GE Fanuc Series 20-FA

Operators Manual

COMPUTER NUMERICAL CONTROLS

B-62174E/03

99990C191134 1DIST

GE Fanuc Automation

B-62174E

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.

Table of Contents

I. GENERAL

1. GENERAL 3

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL 5
1.2 NOTES ON READING THIS MANUAL 7

II. PROGRAMMING

1. GENERAL 11

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE–INTERPOLATION 12
1.2 FEED–FEED FUNCTION 14
1.3 PART DRAWING AND TOOL MOVEMENT 15
1.3.1 Reference Position (Machine–Specific Position) 15
1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System 16
1.3.3 How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands 19
1.4 CUTTING SPEED – SPINDLE SPEED FUNCTION 20
1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION 21
1.6 COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION 22
1.7 PROGRAM CONFIGURATION 23
1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM 26
1.9 TOOL MOVEMENT RANGE – STROKE 27

2. CONTROLLED AXES 28

2.1 CONTROLLED AXES 29
2.2 AXIS NAME 29
2.3 INCREMENT SYSTEM 29
2.4 MAXIMUM STROKE 29

3. PREPARATORY FUNCTION (G FUNCTION) 30

4. INTERPOLATION FUNCTIONS 34

4.1 POSITIONING (G00) 35
4.2 SINGLE DIRECtION POSITIONING (G60) 37
4.3 LINEAR INTERPOLATION (G01) 39
4.4 CIRCULAR INTERPOLATION (G02,G03) 40
4.5 HELICAL INTERPOLATION (G02,G03) 44
4.6 SKIP FUNCTION(G31) 45

5. FEED FUNCTIONS 47

5.1 GENERAL 48
5.2 RAPID TRAVERSE 49
5.3 CUTTING FEED 50

5.4 CUTTING FEEDRATE CONTROL 52
5.4.1 Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63) 53
5.5 DWELL (G04) 54
5.6 AUTOMATIC ACCELERATION/DECELERATION 55
5.6.1 Automatic Acceleration/Deceleration 55
5.6.2 Linear Acceleration/Deceleration after Interpolation for Cutting Feed 57

6. REFERENCE POSITION 59

7. COORDINATE SYSTEM 63

7.1 MACHINE COORDINATE SYSTEM 64
7.1.1 Setting a Machine Coordinate System 64
7.1.2 Selecting a Machine Coordinate System (G53) 64
7.2 WORKPIECE COORDINATE SYSTEM 66
7.2.1 Setting a Workpiece Coordinate System 66
7.2.2 Selecting a Workpiece Coordinate System 67
7.2.3 Changing Workpiece Coordinate System 68
7.3 LOCAL COORDINATE SYSTEM 70
7.4 PLANE SELECTION 72

8. COORDINATE VALUE AND DIMENSION 73

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91) 74
8.2 INCH/METRIC CONVERSION(G20,G21) 75
8.3 DECIMAL POINT PROGRAMMING 76

9. SPINDLE SPEED FUNCTION (S FUNCTION) 77

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE 78
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5–DIGIT COMMAND) 78

10. TOOL FUNCTION (T FUNCTION) 79

10.1 TOOL SELECTION FUNCTION 80

11. AUXILIARY FUNCTION 81

11.1 AUXILIARY FUNCTION (M FUNCTION) 82
11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK 83

12. PROGRAM CONFIGURATION 84

12.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS 86
12.2 PROGRAM SECTION CONFIGURATION 89
12.3 SUBPROGRAM 95

13. FUNCTIONS TO SIMPLIFY PROGRAMMING 99

13.1 CANNED CYCLE 100
13.1.1 High–speed Peck Drilling Cycle (G73) 104
13.1.2 Left–handed Tapping Cycle (G74) 106

13.1.3 Fine Boring Cycle (G76) 108
13.1.4 Drilling Cycle, Spot Drilling (G81) 110
13.1.5 Drilling Cycle Counter Boring Cycle(G82) 112
13.1.6 Peck Drilling Cycle (G83) 114
13.1.7 Tapping Cycle (G84) 116
13.1.8 Boring Cycle (G85) 118
13.1.9 Boring Cycle (G86) 120
13.1.10 Boring Cycle Back Boring Cycle (G87) 122
13.1.11 Boring Cycle (G88) 124
13.1.12 Boring Cycle (G89) 126
13.1.13 Canned Cycle Cancel (G80) 128
13.2 RIGID TAPPING 131
13.2.1 Rigid Tapping (G84) 132
13.2.2 Left–handed Rigid Tapping Cycle (G74) 134
13.2.3 Peck Rigid Tapping Cycle (G84 or G74) 136
13.2.4 Canned Cycle Cancel (G80) 138
13.3 EXTERNAL MOTION FUNCTION (G81) 139

14. COMPENSATION FUNCTION 140

14.1 TOOL LENGTH OFFSET (G43,G44,G49) 141
14.2 OVERVIEW OF CUTTER COMPENSATION C (G40 – G42) 145
14.3 DETAILS OF CUTTER COMPENSATION C 151
14.3.1 General 151
14.3.2 Tool Movement in Start–up 152
14.3.3 Tool Movement in Offset Mode 156
14.3.4 Tool Movement in Offset Mode Cancel 170
14.3.5 Interference Check 176
14.3.6 Overcutting by Cutter Compensation 181
14.3.7 Input Command from MDI 184
14.4 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10) 185
14.5 SCALING (G50,G51) 186
14.6 COORDINATE SYSTEM ROTATION (G68, G69) 192

15. CUSTOM MACRO 198

15.1 VARIABLES 199
15.2 SYSTEM VARIABLES 203
15.3 ARITHMETIC AND LOGIC OPERATION 210
15.4 MACRO STATEMENTS AND NC STATEMENTS 214
15.5 BRANCH AND REPETITION 215
15.5.1 Unconditional Branch (GOTO Statement) 215
15.5.2 Conditional Branch (IF Statement) 215
15.5.3 Repetition (While Statement) 216
15.6 MACRO CALL 219
15.6.1 Simple Call (G65) 219
15.6.2 Modal Call (G66) 224
15.6.3 Macro Call Using G Code 226

15.6.4 Macro Call Using an M Code 227
15.6.5 Subprogram Call Using an M Code 228
15.6.6 Subprogram Calls Using a T Code 229
15.6.7 Sample Program 230
15.7 PROCESSING MACRO STATEMENTS 232
15.8 REGISTERING CUSTOM MACRO PROGRAMS 234
15.9 LIMITATIONS 235
15.10 EXTERNAL OUTPUT COMMANDS 236

16. PROGRAMMABLE PARAMETER ENTRY (G10) 240

III. OPERATION

1. GENERAL 245

1.1 MANUAL OPERATION 246
1.2 TOOL MOVEMENT BY PROGRAMING–AUTOMATIC OPERATION 248
1.3 AUTOMATIC OPERATION 249
1.4 TESTING A PROGRAM 251
1.4.1 Check by Running the Machine 251
1.4.2 How to View the Position Display Change without Running the Machine 252
1.5 EDITING A PART PROGRAM 253
1.6 DISPLAYING AND SETTING DATA 254
1.7 DISPLAY 257
1.7.1 Program Display 257
1.7.2 Current Position Display 258
1.7.3 Alarm Display 258
1.7.4 Parts Count Display, Run Time Display 259
1.8 DATA INPUT/OUTPUT 260

2. OPERATIONAL DEVICES 261

2.1 CRT/MDI PANELS 262
2.2 FUNCTION KEYS AND SOFT KEYS 265
2.2.1 General Screen Operations 265
2.2.2 Function Keys 266
2.2.3 Soft Keys 267
2.2.4 Key Input and Input Buffer 283
2.2.5 Warning Messages 284
2.2.6 Function Selection Soft Keys 285
2.3 EXTERNAL I/O DEVICES 286
2.3.1 FANUC Handy File 288
2.3.2 FANUC Floppy Cassette 288
2.3.3 FANUC FA Card 289
2.3.4 FANUC PPR 289
2.3.5 Portable Tape Reader 290
2.4 POWER ON/OFF 291

2.4.1 Turning on the Power 291
2.4.2 Screen Displayed at Power–on 292
2.4.3 Power Disconnection 293

3. MANUAL OPERATION 294

3.1 MANUAL REFERENCE POSITION RETURN 295
3.2 JOG FEED 297
3.3 INCREMENTAL FEED 299
3.4 MANUAL HANDLE FEED 300
3.5 MANUAL ABSOLUTE ON AND OFF 302

4. AUTOMATIC OPERATION 307

4.1 MEMORY OPERATION 308
4.2 MDI OPERATION (GUIDANCE PROGRAMMING MDI OPERATION) 311
4.3 DNC OPERATION 318
4.4 PROGRAM RESTART 319
4.5 SCHEDULING FUNCTION 326
4.6 SUBPROGRAM CALL FUNCTION 331
4.7 MANUAL HANDLE INTERRUPTION 333
4.8 MIRROR IMAGE 336
4.9 MANUAL INTERVENTION AND RETURN 338

5. TEST OPERATION 340

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK 341
5.2 FEEDRATE OVERRIDE 342
5.3 RAPID TRAVERSE OVERRIDE 343
5.4 DRY RUN 344
5.5 SINGLE BLOCK 345

6. SAFETY FUNCTIONS 347

6.1 EMERGENCY STOP 348
6.2 OVERTRAVEL 349
6.3 STROKE CHECK 350

7. ALARM AND SELF–DIAGNOSIS FUNCTIONS 354

7.1 ALARM DISPLAY 355
7.2 ALARM HISTORY DISPLAY 357
7.3 CHECKING BY SELF–DIAGNOSTIC SCREEN 358

8. DATA INPUT/OUTPUT 361

8.1 FILES 362
8.2 FILE SEARCH 364
8.3 PROGRAM INPUT/OUTPUT 366
8.3.1 Inputting a Program 366
8.3.2 Outputting a Program 368

8.4 OFFSET DATA INPUT AND OUTPUT 370
8.4.1 Inputting Offset Data 370
8.4.2 Outputting Offset Data 371
8.5 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA 372
8.5.1 Inputting Parameters 372
8.5.2 Outputting Parameters 373
8.5.3 Inputting Pitch Error Compensation Data 374
8.5.4 Outputting Pitch Error Compensation Data 375
8.6 INPUTTING/OUTPUTTING CUSTOM MACRO COMMON VARIABLES 376
8.6.1 Inputting Custom Macro Common Variables 376
8.6.2 Outputting Custom Macro Common Variable 377
8.7 DISPLAYING DIRECTORY OF FLOPPY DISK 378
8.7.1 Displaying the Directory 379
8.7.2 Reading Files 382
8.7.3 Outputting Programs 383
8.7.4 Deleting Files 384

9. EDITING PROGRAMS 386

9.1 EDITING WITH GUIDANCE PROGRAMMING 388
9.1.1 Displaying the Contents of a Program 388
9.1.2 Modifying the Contents of a Program 390
9.1.3 Deleting a Block 391
9.1.4 Inserting a Block 392
9.1.5 Deleting a Program 393
9.1.6 Copying a Program 394
9.1.7 Renaming a Program 395
9.1.8 Editing in CNC Language 396
9.2 INSERTING, ALTERING AND DELETING A WORD 398
9.2.1 Word Search 399
9.2.2 Heading a Program 401
9.2.3 Inserting a Word 402
9.2.4 Altering a Word 403
9.2.5 Deleting a Word 404
9.3 DELETING BLOCKS 405
9.3.1 Deleting a Block 405
9.3.2 Deleting Multiple Blocks 406
9.4 PROGRAM NUMBER SEARCH 407
9.5 SEQUENCE NUMBER SEARCH 408
9.6 DELETING PROGRAMS 410
9.6.1 Deleting One Program 410
9.6.2 Deleting All Programs 410
9.6.3 Deleting More Than One Program by Specifying a Range 411
9.7 EDITING OF CUSTOM MACROS 412
9.8 BACKGROUND EDITING 413
9.9 PASSWORD FUNCTION 414

10. CREATING PROGRAMS 416
10.1 CREATING A PROGRAM WITH GUIDANCE PROGRAMMING 417
10.1.1 Program Menu Screen 417
10.1.2 Registering a Program 418
10.1.3 Comment 421
10.1.4 Guidance Programming 422
10.2 CREATING A PROGRAM BY CYCLE PROGRAMMING 433
10.2.1 Facing 435
10.2.2 Hole Mahcining 444
10.2.3 Pattern Positioning 448
10.2.4 Side Cutting 453
10.2.5 Pocketing 466
10.3 CREATING PROGRAM A CNC LANGUAGE WITH THE MDI PANEL 476
10.4 AUTOMATIC INSERTION OF SEQUENCE NUMBERS 478
10.5 CREATING PROGRAMS IN TEACH IN MODE 480

11. SETTING AND DISPLAYING DATA 483
11.1 SCREENS DISPLAYED BY FUNCTION KEY 491
11.1.1 Position Display in the Work Coordinate System 492
11.1.2 Position Display in the Relative Coordinate System 493
11.1.3 Overall Position Display 495
11.1.4 Actual Feedrate Display 496
11.1.5 Display of Run Time and Parts Count 497
11.1.6 Operating Monitor Display 498
11.2 SCREENS DISPLAYED BY FUNCTION KEY (IN MEMORY MODE OR MDI MODE) 499
11.2.1 Program Contents Display 500
11.2.2 Current Block Display Screen 501
11.2.3 Next Block Display Screen 502
11.2.4 Program Check Screen 503
11.2.5 Program Screen for MDI Operation 504
11.3 SCREENS DISPLAYED BY FUNCTION KEY (IN THE EDIT MODE) 505
11.3.1 Displaying Memory Used and a List of Programs 505
11.4 SCREENS DISPLAYED BY FUNCTION KEY 508
11.4.1 Displaying and Setting the Tool Offset Value 509
11.4.2 Displaying and Entering Setting Data 511
11.4.3 Displaying and Setting Run Time,Parts Count, and Time 513
11.4.4 Displaying and Setting the Workpiece Origin Offset Value 515
11.4.5 Input of Measured Workpiece Origin Offsets 516
11.4.6 Displaying and Setting Custom Macro Common Variables 518
11.4.7 Displaying and Setting the Software Operator's Panel 519
11.5 SCREENS DISPLAYED BY FUNCTION KEY 521
11.5.1 Displaying and Setting Parameters 522
11.5.2 Displaying and Setting Pitch Error Compensation Data 524
11.6 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION 526
11.6.1 Displaying the Program Number and Sequence Number 526
11.6.2 Displaying the Status and Warning for Data Setting or Input/Output Operation 527

12. HELP FUNCTION 529

IV. MAINTENANCE

1. METHOD OF REPLACING BATTERY 537
1.1 REPLACING CNC BATTERY FOR MEMORY BACK–UP 538
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER 540
1.3 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER (SERVO AMPLIFIER CONVERTER UNIT ONLY FOR SERIES 20 OR α SERIES SERVO AMPLIFIER MODULE) 541

APPENDIX

A. TAPE CODE LIST 545

B. LIST OF FUNCTIONS AND TAPE FORMAT 548

C. RANGE OF COMMAND VALUE 553

D. NOMOGRAPHS 555
D.1 INCORRECT THREADED LENGTH 556
D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH 558
D.3 TOOL PATH AT CORNER 560
D.4 RADIUS DIRECTIONERROR AT CIRCLE CUTTING 563

E. STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET 564

F. CHARACTER–TO–CODES CORRESPONDENCE TABLE 566

G. ALARM LIST 567

H. OPERATION OF PORTABLE TAPE READER 582

I GENERAL

1 GENERAL

About this manual

This manual consists of the following parts:

I. GENERAL

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

II. PROGRAMMING

Describes each function: Format used to program functions in the NC language, characteristics, and restrictions.

