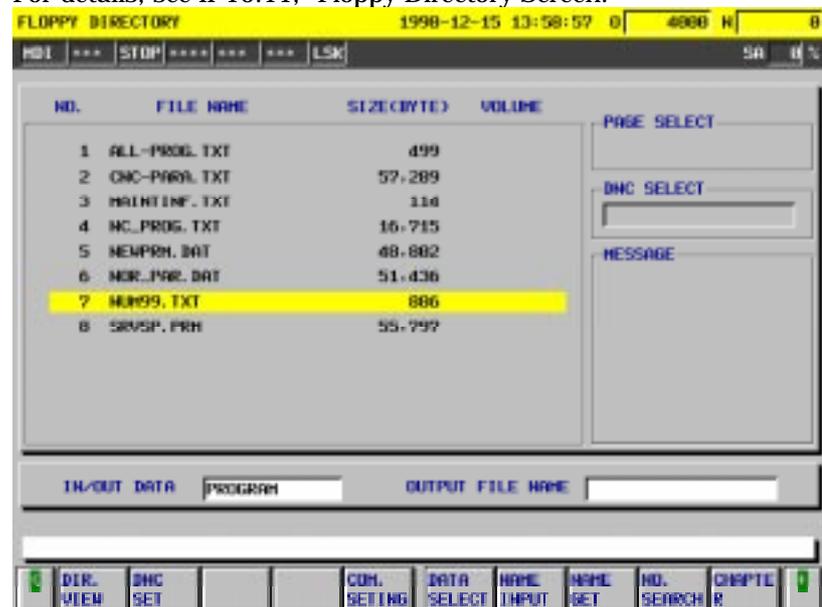


Fig. 12.13 (a) Floppy Directory Screen.

12.14 INPUTTING AND OUTPUTTING DATA USING THE MEMORY CARD SCREEN

Usually, it is necessary to switch to the appropriate screen depending on the type of data to be input/output (for example, the Parameter screen to output and input parameters to and from an external device and the Program screen to output and input programs). By using the Memory Card screen, however, various types of data such as program and offset can be input and output on this single screen.

The screen allows the input/output of the following types of data:

- NC program
- System parameter
- Pitch error compensation data
- Tool offset data
- Workpiece origin offset data
- Custom macro variable
- System log data
- Periodic maintenance data
- Item selection menu (machine system) data
- Volumetric compensation data
- Tool offset data by tool number
- Fixture offset data
- Rotary head dynamic tool compensation data
- System configuration data
- Operation/alarm history data

For details, see II-10.11, "Memory Card Screen."

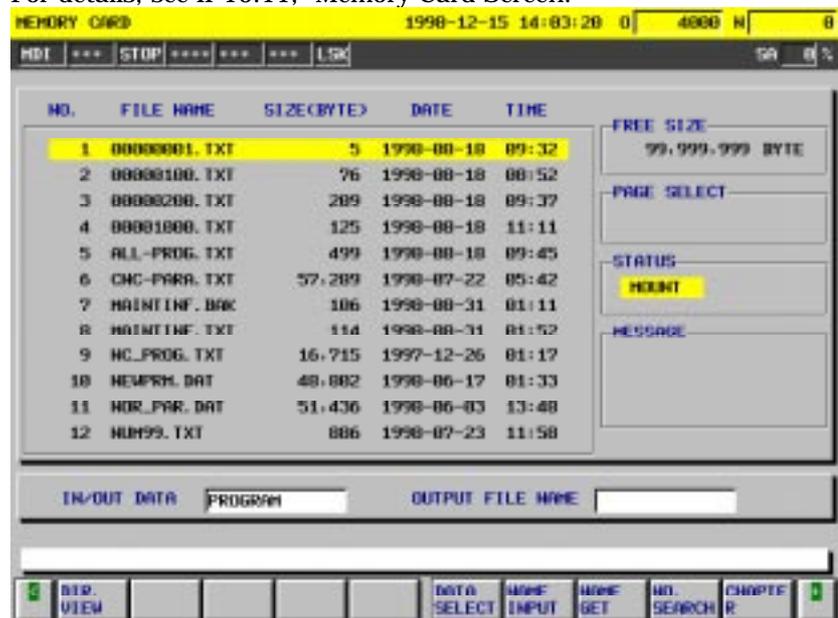


Fig. 12.14 (a) Memory Card Screen

12.15 External I/O Equipment control

General

It is possible to instruct in registration/the output of the program from the outside.