III. OPERATION

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

Refer to the MACHINING GUIDANCE FUNCTION OPERATOR'S MANUAL (B–62174E–1) for details of the machining guidance function.

IV. MAINTENANCE

Describes alarms, self–diagnosis, and procedures for replacing fuses and batteries.

V. APPENDIX

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

This manual does not describe parameters in detail. For details on parameters mentioned in this manual, refer to the manual for parameters (B–62180E).

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

Applicable models

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

Product name	Abbreviations	
FANUC Series 20-FA	20-FA	Series 20

Special symbols

This manual uses the following symbols:

P_ : 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 the FANUC Series 20–FA. In the table, this manual is marked with an asterisk (*).

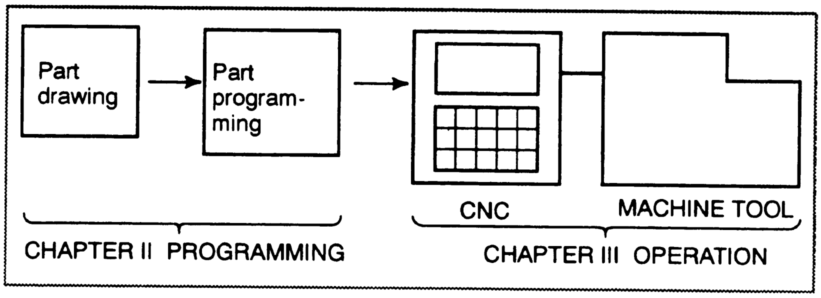
Table 1 Manuals Related to the FANUC Series 20–FA

Manual name	Specification number	
FANUC Series 20–FA DESCRIPTIONS	B–62172E	
FANUC Series 20–FA CONNECTION MANUAL	B–62173E	
FANUC Series 20–FA OPERATOR'S MANUAL	B–62174E	*
FANUC Series 20–FA MACHINING GUIDANCE FUNCTION OPERATOR'S MANUAL	B–62174E–1	
FANUC Series 20–FA MAINTENANCE MANUAL	B–62175E	
FANUC Series 20–FA PARAMETER MANUAL	B–62180E	
FANUC PMC–MODEL RA1/RA2/RB/RB2/RC PROGRAMMING MANUAL (LADDER LANGUAGE)	B–61863E	
FANUC Series 16/18/20 PROGRAMMING MANUAL (Macro Compiler / Macro Executer)	B–61803E–1	
FAPT MACRO COMPILER (For personal computer) PROGRAMMING MANUAL	B–66102E	

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 Chapter II. 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 Chapter III. OPERATION.

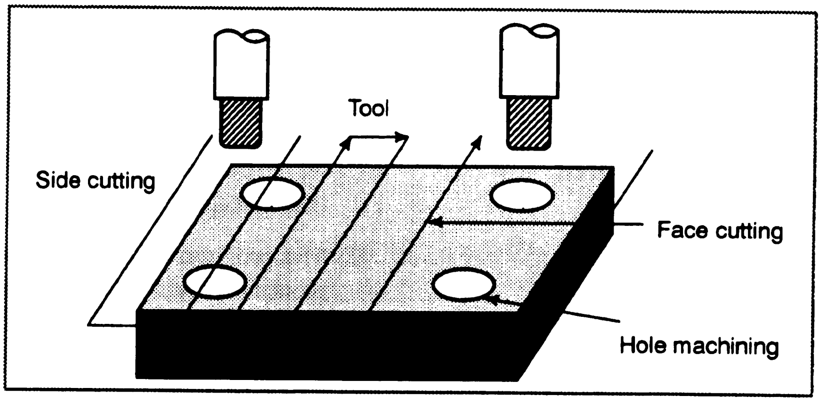


Before the actual programming, make the machining plan for how to machine the part.
Machining plan

1. Determination of workpieces machining range
2. Method of mounting workpieces on the machine tool
3. Machining sequence in every machining process
4. Cutting tools and machining conditions

Decide the machining method in every machining process.

Machining process / Machining procedure	1	2	3
	Feed cutting	**Side cutting**	**Hole machining**
1. Machining method : Rough Semi Finish			
2. Machining tools			
3. Machining conditions : Feedrate Cutting depth			
4. 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

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 though these headings.

> 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.

> 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 PROGRAMMING

1 GENERAL

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE– INTERPOLATION

The tool moves along straight lines and arcs constituting the workpiece parts figure (See II–4).

Explanations

The function of moving the tool along straight lines and arcs is called the interpolation.

- **Tool movement along a straight line**

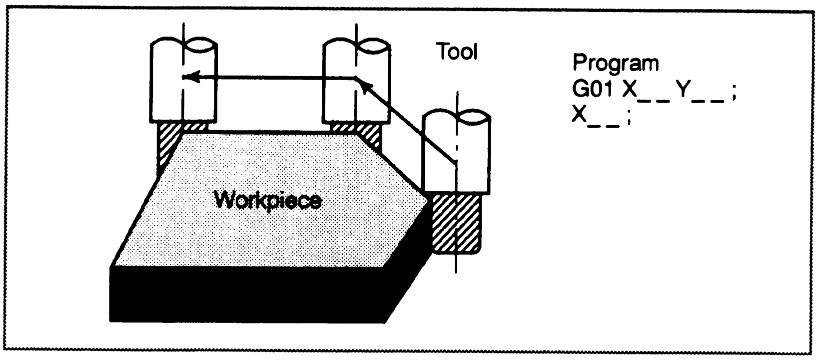


Fig.1.1 (a) Tool movement along a straight line

- **Tool movement along an arc**

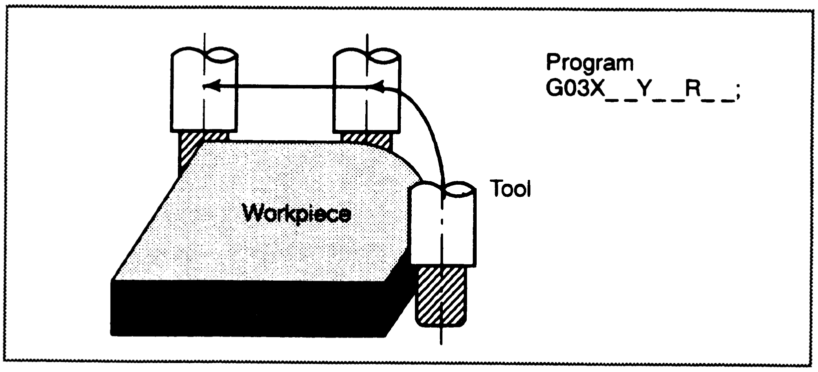


Fig. 1.1 (b) Tool movement along an arc

Symbols of the programmed commands G01, G02, ... are called the preparatory function and specify the type of interpolation conducted in the control unit.

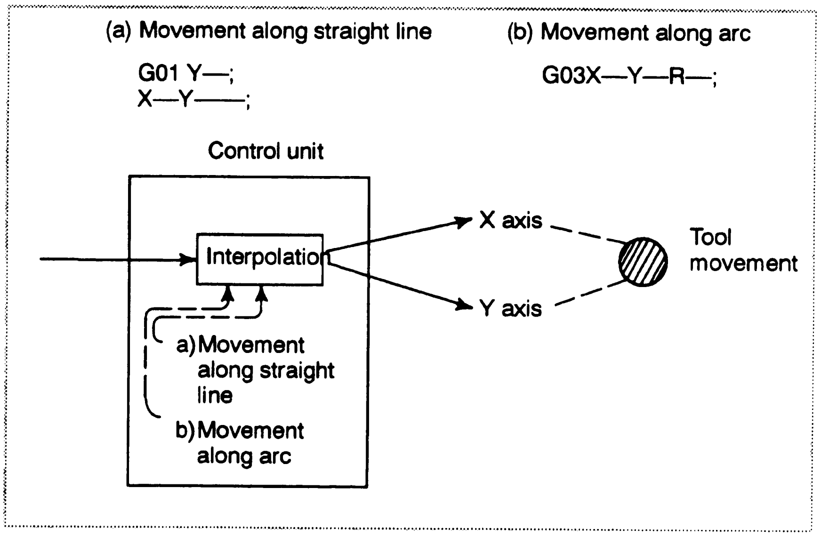


Fig. 1.1 (c) Interpolation function

> **Note**
> Some machines move tables instead of tools but this manual assumes that tools are moved against workpieces.

1.2 FEED– FEED FUNCTION

Movement of the tool at a specified speed for cutting a workpiece is called the feed.

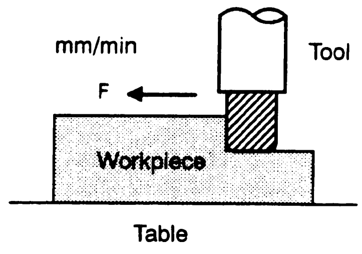


Fig. 1.2 (a) Feed function

Feedrates can be specified by using actual numerics. For example, to feed the tool at a rate of 150 mm/min, specify the following in the program: F150.0

The function of deciding the feed rate is called the feed function (See II–5).

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine–Specific Position)

A CNC machine tool is provided with a fixed position. Normally, tool change and programming of absolute zero point as described later are performed at this position. This position is called the reference position.

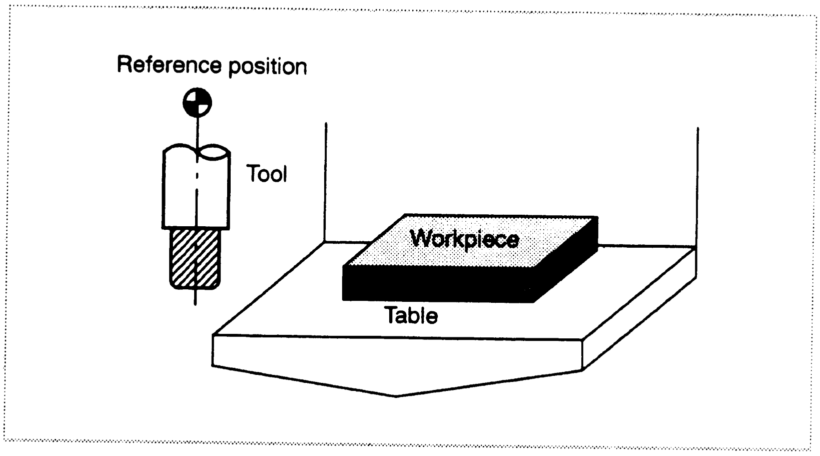


Fig. 1.3 (a) Reference position

Explanations

The tool can be moved to the reference position in two ways:

(1) Manual reference position return (See III–3.1)
Reference position return is performed by manual button operation.

(2) Automatic reference position return (See II–6)
In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System

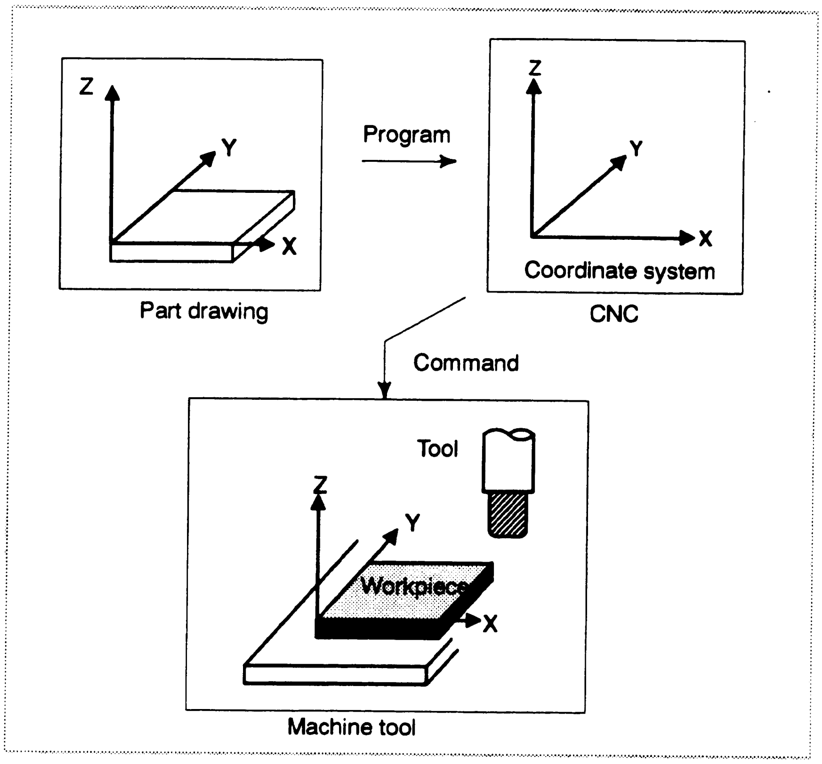


Fig. 1.3.2 (a) Coordinate system

Explanations

- **Coordinate system**

The following two coordinate systems are specified at different locations: (See II–7)

(1) Coordinate system on part drawing
The coordinate system is written on the part drawing. As the program data, the coordinate values on this coordinate system are used.

(2) Coordinate system specified by the CNC
The coordinate system is prepared on the actual machine tool table. This can be achieved by programming the distance from the current position of the tool to the zero point of the coordinate system to be set.