. Registration

The program is registered in the program memory by using the edit function from the external input equipment by external lead beginning signal EXRD.

. Output

All programs registered in the program memory are output by using the edit function for the external output equipment by external punch beginning signal EXPUN.

13

AXIS CONTROL

13.1 AXIS CONTROL BY THE PMC

Any axes can be isolated from control of the CNC and controlled directly by the PMC. The PMC can issue commands specifying a traveled distance, feedrate, and so forth to move the tool along these axes independently of the other axes operating under the CNC. Therefore, peripheral equipment such as a turret, pallet, or index table can be controlled by using an axis of the CNC.

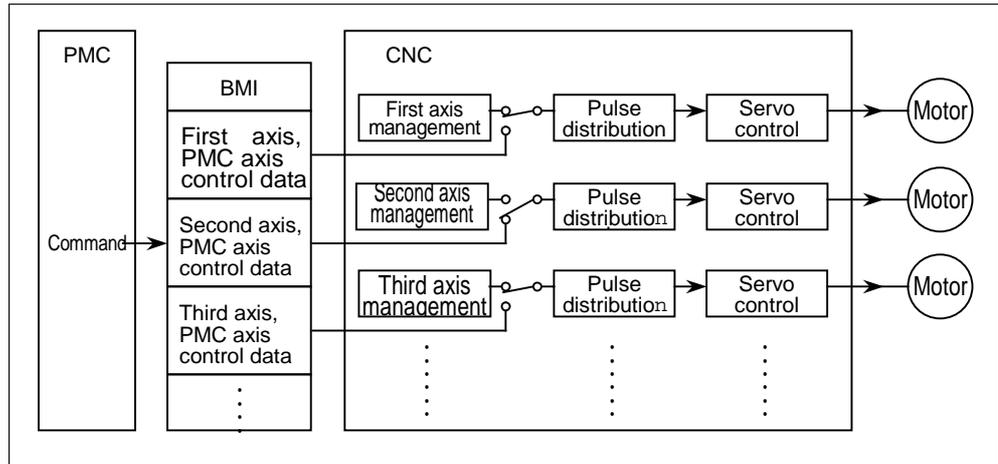
Whether to place each axis under control of the CNC or PMC can be selected with a relevant input signal.

The following operations can be controlled by the PMC directly:

- (1) Rapid traverse with a traveled distance specified
- (2) Cutting feed with amount of travel specified feed per minute
- (3) Cutting feed with amount of travel specified sec/block specification
- (4) Dwell
- (5) Reference position return
- (6) Feedrate command
- (7) Auxiliary function
- (8) Machine coordinate system selection
- (9) Torque control command

With cutting feed with an amount of travel specified, mentioned in (2) and (3), the following are possible:

- Specification of the feedrate and override
- Simultaneous start of cutting feed with another PMC-controlled axis



Refer to the manual issued by the machine tool builder for details.

13.2 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. To use this function, set the setting parameter MIR (bit 0 of No.0012) mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the MDI panel.

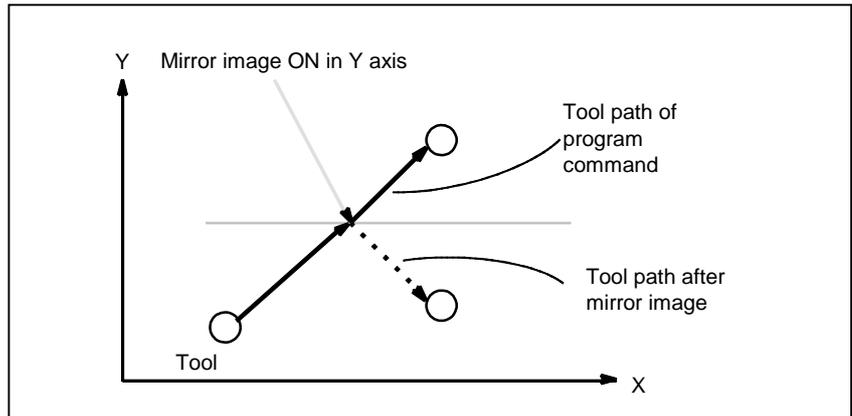


Fig13.2 (b) Mirror Image

Procedure

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

- 1 Press the single block switch to stop automatic operation. When the mirror image function is used from the beginning of operation, this step is omitted.
- 2 Press the mirror image switch for the target axis on the machine operator's panel.
- 3 Enter an automatic operation mode (memory mode or MDI mode).
- 4 then press the cycle start button to start automatic operation.

Explanation

If the mirror image function is enabled, the movement of the machine and the coordinates will be as follows for a positive-direction move command:

	Machine movement	Machine coordinate	Relative coordinate	Absolute coordinate
Automatic operation	Positive direction	Positive direction	Positive direction	Positive direction
Manual operation	Negative direction	Negative direction	Negative direction	Negative direction

Limitation

The direction of movement during manual operation, the direction of movement from an intermediate point to the reference position during automatic reference position return (G28), the direction of approach during unidirectional positioning (G60), and the shift direction in a boring cycle (G76, G87) cannot be reserved.

NOTE

1 The mirror image cannot be switched during axis movement. Always switch the mirror image during stopped state.

2 For mirror image during manual operation, either on or off of mirror image that was valid by setting operation or by external mirror image signal when the mode was switched to the manual operation is effective. Switching of mirror image state either by setting operation or by external mirror image signal is not effective in the manual operation.

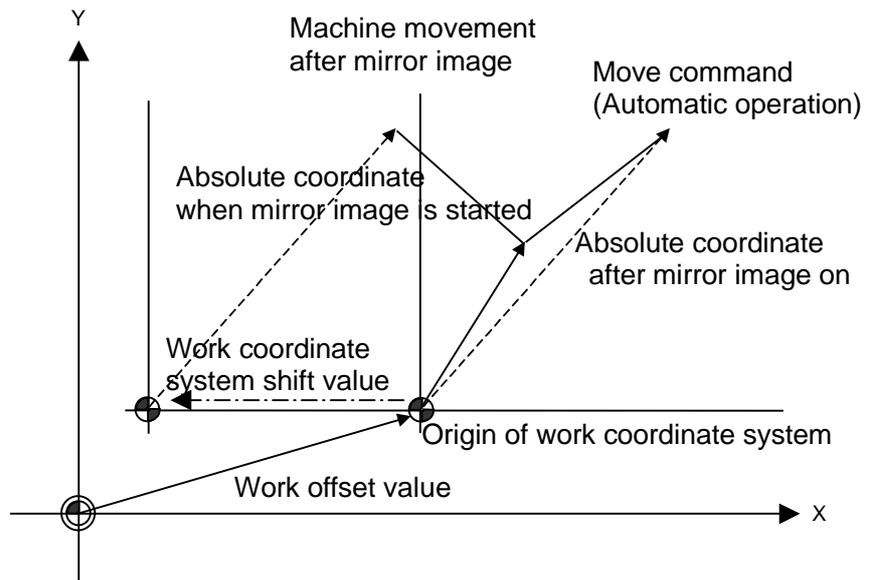


Fig13.2 (c) Shift of the Work Coordinate System Due to Movement

14 ABSOLUTE-POSITION DETECTION

On an axis equipped with an absolute pulse coder, the machine position is monitored even when the CNC power is turned off, so return to the reference position is unnecessary when the power is turned on again. An absolute pulse coder consists of an ordinary incremental pulse coder equipped with an absolute counter. Based on the counter value, an absolute position can be detected.

The absolute pulse coder uses the backup battery so that the pulse coder monitors the machine position by the counter if the CNC is turned off. If an absolute pulse coder is installed on an axis, the current position on that axis is read from the absolute counter when the CNC is turned on, and the machine coordinate system and workpiece coordinate system are automatically set based on the read value. Automatic operation can thus start immediately.

The requirements for reference position return after power-on operation, described in this manual, do not apply to axes equipped with an absolute pulse coder.

Explanation

£ Initial Setting at Power-On

- (1) The machine coordinate system and a workpiece coordinate system are set automatically. The G54 coordinate system is selected as the workpiece coordinate system.
- (2) The offsets from the workpiece reference point which are specified with G92 and G52 are cleared.
- (3) Relative coordinates (RELATIVE) for indicating a position are set with the DSE bit, bit 1, of parameter No. 2202.
- (4) The other initial settings at power-on are the same as when the absolute position detection function is not provided.