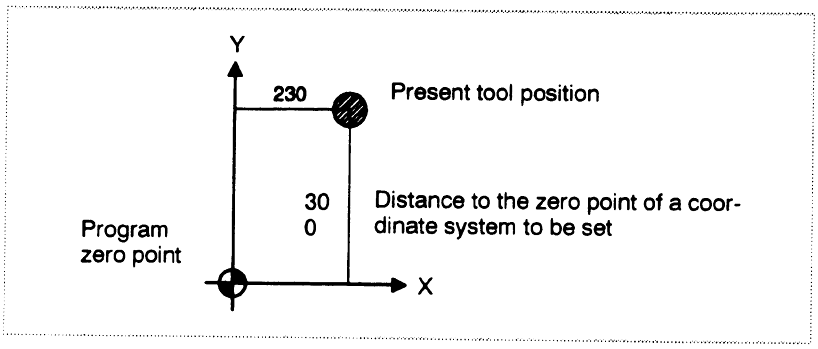


Fig. 1.3.2 (b) Coordinate system specified by the CNC

The positional relation between these two coordinate systems is determined when a workpiece is set on the table.

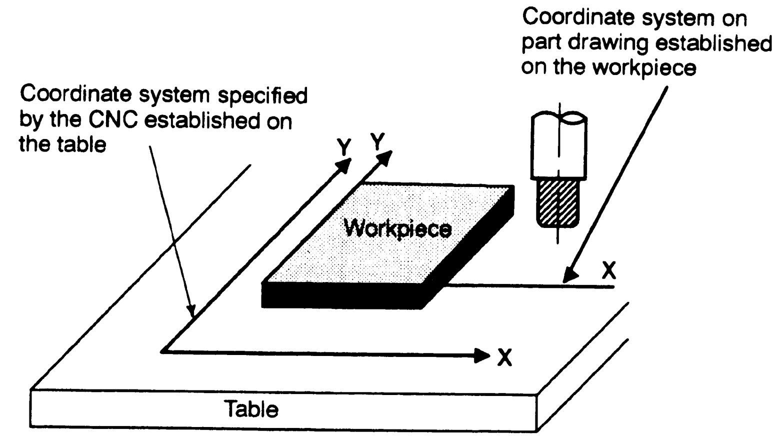


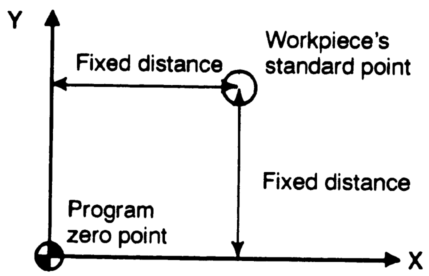
Fig. 1.3.2 (c) Coordinate system specified by CNC and coordinate systemon part drawing

The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.
Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

- **Methods of setting the two coordinate systems in the same position**

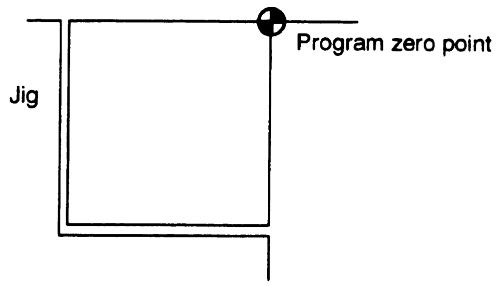
To set the two coordinate systems at the same position, simple methods shall be used according to workpiece shape, the number of machinings.

(1) Using a standard plane and point of the workpiece.



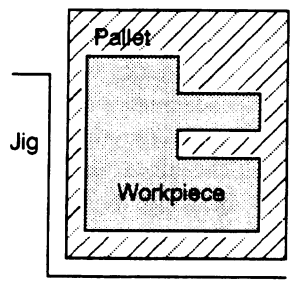
Bring the tool center to the workpiece standard point.
And set the coordinate system specified by CNC at this position.

(2) Mounting a workpiece directly against the jig



Meet the tool center to the reference position. And set the coordinate system specified by CNC at this position. (Jig shall be mounted on the predetermined point from the reference position.)

(3) Mounting a workpiece on a pallet, then mounting the workpiece and pallet on the jig



(Jig and coordinate system shall be specified by the same as (2)).

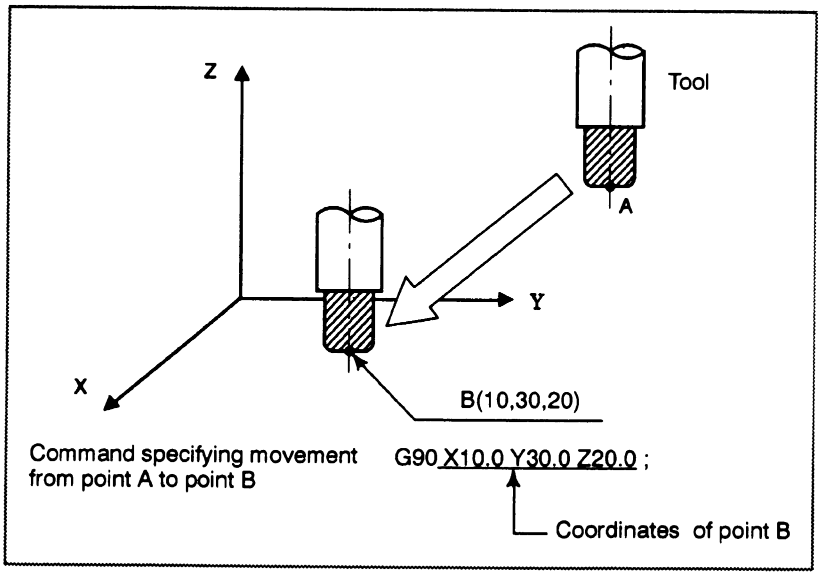
1.3.3 How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands

Explanations

Command method for moving the tool can be indicated by absolute or incremental command (See II–8.1).

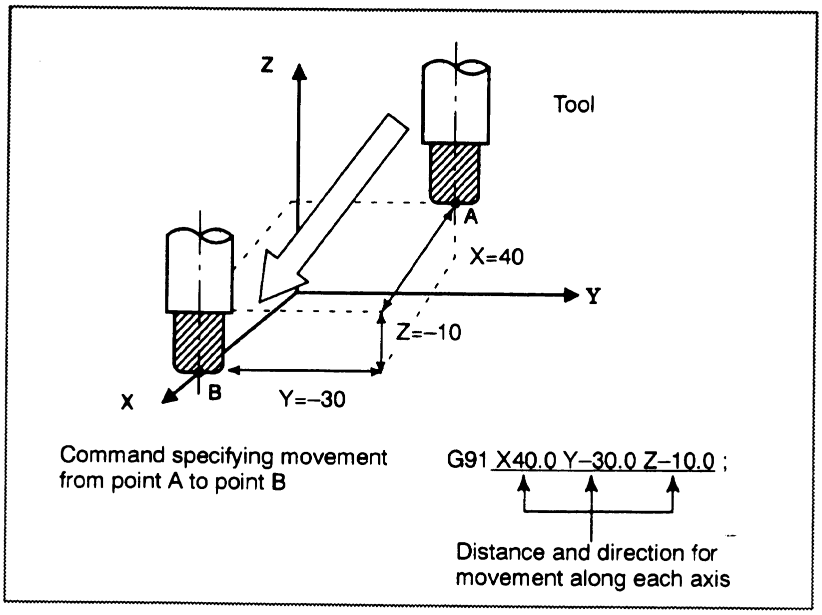
- **Absolute commands**

The tool moves to a point at "the distance from zero point of the coordinate system" that is to the moving position of the coordinate values.



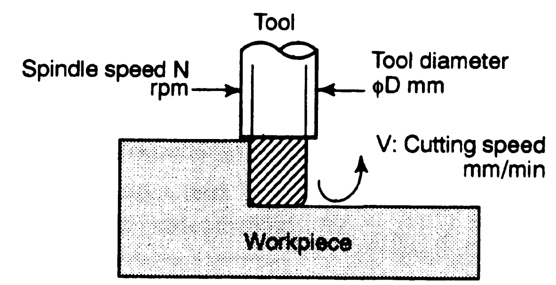
- **Incremental commands**

Specify the distance from the previous tool position to the next tool position.



1.4 CUTTING SPEED – SPINDLE SPEED FUNCTION

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed.
As for the CNC, the cutting speed can be specified by the spindle speed in rpm unit.



Examples

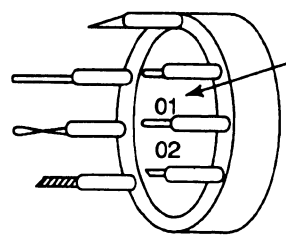
<When a workpiece should be machined with a tool 100 mm in diameter at a cutting speed of 80 mm/min. >
The spindle speed is approximately 250 rpm, which is obtained from $N=1000v/\pi D$. Hence the following command is required:

S250;

Commands related to the spindle speed are called the spindle speed function (See II–10) .

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION

When drilling, tapping, boring, milling or the like, is performed, it is necessary to select a suitable tool. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.

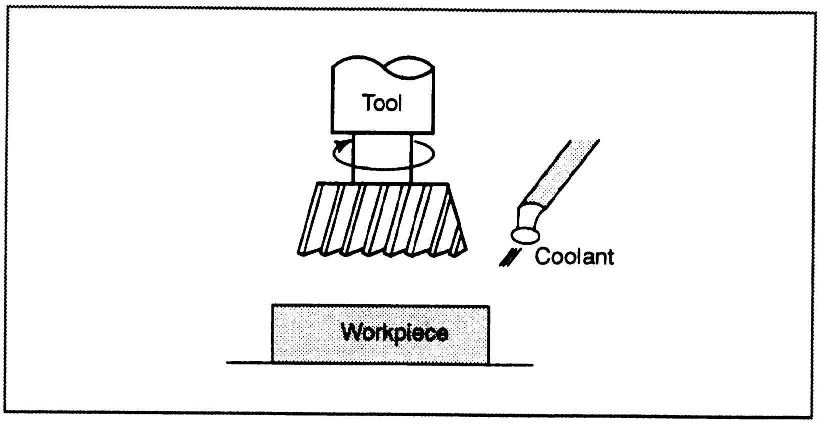


Examples

<When No.01 is assigned to a drilling tool>
When the tool is stored at location 01 in the ATC magazine, the tool can be selected by specifying T01. This is called the tool function (See II–10).

1.6 COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION

When machining is actually started, it is necessary to rotate the spindle, and feed coolant. For this purpose, on–off operations of spindle motor and coolant valve should be controlled.



The function of specifying the on–off operations of the components of the machine is called the miscellaneous function. In general, the function is specified by an M code (See II–11).
For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

1.7 PROGRAM CONFIGURATION

A group of commands given to the CNC for operating the machine is called the program. By specifying the commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off.
In the program, specify the commands in the sequence of actual tool movements.

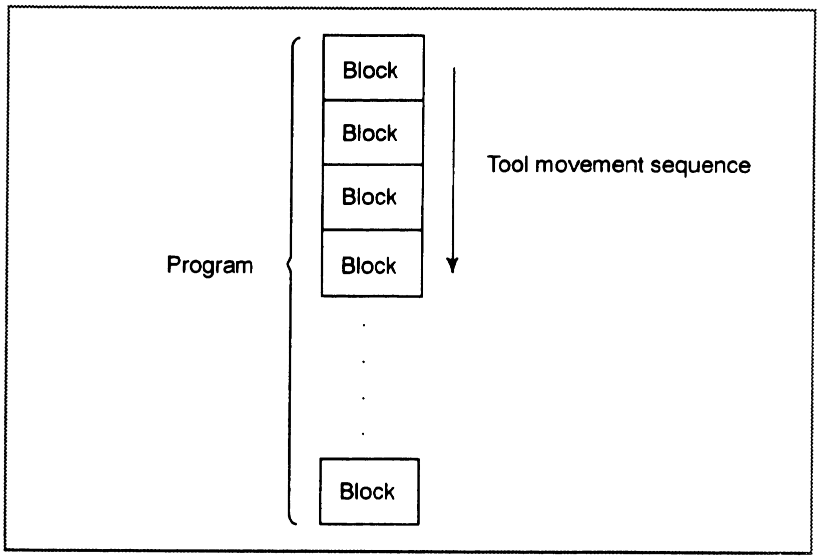


Fig. 1.7 (a) Program configuration

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number (See II–12).

Explanations

• Block

The block and the program have the following configurations.

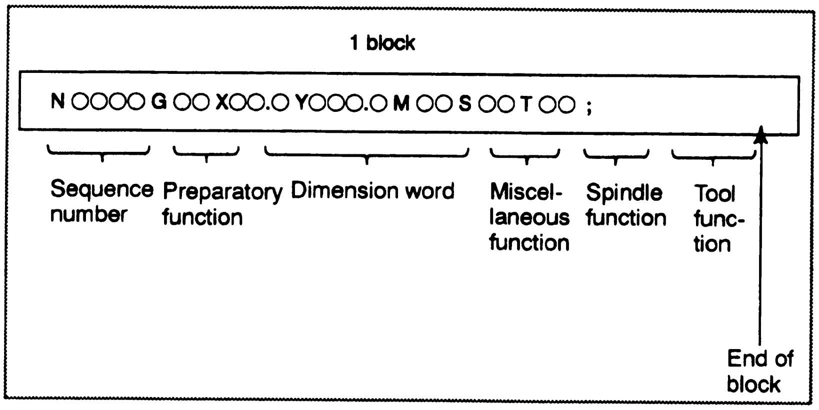


Fig. 1.7 (b) Block configuration

A block starts with a sequence number that identifies the block and ends with an end–of–block code.
This manual indicates the end–of–block code by ; (LF in the ISO code and CR in the EIA code).

• Program

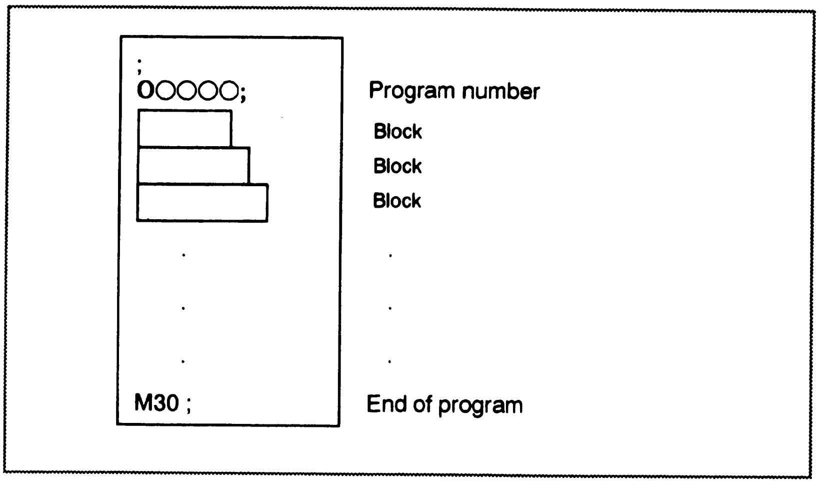
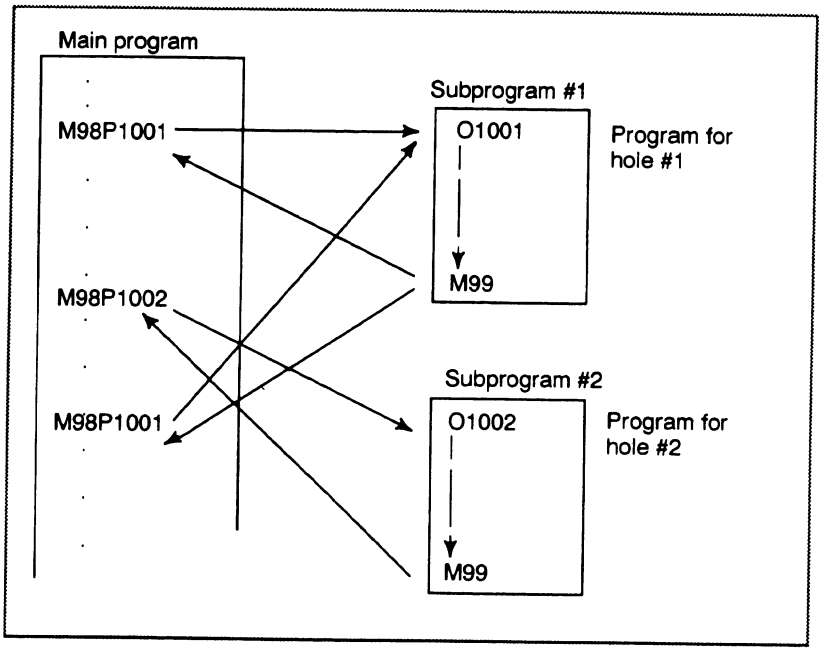


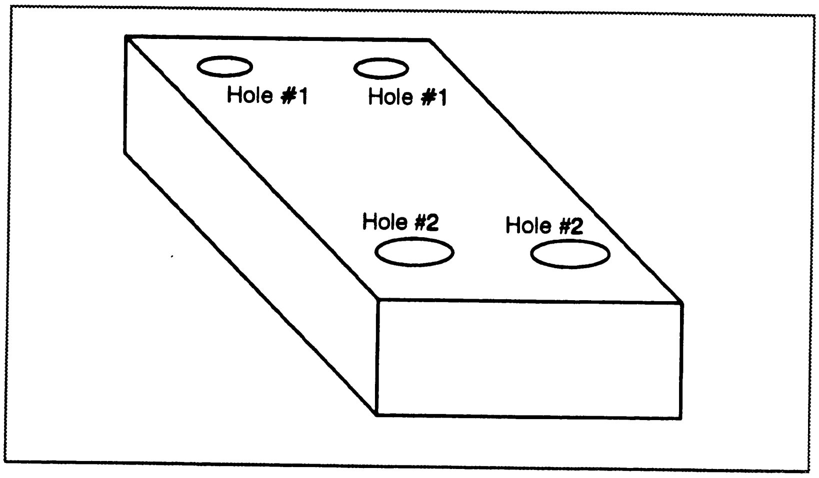
Fig. 1.7 (c) Program configuration

Normally, a program number is specified after the end–of–block (;) code at the beginning of the program, and a program end code (M02 or M30) is specified at the end of the program.

- **Main program and subprogram**

When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands of the subprogram are executed. When execution of the subprogram is finished, the sequence returns to the main program.



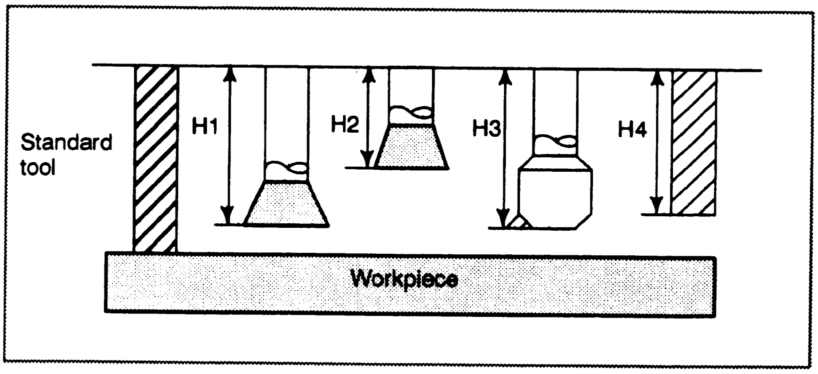


1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM

Explanations

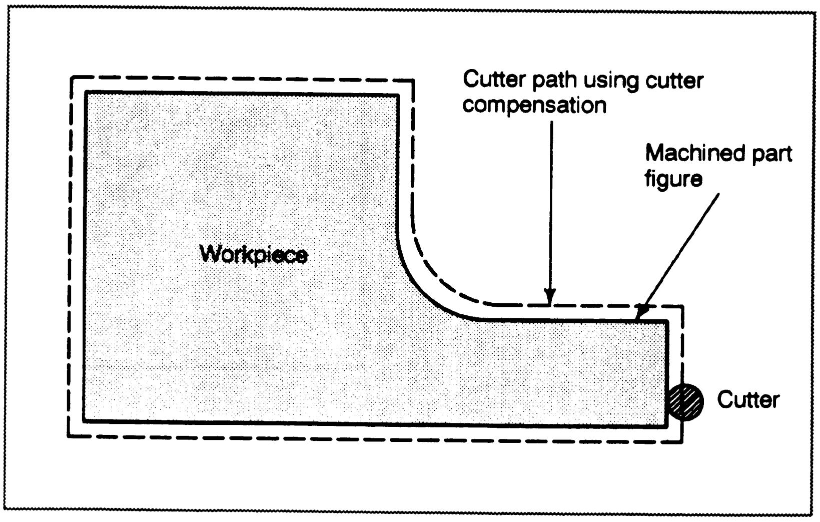
- **Machining using the end of cutter – Tool length compensation function (See II–14.1)**

Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.
Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (data display and setting : see III–11), machining can be performed without altering the program even when the tool is changed. This function is called tool length compensation.



- **Machining using the side of cutter – Cutter compensation function (See II–14.2, 14.3)**

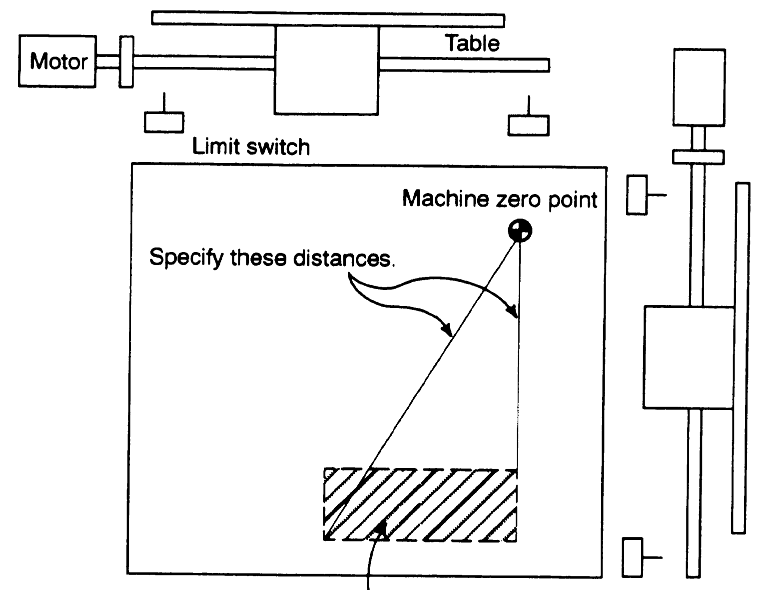
Because a cutter has a radius, the center of the cutter path goes around the workpiece with the cutter radius deviated.



If radius of cutters are stored in the CNC (Data Display and Setting : see III–11), the tool can be moved by cutter radius apart from the machining part figure. This function is called cutter compensation.

1.9 TOOL MOVEMENT RANGE – STROKE

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends. The range in which tools can move is called the stroke.



Tools cannot enter this area. The area is specified by data in memory or a program.

Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory. This function is called stroke check (see III–6.3).

2 CONTROLLED AXES

2.1 CONTROLLED AXES

Number of controlled axes	3 axes
Simultaneously controlled axes	3 axes

Note
The number of simultaneously controllable axes for manual operation (continuous manual feed, manual reference point return, or manual rapid traverse) is 1 or 3 (1 when bit 0 (JAX) of parameter 1002 is set to 0 and 3 when it is set to 1).

2.2 AXIS NAME

The name of the first, second, and third axes are fixed to X, Y, and Z respectively.

2.3 INCREMENT SYSTEM

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment in the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm or inches.

Name of increment system	Least input increment	Least command increment	Maximum stroke
IS–B	0.001mm 0.0001inch	0.001mm 0.0001inch	99999.999mm 9999.9999inch

The least command increment is either metric or inch depending on the machine tool. Set metric or inch to the parameter INM (No. 1001#0)
For selection between metric and inch for the least input increment, G code (G20 or G21) or a setting parameter selects it.
Combined use of the inch system and the metric system is not allowed. There are functions that cannot be used between axes with different unit systems (circular interpolation, cutter compensation, etc.). For the increment system, see the machine tool builder's manual.

2.4 MAXIMUM STROKE

The maximum stroke controlled by this CNC is shown in the table below:
Maximum stroke=Least command increment ±99999999

Table 2.4 (a) Maximum storoke

Increment system		Maximum strokes
IS–B	Metric machine system	±99999.999 mm
	Inch machine system	±9999.9999 inch

Notes
1. A command exceeding the maximum stroke cannot be specified.
2. The actual stroke depends on the machine tool.

3 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

Type	Meaning
One–shot G code	The G code is effective only in the block in which it is specified.
Modal G code	The G code is effective until another G code of the same group is specified.

(Example)
G01 and G00 are modal G codes in group 01.

```
G01X–;  ⎫
   Z—;  ⎬ G01 is effective in this range.
   X—;  ⎭
G00Z—;
```

Explanations

1. When the clear state (bit 6 (CLR) of parameter No. 3402) is set at power–up or reset, the modal G codes are placed in the states described below.

(1) The modal G codes are placed in the states marked with as indicated in Table 3.

(2) G20 and G21 remain unchanged when the clear state is set at power–up or reset.(3)

(3) The user can select whether G22 or G23 is set at power–up by setting bit 7 (G23) of parameter No. 3402 accordingly. G22 and G23 remain as is when the clear state is set upon reset.

(4) The user can select G00 or G01 by setting bit 0 (G01) of parameter No. 3402.

(5) The user can select G90 or G91 by setting bit 3 (G91) of parameter No. 3402.

(6) The user can select G17, G18, or G19 by setting bit 1 (G18) and bit 2 (G19) of parameter No. 3402.

2. G codes other than G10 and G11 are one–shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm No. 010 is output.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. If a G code belonging to group 01 is specified in a canned cycle, the canned cycle is cancelled. This means that the same state set by specifying G80 is set. Note that the G codes in group 01 are not affected by a G code specifying a canned cycle.
6. G codes are indicated by group.

Table 3 G code list (1/3)

G code	Group	Function
G00	01	Positioning
G01		Linear interpolation
G02		Circular interpolation/Helical interpolation CW
G03		Circular interpolation/Helical interpolation CCW
G04	00	Dwell, Exact stop
G09		Exact stop
G10		Data setting
G11		Data setting mode cancel
G17	02	XY plane selection
G18		ZX plane selection
G19		YZ plane selection
G20	06	Input in inch
G21		Input in mm
G22	04	Stored stroke check function on
G23		Stored stroke check function off
G27	00	Reference position return check
G28		Return to reference position
G29		Return from reference position
G30		2nd reference position return
G31		Skip function

Table 3 G code list (2/3)

G code	Group	Function
G40	07	Cutter compensation cancel
G41		Cutter compensation left
G42		Cutter compensation right
G43	08	Tool length compensation + direction
G44		Tool length compensation – direction
G49	08	Tool length compensation cancel
G50	11	Scaling cancel
G51		Scaling
G52	00	Local coordinate system setting
G53		Machine coordinate system selection
G54	14	Workpiece coordinate system 1 selection
G55		Workpiece coordinate system 2 selection
G56		Workpiece coordinate system 3 selection
G57		Workpiece coordinate system 4 selection
G58		Workpiece coordinate system 5 selection
G59		Workpiece coordinate system 6 selection
G60	00	Single direction positioning
G61	15	Exact stop mode
G63		Tapping mode
G64		Cutting mode

Table 3 G code list (3/3)

G code	Group	Function
G65	00	Macro call
G66	12	Macro modal call
G67		Macro modal call cancel
G68	16	Coordinate rotation
G69		Coordinate rotation cancel
G73	09	Peck drilling cycle
G74		Counter tapping cycle
G76	09	Fine boring cycle
G80	09	Canned cycle cancel/external operation function cancel
G81		Drilling cycle, spot boring cycle or external operation function
G82		Drilling cycle or counter boring cycle
G83		Peck drilling cycle
G84		Tapping cycle
G85		Boring cycle
G86		Boring cycle
G87		Back boring cycle
G88		Boring cycle
G89		Boring cycle
G90	03	Absolute command
G91		Increment command
G92	00	Setting for work coordinate system or clamp at maximum spindle speed
G94	05	Feed per minute
G98	10	Return to initial point in canned cycle
G99		Return to R point in canned cycle

4 INTERPOLATION FUNCTIONS

4.1 POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.
In the absolute command, coordinate value of the end point is programmed.
In the incremental command the distance the tool moves is programmed.