£ Manual Reference Position Return

In the following cases, be sure to manually return to the reference position to associate the reference position with the counter value of the absolute pulse coder:

- (a) First field adjustment (after the reference position is defined)
- (b) When the reference position is changed
- (c) When the absolute pulse coder is replaced.
- (d) When alarm OT32 occurs
- (e) When the Cife memory is replaced
- (f) When the Pile memory is cleared

Manually return to the reference position as follows:

- (1) Set APZ, bit 4 of parameter No. 1815, to 0.
- (2) Turn the power off then on again. Alarm OT32 is then displayed.
- (3) Return to the reference position manually. When tool reaches the reference position, parameter APZ is automatically set to 1.
- (4) Press the reset button. Alarm OT32 is then released.

£ Setting the origin with MDI operation

When an absolute pulse coder or servo motor is replaced with another, the setting of the origin of the absolute pulse coder (association of the reference position with the counter value of the new absolute pulse coder) is required.

There are two ways to setting the origin of the absolute pulse coder: using a manual reference position return, and moving the machine to the reference position manually and then performing MDI operation necessary for setting the origin is performed.

The setting of the origin of the absolute pulse coder using MDI operation eliminates the need for the dog for deceleration for a manual reference position return or the manual reference position mode selection switch. It does, however, require some means for ensuring that the machine is at the reference position.

The origin can be set by moving the machine to the reference position by means of jog feed or manual handle feed and then specifying the appropriate parameters from the MDI.

The procedure is as described below:

- (1) Set parameter APZ (bit 4 of No. 1815) to "0."
An alarm message "POWER MUST BE OFF" is displayed.
- (2) Turn the power off and then on again.
Alarm OT32 NEED ZRN (ABS PCDR) is displayed.
- (3) Move the machine to the reference position, using jog feed, manual handle feed, or manual incremental feed.

- (4) Set parameter APZ (bit 4 of No. 1815) to "1."
An alarm message "POWER MUST BE OFF" is displayed.
- (5) Turn the power off and then on again.

With the following procedure, the origin is set.

£ Removing controlled axes

On the axes on which absolute pulse coders are mounted, controlled axes cannot be removed. For an explanation of removing controlled axes, refer to the manual supplied by the machine tool builder.

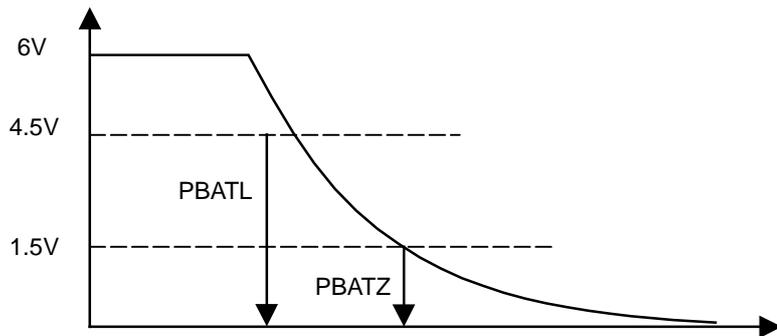
£ Alarm

The following alarms may be detected when the power is turned on.

Number	Message	Contents
OT0032	NEED ZRN(ABS PCDR)	The reference position and the absolute pulse coder counter value do not match.
OT0034	BATTERY ZERO(ABS PCDR)	The battery voltage of the absolute position detector has fallen to "0", or power is supplied to the pulse coder for the first time.
OT0035	IMPOSSIBLE ZRN(SERIAL)	An attempt was made to create correspondence between the reference position and the absolute position detector when the origin cannot be established.
OT0036	BATTERY DOWN (ABS PCDR)	Low absolute position detector battery voltage
SV0101	DATA ERROR(ABS PCDR)	A correct machine position cannot be obtained as the absolute pulse coder has malfunctioned or the machine moved too far during power ON.

CAUTION

The voltage level of the backup battery for the absolute pulse coder is normally 6 V. Over time, however, the voltage level drops.



DO.PBATL:Battery

alarm

When this alarm occurs, "BAT" is displayed on the status bar on the screen.

In addition, alarm OT36 BATTERY DOWN (ABS PCDR) is indicated.

DO.PBATZ:Battery

zero

alarm

This alarm occurs if the power to the NC unit is turned off when the voltage

is approximately 1.5 V or less.

In this case, alarm OT32 NEED ZRN (ABS PCDR) or OT34 BATTERY ZERO (ABS PCDR) is indicated.