Format

G00IP_;

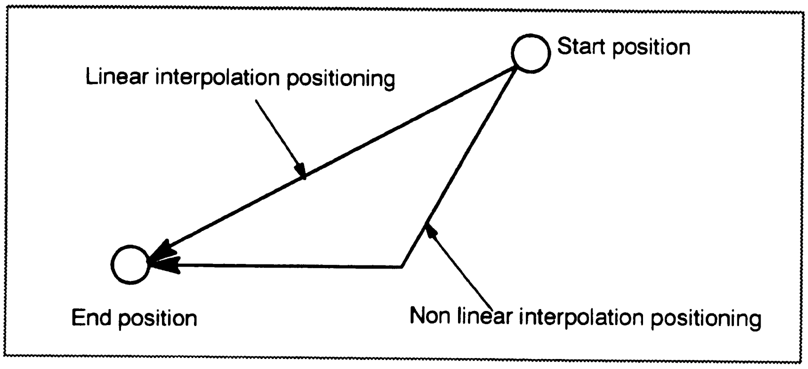
IP_: For an absolute command, the coordinates of an end position, and for an incremental commnad, the distance the tool moves.

Explanations

Either of the following tool paths can be selected according to bit 1 of parameter LRP (No. 1401).

- **Nonlinear interpolation positioning**
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.

- **Linear interpolation positioning**
  The tool path is the same as in linear interpolation (G01). The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



The rapid traverse rate in G00 command is set to the parameter No. 1420 for each axis independently by the machine tool builder. In the posiitoning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in–position.
"In–position " means that the feed motor is within the specified range. This range is determined by the machine tool builder by setting to parameter No. 1826. The user can disable in–position check for each block by setting bit 5 (NCI) of parameter No. 1601 accordingly.

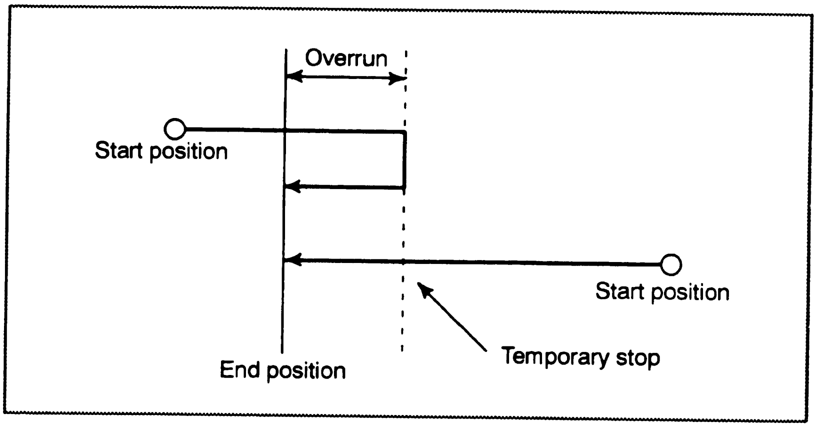
Limitations

The rapid traverse rate cannot be specified in the address F.
Even if linear interpolation positioning is specified, nonlinear interpolation positioning is used in the following cases. Therefore, be careful to ensure that the tool does not foul the workpiece.

- G28 specifying positioning between the reference and intermediate positions.
- G53

4.2 SINGLE DIRECTION POSITIONING (G60)

For accurate positioning without play of the machine (backlash), final positioning from one direction is available.



Format

G60 P_;

P_ : For an absolute command, the coordinates of an end position, and for an incremental commnad, the distance the tool moves.

Explanations

An overrun and a positioning direction are set by the parameter (No. 5440). Even when a commanded positioning direction coincides with that set by the parameter, the tool stops once before the end point.
G60, which is an one–shot G–code, can be used as a modal G–code in group 01 by setting 1 to the parameter MDL (No. 5431 bit 0).
One–shot G codes other than G60 can be used in unidirectional positioning mode.
This setting can eliminate specifying a G60 command for every block.
Other specifications are the same as those for an one–shot G60 command.
When an one–shot G code is sepcified in the single direction positioning mode, the one–shot G command is effective like G codes in group 01.

Examples

When one–shot G60 commands are used.		**When modal G60 command is used.**	
⋮		⋮	
G90;		G90G60;	Single direction positioning mode start
G60 X0Y0;	Single direction positioning	X0Y0;	Single direction positioning
G60 X100;		X100;	
G60 Y100;		Y100;	
G04 X10;		G04X10;	Dwell
G00 X0Y0;		G00X0Y0;	Single direction positioning mode cancel
⋮		⋮	

Restrictions

- During drilling canned cycle, no single direction positioning is effected in Z axis.
- No single direction positioning is effected in an axis for which no overrun has been set by the parameter.
- When the move distance 0 is commanded, the single direction positioning is not performed.
- The direction set to the parameter is not effected by mirror image.
- The single direction positioning does not apply to the shift motion in the canned cycles of G76 and G87.

4.3 LINEAR INTERPOLATION (G01)

Tools can move along a line.

Format

G01 IP_F_;

IP_:For an absolute command, the coordinates of an end point , and for an incremental commnad, the distance the tool moves.

F_:Speed of tool feed (Feedrate)

Explanations

A tools move along a line to the specified position at the feedrate specified in F.
The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.
The feedrate commanded by the F code is measured along the tool path.
If the F code is not commanded, the feedrate is regarded as zero.
The feedrate of each axis direction is as follows.

G01$\alpha\underline{\alpha}\beta\underline{\beta}\gamma\underline{\gamma}F\underline{f}$;

Feed rate of α axis direction : $F\alpha = \frac{\alpha}{L} \times f$

Feed rate of Β axis direction : $F_\beta = \frac{\beta}{L} \times f$

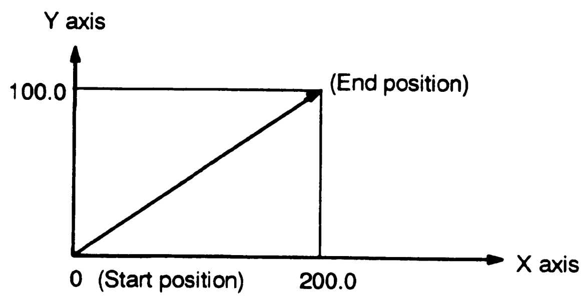
Feed rate of Γ axis direction : $F\gamma = \frac{\gamma}{L} \times f$

$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2}$

Examples

- **Linear interpolation**

(G91) G01X200.0Y100.0F200.0 ;



4.4 CIRCULAR INTERPOLATION (G02,G03)

The command below will move a tool along a circular arc.

Format

Arc in the XpYp plane

$$G17\left\{\begin{matrix}G02\\G03\end{matrix}\right\}\ X\_Y\_\ \left\{\begin{matrix}I\_J\_\\R\_\end{matrix}\right\}\ F\_;$$

Arc in the ZpXp plane

$$G18\left\{\begin{matrix}G02\\G03\end{matrix}\right\}\ X\_\ \left\{\begin{matrix}I\_K\_\\R\_\end{matrix}\right\}\ F\_$$

Arc in the YpZp plane

$$G19\left\{\begin{matrix}G02\\G03\end{matrix}\right\}\ Y\_Z\_\ \left\{\begin{matrix}J\_K\_\\R\_\end{matrix}\right\}\ F\_$$

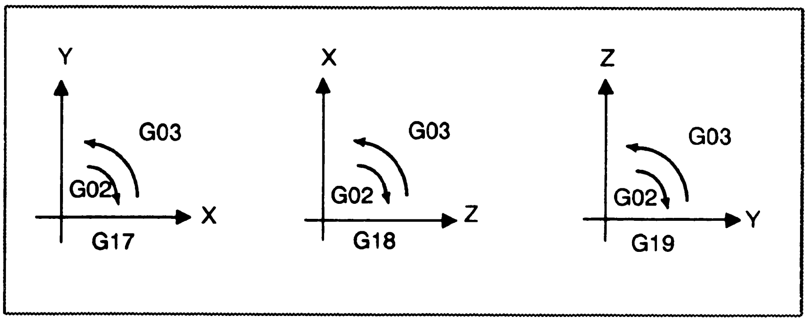
Table.4.4 Description of the Command Format

Command	Description
G17	Specification of arc on XY plane
G18	Specification of arc on ZX plane
G19	Specification of arc on YZ plane
G02	Circular Interpolation Clockwise direction (CW)
G03	Circular Interpolation Counterclockwise direction (CCW)
X_	Command values of X axis
Y_	Command values of Y axis (set by parameter No. 1022)
Z_	Command values of Z axis or its parallel axis (set by parameter No. 1022)
I_	X axis distance from the start point to the center of an arc with sign
J_	Y axis distance from the start point to the center of an arc with sign
k_	Z axis distance from the start point to the center of an arc with sign
R_	Arc radius with sign
F_	Feedrate along the arc

Explanations

• Direction of the circular interpolation

”Clockwise”(G02) and ”counterclockwise”(G03) on the XY plane (ZX plane or YZ plane) are defined when the XY plane is viewed in the positive–to–negative direction of the Z axis (Y axis or X axis, respectively) in the Cartesian coordinate system. See the figure below.

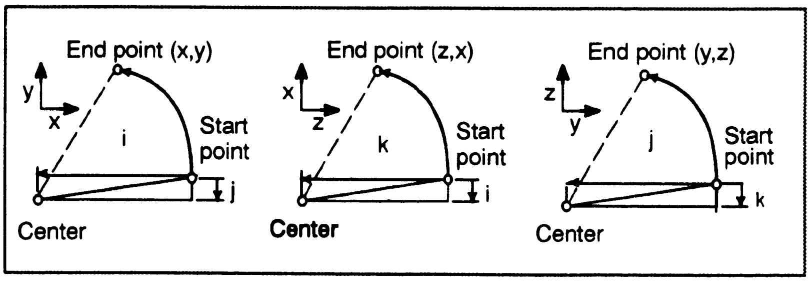


• Distance moved on an arc

The end point of an arc is specified by address X, Y or Z and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified.

• Distance from the start point to the center of arc

The arc center is specified by addresses I, J, and K for the X, Y, and Z axes, respectively. The numerical value following I, J, or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.
I, J, and K must be signed according to the direction.



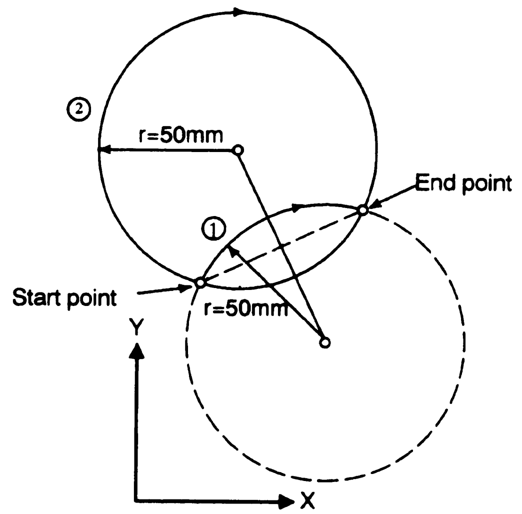
I0,J0, and K0 can be omitted. When X, Y, and Z are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360°-arc (circle) is specified.
G02I_; Command for a circle
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm (No.024) occurs.

• Arc radius

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K. In this case, one arc is less than 180°, and the other is more than 180° are considered. When an arc exceeding 180° is commanded, the radius must be specified with a negative value. If X, Y, and Z are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0° is programmed

G02R ; (The cutter does not move.)

For arc (1)(less than 180°)
G91 G02 X_p60.0 Y_p20.0 R50.0 F300.0 ;
For arc (2)(greater than 180°)
G91 G02 X_p60.0 Y_p20.0 R–50.0 F300.0 ;



• Feedrate

The feedrate in circular interpolation is equal to the feed rate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.

The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feed rate is measured along the arc after the cutter compensation is applied

Restrictions

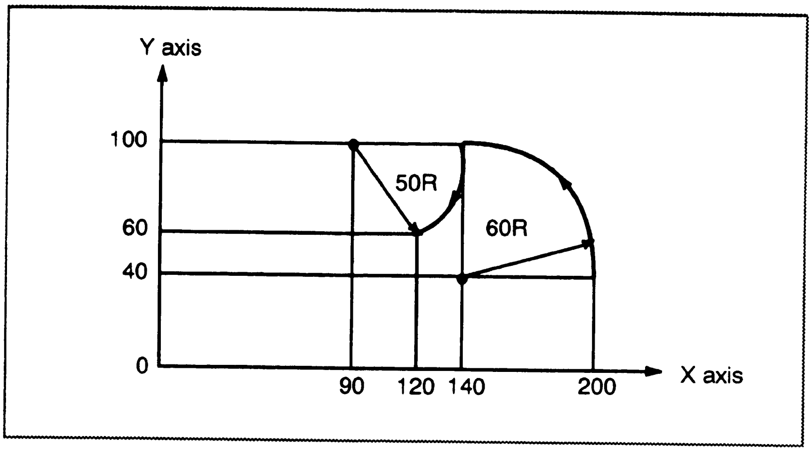
If I, J, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.

If an axis not comprising the specified plane is commanded, an alarm is displayed.

For example, if Z axis is specified when plane XY is specified, an alarm (No.028)is displayed.

When an arc having a center angle close to 180° is specified using its radius R, the system may fail to calculate the center of the arc correctly. Therefore, specify the arc with I, J, and K.

Examples



The above tool path can be programmed as follows ;

(1) In absolute programming

G92X200.0 Y40.0 Z0 ;
G90 G03 X140.0 Y100.0R60.0 F300.;
G02 X120.0 Y60.0R50.0 ;
or
G92X200.0 Y40.0Z0 ;
G90 G03 X140.0 Y100.0I–60.0 F300.;
G02 X120.0 Y60.0I–50.0 ;

(2) In incremental programming

G91 G03 X–60.0 Y60.0 R60.0 F300.;
G02 X–20.0 Y–40.0 R50.0 ;
or
G91 G03 X–60.0 Y60.0 I–60.0 F300. ;
G02 X–20.0 Y–40.0 I–50.0 ;

4.5 HELICAL INTERPOLATION (G02,G03)

The tool can move helically when herical interpolation is applied this being a combination of circular interpolation and movement along axis that is not on the plane for circular interpolation.

Format

Synchronously with arc of XpYp plane

$$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_Y\_ \begin{Bmatrix} I\_J\_ \\ R\_ \end{Bmatrix} \alpha\_F\_;$$

Synchronously with arc of ZpXp plane

$$G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_Z\_ \begin{Bmatrix} I\_K\_ \\ R\_ \end{Bmatrix} \alpha\_F\_;$$

Synchronously with arc of YpZp plane

$$G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_Z\_ \begin{Bmatrix} J\_K\_ \\ R\_ \end{Bmatrix} \alpha\_F\_;$$

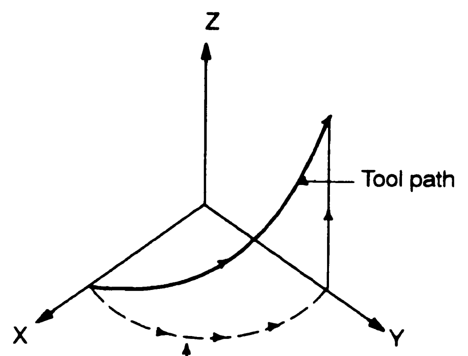
α : Any one axis where circular interpolation is not applied.

Explanations

The command method is to simply add a move command axis which is not circular interpolation axes. An F command specifies a feed rate along a circular arc. Therefore, the feed rate of the linear axis is as follows:

$$F \times \frac{\text{Length of linear axis}}{\text{Length of circular arc}}$$

Determine the feed rate so the linear axis feed rate does not exceed any of the various limit values.Bit 0 (HFC) of parameter No. 1404 can be used to prevent the linear axis feedrate from exceeding various limit values.



The feedrate along the circumference of two circular interpolated axes is the specified feedrate.

Restrictions

·Cutter compensation is applied only for a circular arc.
·Tool offset cannot be used in a block in which a helical cutting is commanded.

4.6 SKIP FUNCTION(G31)

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.

Format

G31 P_ ;

G31: One–shot G code (If is effective only in the block in which it is specified)

Explanations

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #5061 to #5063, as follows:

#5061	X axis coordinate value
#5062	Y axis coordinate value
#5063	Z axis coordinate value

Notes

1. If G31 command is issued while cutter compensation C is applied, an P/S alarm of No.035 is displayed. Cancel the cutter compensation with the G40 command before the G31 command is specified.
2. During movement of the tool triggered by the skip function, feedrate override, drg run, and automatic acceleration/deceleration are disabled. This is intended to improve the tool positioning accuracy when a skip signal is input, However, they can be enabled by bit 7 (SKF) of parameter No. 6200.

Examples

- **The next block to G31 is an incremental command**

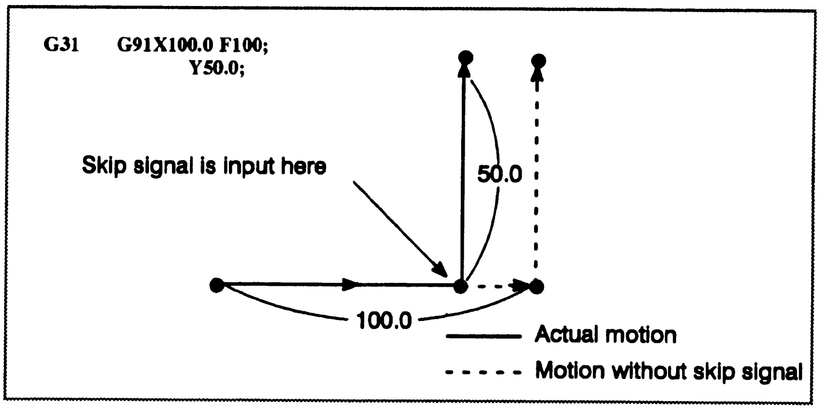


Fig.4.6 (a) The next block is an incremental command

- **The next block to G31 is an absolute command for 1 axis**

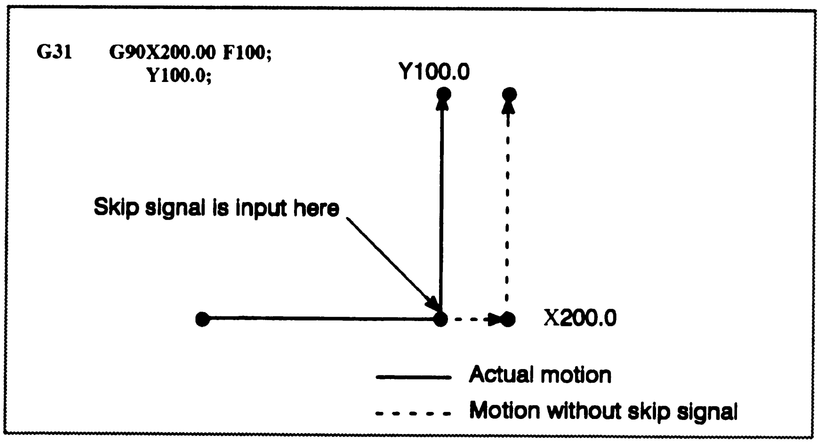


Fig.4.6 (b) The next block is an absolute command for 1 axis

- **The next block to G31 is an absolute command for 2 axes**

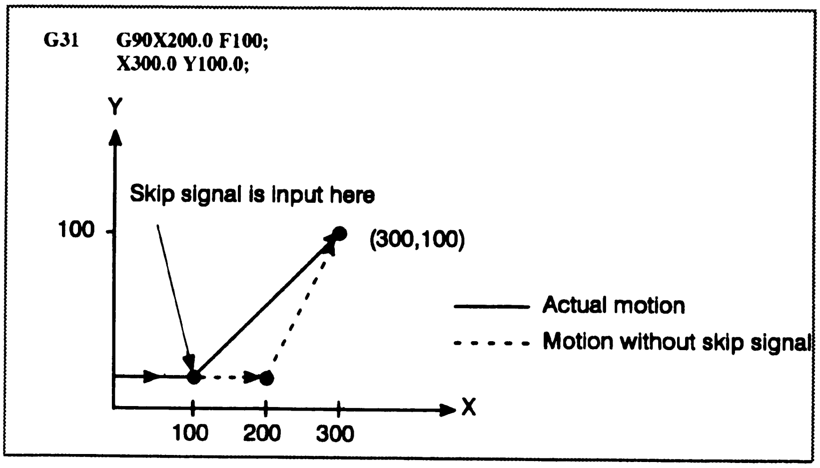


Fig 4.6 (c) The next block is an absolute command for 2 axes

5 FEED FUNCTIONS

5.1 GENERAL

The feed functions control the feedrate of the tool. The following two feed functions are available:

• Feed functions

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse feedrate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

• Override

Override can be applied to a rapid traverse rate or cutting feedrate using the switch on the machine operator's panel.

• Automatic acceleration/ deceleration

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement.

5.2 RAPID TRAVERSE

Format

G00 IP_ ;

G00 : G code (group 01) for positioning (rapid traverse)
IP_ ; Dimension word for the end point

Explanations

The positioning command (G00) positions the tool by rapid traverse. In rapid traverse, the next block is executed after the specified feedrate becomes 0 and the servo motor reaches a certain range set by the machine tool builder (in–position check).
In–position check for each block can be skipped by setting NC1, bit 5 of parameter No. 1601, to 1.
A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse feedrate need be programmed.
The following overrides can be applied to a rapid traverse rate with the switch on the machine operator's panel:F0, 25, 50, 100%
F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.
For detailed information, refer to the appropriate manual of the machine tool builder.

- **Specifiable range of rapid traverse**

	Specifiable range of rapid traverse
mm SYSTEM	30–240,000 mm/min
inch SYSTEM	30–9,600 inch/min

5.3 CUTTING FEED

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are commanded with numbers after the F code.
In cutting feed, the next block is executed so that the feedrate change from the previous block is minimized.

Format

Feed per minute
G94 ; G code (group 05) for feed per minute
F_ ; Feedrate command (mm/min or inch/min)

Explanations

- **Tangential speed constant control**

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

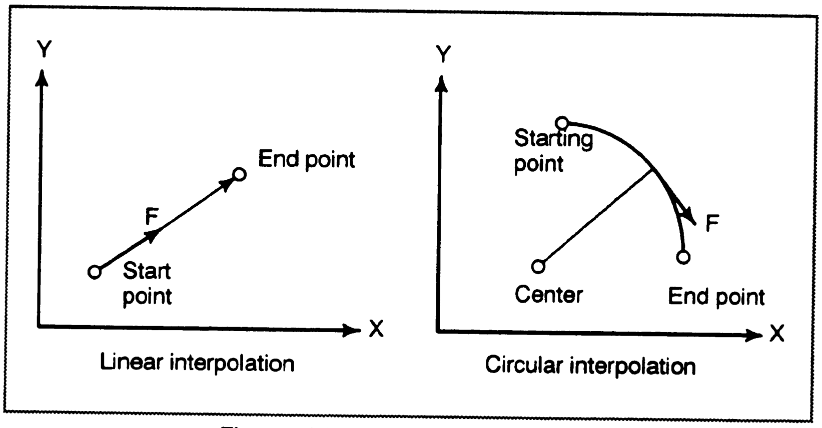


Fig. 5.3 (a) Tangential feedrate (F)

- **Feed per minute (G94)**

The amount of feed of the tool per minute is to be directly specified by setting a number after F in the feed per minute mode G94. At power–on, feed per minute mode (G94) is set. The user usually need not specify the G94 command.
An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see appropriate manual of the machine tool builder.

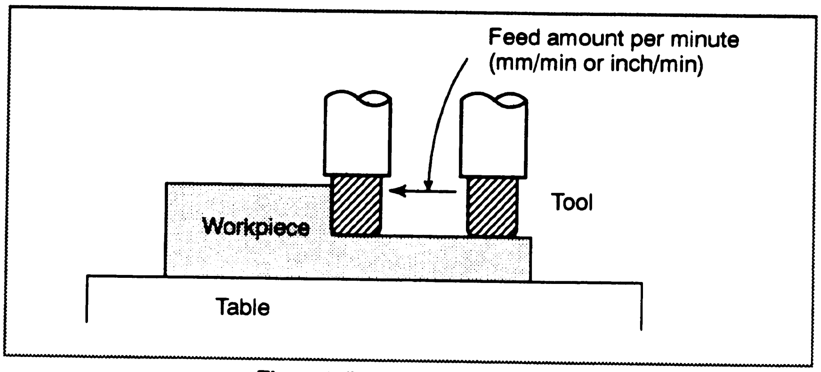


Fig. 5.3 (b) Feed per minute

Note
No override can be used for some commands such as for tapping cycly (G84).

• Cutting feedrate clamp

A common upper limit can be set on the cutting feedrate along each axis with parameter No. 1422. If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit. Parameter No. 1430 can be used to specify the maximum cutting feedrate for each axis only for linear interpolation and circular interpolation. When the cutting feedrate along an axis exceeds the maximum feedrate for the axis as a result of interpolation, the cutting feedrate is clamped to the maximum feedrate.

> **Note**
> CNC calculation may involve a feedrate error of ±2% with respect to a specified value. However, this is not true for acceleration/deceleration. To be more specific, this error is calculated with respect to a measurement on the time the tool takes to move 500 mm or more during the steady state:

• Reference

See Appendix C for a range of feedrates that can be specified.

5.4 CUTTING FEEDRATE CONTROL

Cutting feedrate can be controlled, as indicated in Table 5.4(a).

Table 5.4(a) Cutting Feedrate Control

Function name	G code	Validity of G code	Description
Exact stop	G09	This function is valid for specified blocks only.	The tool is decelerated at the end point of a block, then an in–position check is made. Then the next block is executed.
Exact stop mode	G61	Once specified, this function is valid until G63 or G64 is specified.	The tool is decelerated at the end point of a block, then an in–position check is made. Then the next block is executed.
Cutting mode	G64	Once specified, this function is valid until G61 or G63 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed.
Tapping mode	G63	Once specified, this function is valid until G61 or G64 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed. When G63 is specified, feedrate override and feed hold are invalid.

Note
1. The purpose of in–position check is to check that the servo motor has reached within a specified range (specified with a parameter by the machine tool builder).
   In–position check is not performed when bit 5 (NCI) of parameter No. 1601 is set to 1.

Format

Exact stop	G09 IP_ ;
Exact stop mode	G61 ;
Cutting mode	G64 ;
Tapping mode	G63 ;

5.4.1 Exact Stop (G09, G61) Cutting Mode (G64) Tapping Mode (G63)

Explanations

The inter–block paths followed by the tool in the exact stop mode, cutting mode, and tapping mode are different (Fig. 5.4.1 (a)).

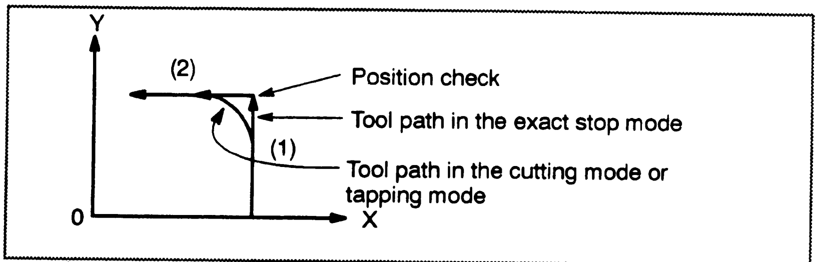


Fig. 5.4.1 (a) Example of Tool Paths from Block (1) to Block (2)

> **Note**
> The cutting mode (G64 mode) is set at power–on or system clear.

5.5 DWELL (G04)

Format

Dwell	**G04 X_** ; or **G04 P_** ; **X_** : Specify a time (decimal point permitted) **P_** : Specify a time (decimal point not permitted)

Explanations

By specifying a dwell, the execution of the next block is delayed by the specified time. In addition, a dwell can be specified to make an exact check in the cutting mode (G64 mode).
When neither P nor X is specified, exact stop is performed.

Table 5.5 (a) Command value range of the dwell time (Command by X)

Increment system	Command value range	Dwell time unit
IS–B	0.001A99999.999	s

Table 5.5 (b) Command value range of the dwell time (Command by P)

Increment system	Command value range	Dwell time unit
IS–B	1A99999999	0.001 s

5.6 AUTOMATIC ACCELERATION/ DECELERATION

5.6.1 Automatic Acceleration/ Deceleration

General

- **Automatic acceleration/ deceleration**

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 5.6.1 (a)).