IV MAINTENANCE

1

INVESTIGATION OF TROUBLE GENERATION SITUATION

- Software configuration screen

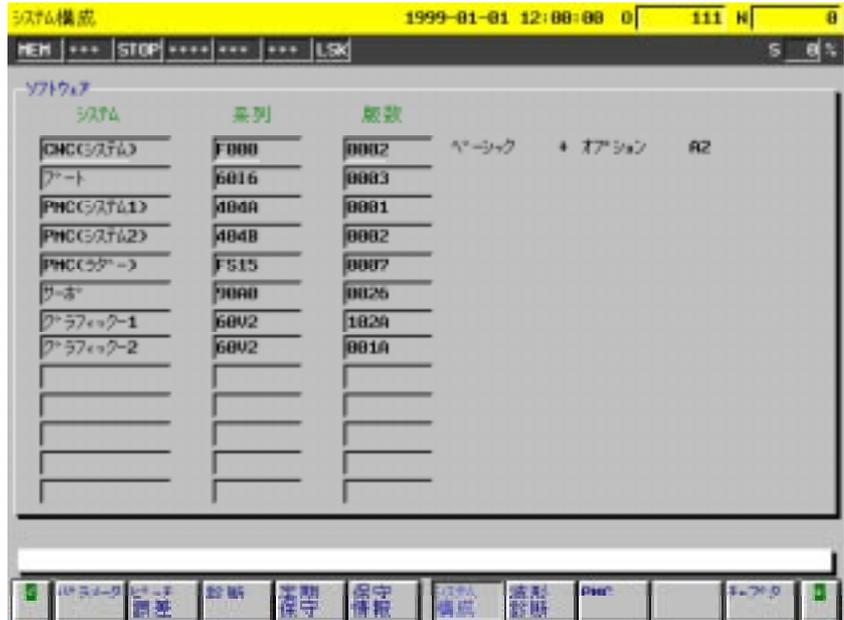


Fig. 1.1 (b) Software configuration screen

- Module configuration screen



Fig. 1.1 (c) Module configuration screen

- Content of display

Refer to FANUC Series 15i/150i-MODEL A Maintenance Manual (B-63325EN), for details of content of display.

APPENDIX

1

STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET

Parameter CLR (No. 3402#6) is used to select whether resetting the CNC places it in the cleared state or in the reset state (0: reset state/1: cleared state).

The symbols in the tables below mean the following:

- ☐ The status is not changed or the movement is continued.
- ~ ☐ The status is cancelled or the movement is interrupted.

Item		When turning power on	Cleared	Reset
Setting data	Offset value			
	Data set by the MDI setting operation			
	Parameter			
Various Data	Programs in memory			
	Contents in the buffer storage	~	~	☐MDI mode ~ ☐Other mode
	Display of sequence number		(Note 1)	(Note 1)
	One shot G code	~	~	~
	Modal G code	Initial G codes. (The G20 and G21 codes return to the same state they were in when the power was last turned off.)	Initial G codes. (G20/G21 are not changed.)	
	F	Zero	Zero	
	S, T, M	~		
K (Number of repeats)	~	~	~	
Work coordinate value		Zero		

Item		When turning power on	Cleared	Reset
Action in operation	Movement	~	~	~
	Dwell	~	~	~
	Issuance of M, S and T codes	~	~	~
	Tool length compensation	~	Depending on parameter (No.5003#6) on LVK	≠MDI mode Other modes depend on parameter LVK(No.5003#6).
	Cutter compensation	~	~	≠MDI mode ~ ≠Other mode
	Storing called subprogram number	~	~(Note 2)	≠MDI mode ~ ≠Other mode (Note 2)
Output signals	CNC alarm signal AL	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm
	Reference position return completion LED	~	(~ ≠Emergency stop)	(~ ≠Emergency stop)
	S, T and B codes	~		
	M code	~	~	~
	M, S and T strobe signals	~	~	~
	Spindle revolution signal (S analog signal)	~		
	CNC ready signal MA	ON		
	CNC ready signal SA	ON (When other than servo alarm)	ON (When other than servo alarm)	ON (When other than servo alarm)
	Cycle start LED (STL)	~	~	~
	Feed hold LED (SPL)	~	~	~

- *No part of this manual may be reproduced in any form.*
- *All specifications and designs are subject to change without notice.*