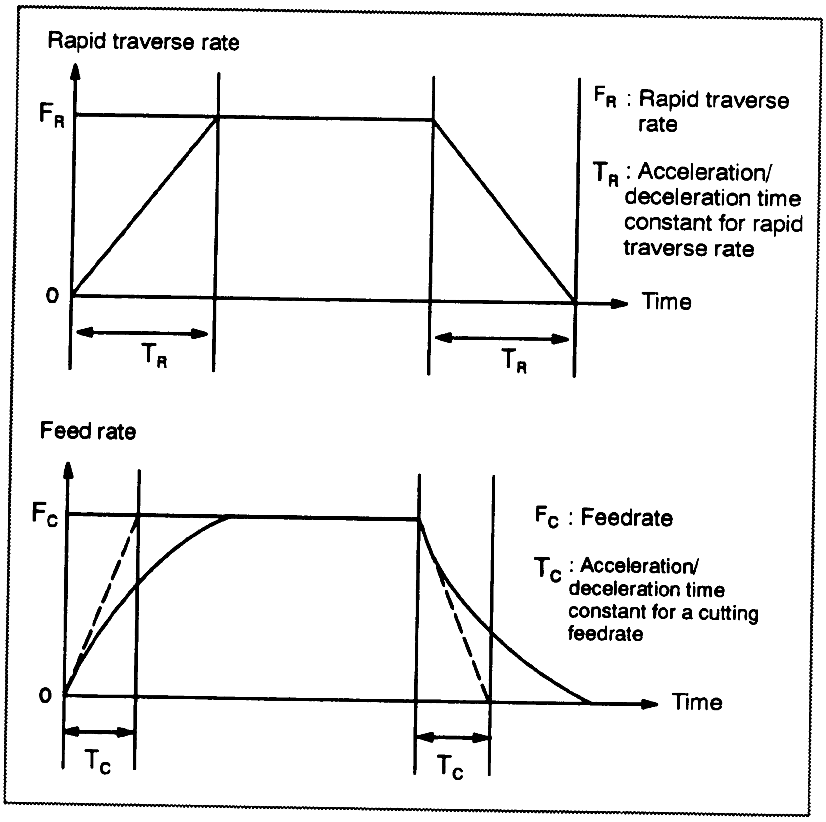


Fig. 5.6.1 (a) Automatic acceleration/deceleration (example)

Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.
Automatic acceleration/deceleration is also performed when feedrate changes, so change in speed is also smoothly done.
It is not necessary to take acceleration/deceleration into consideration when programming.

Rapid traverse: Linear acceleration/deceleration (time constant per axis is set by parameter 1620)

- **Tool path in a cutting feed**

If the direction of movement changes between specified blocks during cutting feed, a rounded–corner path may result (Fig 5.6.1 (b)).

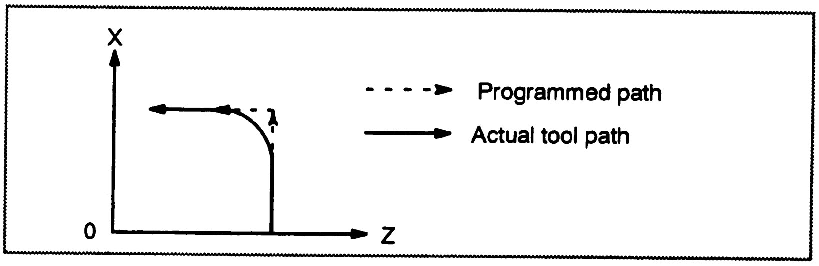


Fig. 5.6.1 (b) Example of Tool Path between Two Blocks

In circular interpolation, a radial error occurs (Fig. 5.6.1 (c)).

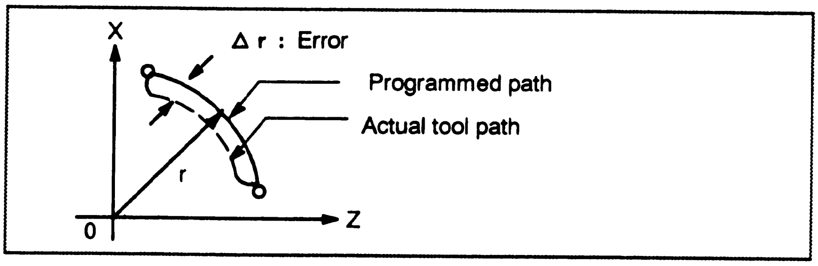


Fig. 5.6.1 (c) Example of Radial Error in Circular Interpolation

The rounded–corner path shown in Fig. 5.6.1 (b) and thee error shown in Fig. 5.6.1 (c) depend on the feedrate. So, the feedrate needs to be controlled for the tool to move as programmed.

5.6.2 Linear Acceleration/ Deceleration after Interpolation for Cutting Feed

General

If linear acceleration/deceleration after interpolation for cutting feed is enabled (bit 0 of parameter No. 1610, CTL), acceleration/deceleration is performed as follows:

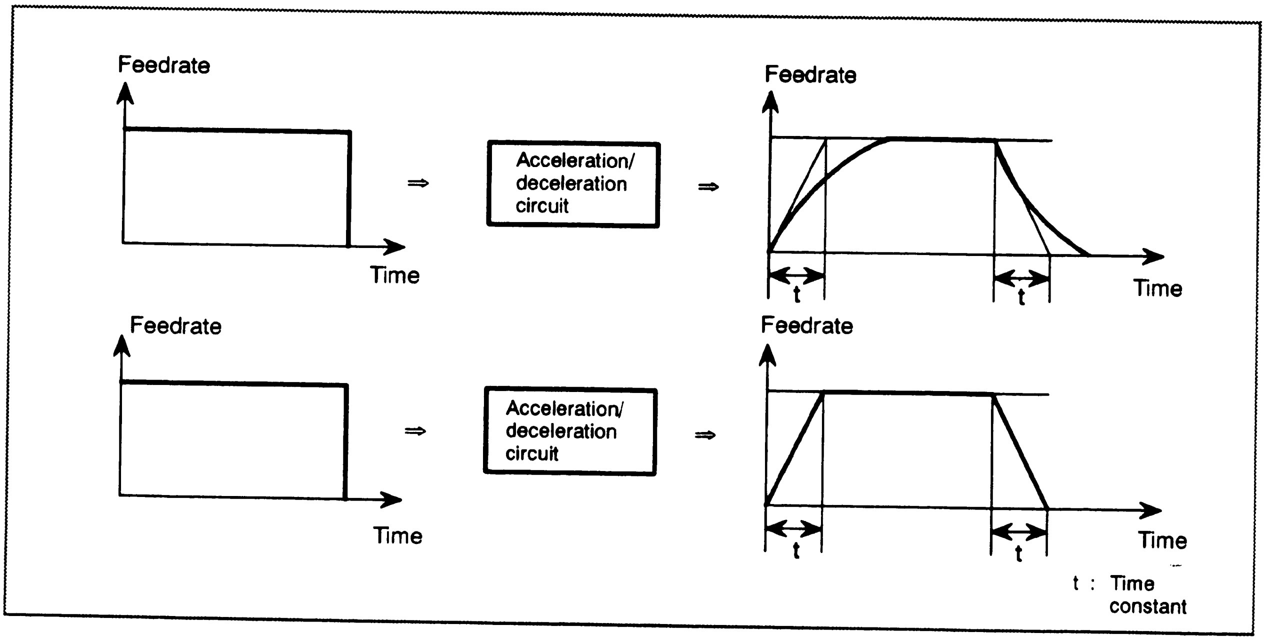
Cutting feed: Linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed: Exponential or linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.

If an identical time constant is specified, linear acceleration/deceleration can halve the delay relative to the programmed time, in comparison with exponential acceleration/deceleration, thus reducing the time needed for acceleration and deceleration. If circular interpolation is performed, especially when high–speed cutting is being performed, the actual tool path created after acceleration/deceleration will deviate from the programmed arc in the radial direction. This deviation can also be reduced, in comparison with exponential acceleration/deceleration, by applying linear acceleration/deceleration.

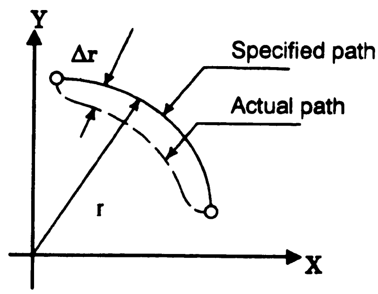


Linear acceleration/deceleration after interpolation for cutting feed is an optional function. The time constants for cutting feed and fog feed for each axis are specified in parameter Nos. 1622 and 1624 respectively, in the same way as for exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Note

If the optional function for linear acceleration/deceleration after interpolation for cutting feed is not provided, exponential acceleration/deceleration is always selected, irrespective of the setting.

- If linear acceleration/deceleration after interpolation for cutting feed is enabled, linear acceleration/deceleration is executed during cutting feed and during a dry run. Linear acceleration/deceleration can also be executed during jog feed if the JGL (bit 4 of parameter No. 1610) is specified accordingly.
- In circular interpolation especially when circular cutting is executed at high speed, the actual path of the accelerated or decelerated tool deviates from the specified arc in the direction of the radius.



Δr : Maximum radius error (mm)
v : Feedrate (mm/s)
r : Radius of arc (mm)
T_1 : Acceleration/deceleration time constant (s)
T_2 : Time constant of servo loop (s)
(Reciprocal of position loop gain)

The maximum error in the radial direction (Δr) can be approximated by the following expressions:

$$\Delta r = \left(\frac{1}{2}T_1^2+\frac{1}{2}T_2^2\right)\frac{v^2}{r} \quad \cdots\cdots\cdots\cdots\cdots \text{Exponential acceleration/deceleration}$$

$$\Delta r = \left(\frac{1}{24}T_1^2+\frac{1}{2}T_2^2\right)\frac{v^2}{r} \quad \cdots\cdots\cdots\cdots\cdots \text{Linear acceleration/deceleration after interpolation}$$

If the error caused by the time constant of the servo loop is excluded, the error cause by linear acceleration/deceleration after interpolation is 1/12 of that caused by exponential acceleration/deceleration.

- Linear acceleration/deceleration can be executed both for cutting feed and for jog feed along a PMC axis. Acceleration/deceleration for cutting feed is executed even if acceleration/deceleration for jog feed is selected. In jog feed along the PMC axis, the time constant for cutting feed is used instead of that for jog feed.

6 REFERENCE POSITION

General

• Reference position

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function.
For example, the reference position is used as a position at which tools are automatically changed. Up to two reference positions can be specified by setting coordinates in the machine coordinate system in parameters (No. 1240, 1241).

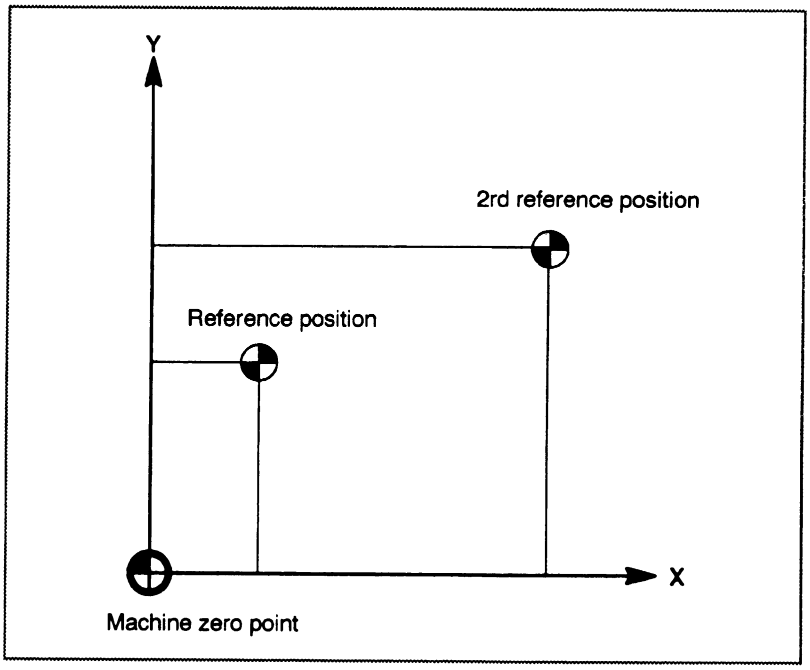


Fig. 6 (a) Machine zero point and reference positions

- **Reference position return and movement from the reference position**

Tools are automatically moved to the reference position via an intermediate position along a specified axis. Or, tools are automatically moved from the reference position to a specified position via an intermediate position along a specified axis. When reference position return is completed, the lamp for indicating the completion of return goes on.

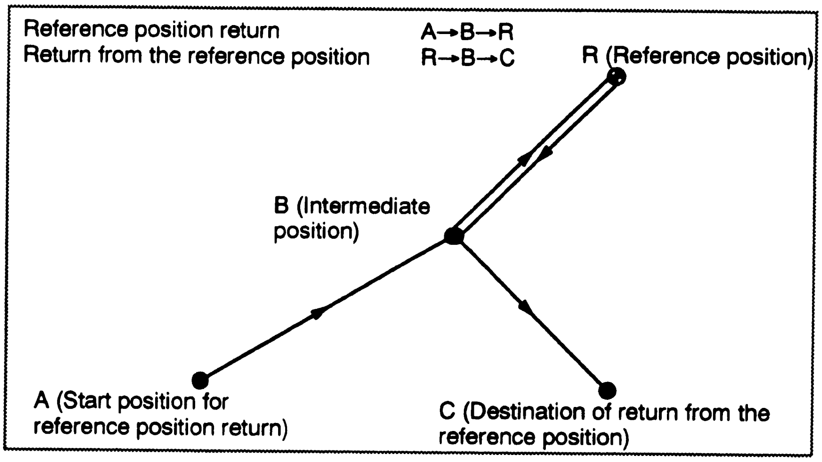


Fig. 6 (b) Reference position return and return form the reference position

- **Reference position return check**

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis goes on.

Format

- **Reference position return**

G28 IP_ ; Reference position return

G30 P2 IP_ ;2nd reference position return (P2 can be omitted.)

IP : Command specifying the intermediate position (Absolute/incremental command)

- **Return from reference position**

G29 IP_ ;

IP : Command specifying the destination of return from reference position (Absolute/incremental command)

- **Reference position return check**

G27 IP_ ;

IP : Command specifying the reference position (Absolute/incremental command)

Explanations

• Reference position return (G28)

Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis.
Therefore, for safety, the cutter compensation, and tool length compensation should be cancelled before executing this command.
The coordinates for the intermediate position are stored in the CNC only for the axes for which a value is specified in a G28 block. For the other axes, the previously specified coordinates are used.

Example N1 G28 X40.0 ; Intermediate position (X40.0)
N2 G28 Y60.0 ; Intermediate position (X40.0, Y60.0)

• 2nd reference position return (G30)

In a system without an absolute–position detector, the second reference position return functions can be used only after the reference position return (G28) or manual reference position return (see III–3.1) is made. The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.

• Return from the reference position (G29)

In general, it is commanded immediately following the G28 command or G30. For incremental programming, the command value specifies the incremental value from the intermediate point.
Positioning to the intermediate or reference points are performed at the rapid traverse rate of each axis.
When the workpiece coordinate system is changed after the tool reaches the reference position through the intermediate point by the G28 command, the intermediate point also shifts to a new coordinate system. If G29 is then commanded, the tool moves to to the commanded position through the intermediate point which has been shifted to the new coordinate system.
The same operations are performed also for G30 commands.

• Reference position return check (G27)

G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the reference position return lamp lights up. However, if the position reached by the tool is not the reference position, an alarm (No. 092) is displayed.

Restrictions

• Status the machine lock being turned on

The lamp for indicating the completion of return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a G27 command is specified.

• First return to the reference position after the power has been turned on (without an absolute position detector)

When the G28 command is specified when manual return to the reference position has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference position.
In this case, the tool moves in the direction for reference position return specified in parameter ZMIx (bit 5 of No. 1006). Therefore the specified intermediate position must be a position to which reference position return is possible.

- **Reference position return check in an offset mode**

In an offset mode, the position to be reached by the tool with the G27 command is the position obtained by adding the offset value. Therefore, if the position with the offset value added is not the reference position, the lamp does not light up, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.

- **Lighting the lamp when the programmed position does not coincide with the reference position**

When the machine tool system is an inch system with metric input, the reference position return lamp may also light up, even if the programmed position is shifted from the reference position by the least input increment. This is because the least input increment of the machine tool system is smaller than its least command increment.

Reference

- **Manual reference position return**

See III–3.1.

Examples

G28G90X1000.0Y500.0 ;	(Programs movement from A to B)
T1111 ;	(Changing the tool at the reference position)
G29X1300.0Y200.0 ;	(Programs movement from B to C)

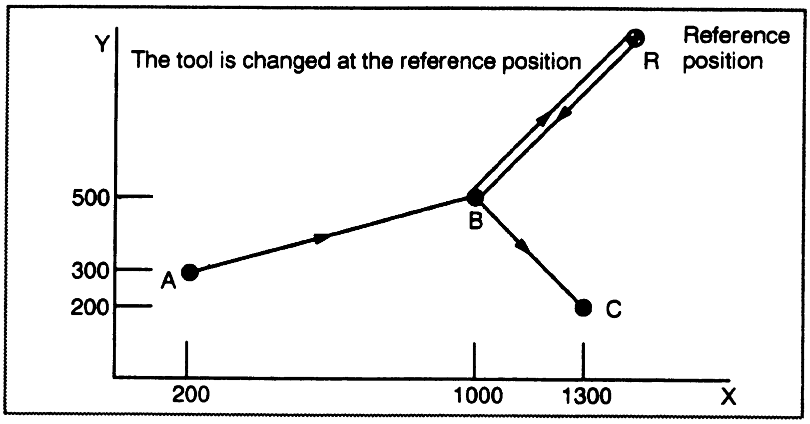


Fig. 6 (c) Reference position return and return from the reference position

7 COORDINATE SYSTEM

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes.
When three program axes, the X–axis, Y–axis, and Z–axis, are used, coordinates are specified as follows:

X_Y_Z_

This command is referred to as a dimension word.

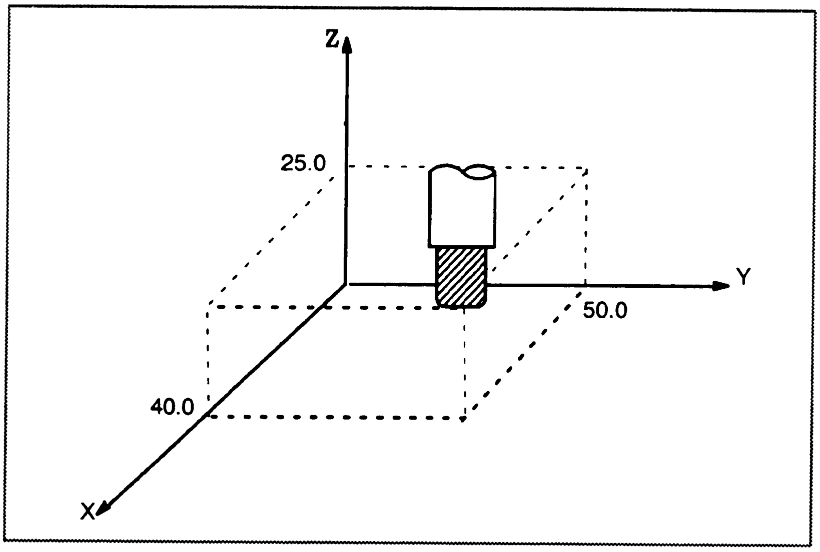


Fig. 7 Tool Position Specified by X40.0Y50.0Z25.0

Coordinates are specified in one of following three coordinate systems:
(1) Machine coordinate system
(2) Workpiece coordinate system
(3) Local coordinate system

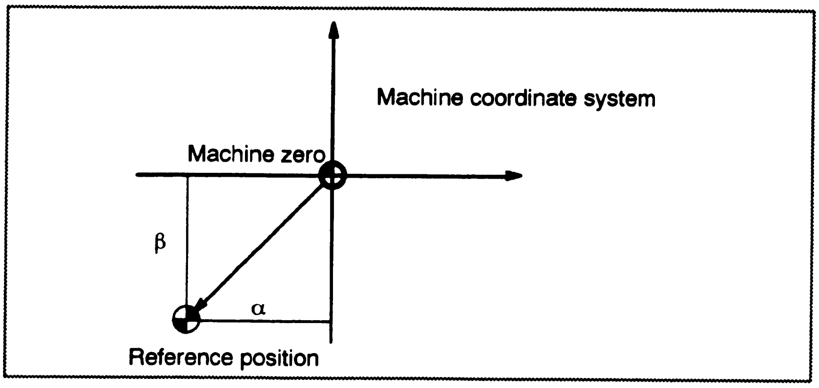
The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

7.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine. A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.
A machine coordinate system is set by performing manual reference position return after power–on (see III–3.1). A machine coordinate system, once set, remains unchanged until the power is turned off.

7.1.1 Setting a machine coordinate system

When manual reference position return is performed after power–on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.



7.1.2 Selecting a machine coordinate system (G53)

When manual reference position return is performed after power–on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.

Format

```
G90 G53 P__;
        P__; Absokute dimension word
```

Explanations

- **Selecting a machine coordinate system (G53)**

When a command is specified to move to a position in a machine coordinate system, the tool moves to the specified position at rapid traverse. G53, used to select a machine coordinate system, is a one–shot G code; that is, a command based on the machine coordinate system is valid only in the G53 block specifying the coordinate system. G53 must be specified in absolute programming (enabled with an absolute command, G90). If G53 is specified in incremental programming (enebled with an incremental command, G91), it is ignored. When the tool is to be moved to a machine–specific position such as a tool change position. program the movement in a machine coordinate system based on G53.

Restrictions

- **Cancel of the compensation function**

When the G53 command is specified, cancel the cutter compensation and tool length offset.

- **G53 specification immediately after power–on**

Since the machine coordinate system must be set before the G53 command is specified, at least one manual reference position return or automatic reference position return by the G28 command must be performed after the power is turned on. This is not necessary when an absolute–position detector is attached.

7.2 WORKPIECE COORDINATE SYSTEM

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the CNC beforehand (setting a workpiece coordinate system).
A machining program sets a workpiece coordinate system (selecting a workpiece coordinate system).
A set workpiece coordinate system can be changed by shifting its origin (changing a workpiece coordinate system).

7.2.1 Setting a Workpiece Coordinate System

A workpiece coordinate system can be set using one of three methods:

(1) Method using G92
A workpiece coordinate system is set by specifying a value after G92 in the program.

(2) Automatic setting
If parameter ZPR (No. 1201#0) is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed (see Part III–3.1.).

(3) Input using the CRT/MDI panel
Six workpiece coordinate systems can be set beforehand using the CRT/MDI panel (see Part III–11.4.4.).
Before absolute commands can be used, a workpiece coordinate system must be established by applying one of the above methods.

Format

Setting a workpiece coordinate system by G92

```
(G90) G92 P_
```

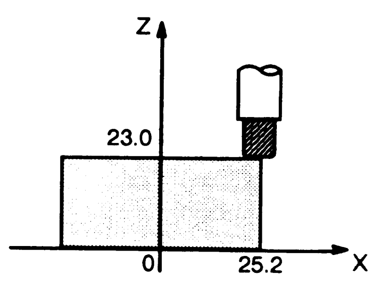
Explanations

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates. If a coordinate system is set using G92 during tool length offset, a coordinate system in which the position before offset matches the position specified in G92 is set.
Cutter compensation is cancelled temporarily with G92.

Examples

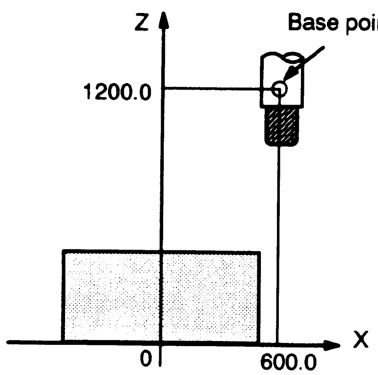
Example 1

Setting the coordinate system by the G92X25.2Z23.0; command
(The tool tip is the start point for the program.)



Example 2

Setting the coordinate system by the G92X600.0Z1200.0; command
(The base point on the tool holder is the start point for the program.)



If an absolute command is issued, the base point moves to the commanded position. In order to move the tool tip to the commanded position, the difference from the tool tip to the base point is compensated by tool length offset (See II–14.1).

7.2.2 Selecting a Workpiece Coordinate System

The user can choose from set workpiece coordinate systems as described below. (For information about the methods of setting, see Section 7.2.1.)

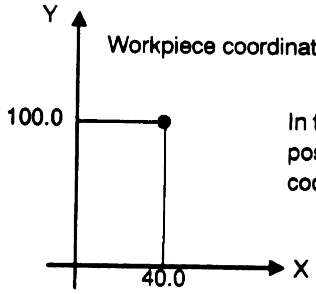
(1) **Setting by G92 or automatic workpiece coordinate system setting**
Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

(2) **Choosing from six workpiece coordinate systems set using the CRT/MDI panel**
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.
G54 .... Workpiece coordinate system 1
G55 .... Workpiece coordinate system 2
G56 .... Workpiece coordinate system 3
G57 .... Workpiece coordinate system 4
G58 .... Workpiece coordinate system 5
G59 .... Workpiece coordinate system 6

Workpiece coordinate system 1 to 6 are established after reference position return after the power is turned on. When the power is turned on, G54 coordinate system is selected.

Examples

G90 G55 G00 X40.0 Y100.0 ;



In this example, positioning is made to positions (X=40.0, Y=100.0) in workpiece coordinate system 2.

Fig. 7.2.2 (a)

7.2.3 Changing Workpiece Coordinate System

The six workpiece coordinate systems specified with G54 to G59 can be changed by changing an external workpiece zero point offset value or workpiece zero point offset value.
Three methods are available to change an external workpiece zero point offset value or workpiece zero point offset value.

(1) Inputting from the CRT/MDI panel (see III–11.4.4)
(2) Programming by G10 or G92
(3) Using external workpiece. Coordinate system shift function.
An input signal to the CNC from the PMC enables you to change the external workpiece zero point offset amount.
Refer to machine tool builder's manual for details.

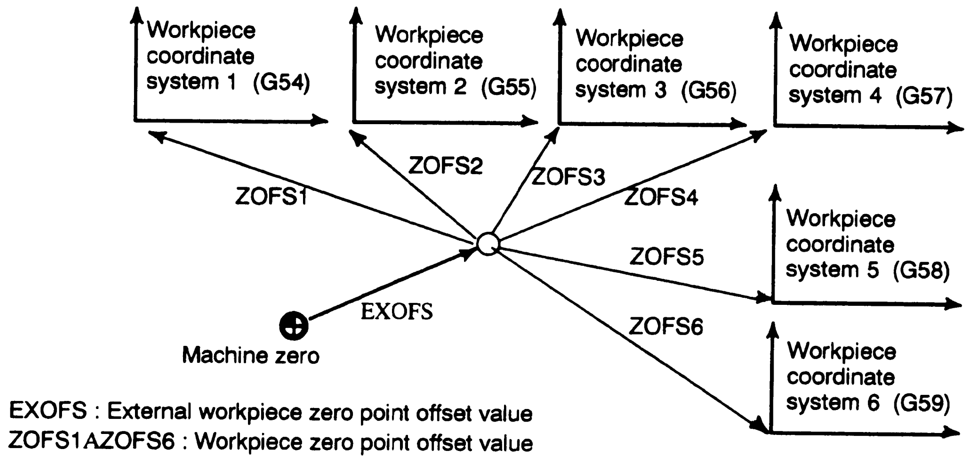


Fig. 7.2.3 (a) Changing an external workpiece zero point offset value or workpiece zero point offset value

Format

Changing by G10

G10 L2 Pp IP_; **p=0 : External workpiece zero point offset value** **p=1 to 6 : Workpiece zero point offset value correspond to workpiece coordinate system 1 to 6** **IP : Workpiece zero point offset value of each axis**

Changing by G92

G92 IP_;

Explanations

- **Changing by G10**

With the G10 command, each workpiece coordinate system can be changed separately.

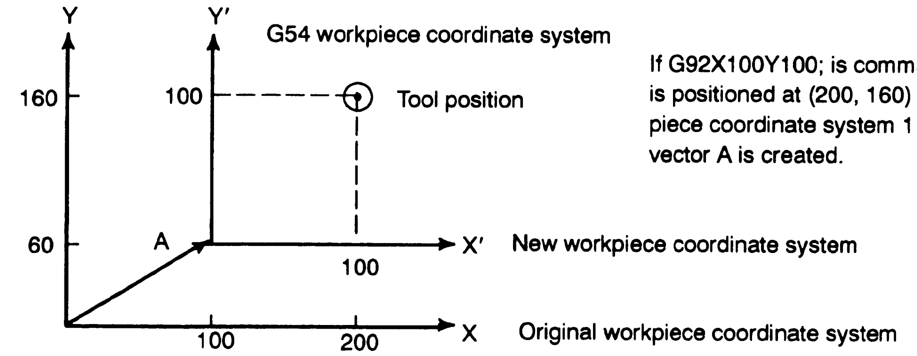
- **Changing by G92**

By specifying G92IP_;, a workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).
Then, the amount of coordinate system shift is added to all the workpiece zero point offset values. This means that all the workpiece coordinate systems are shifted by the same amount.

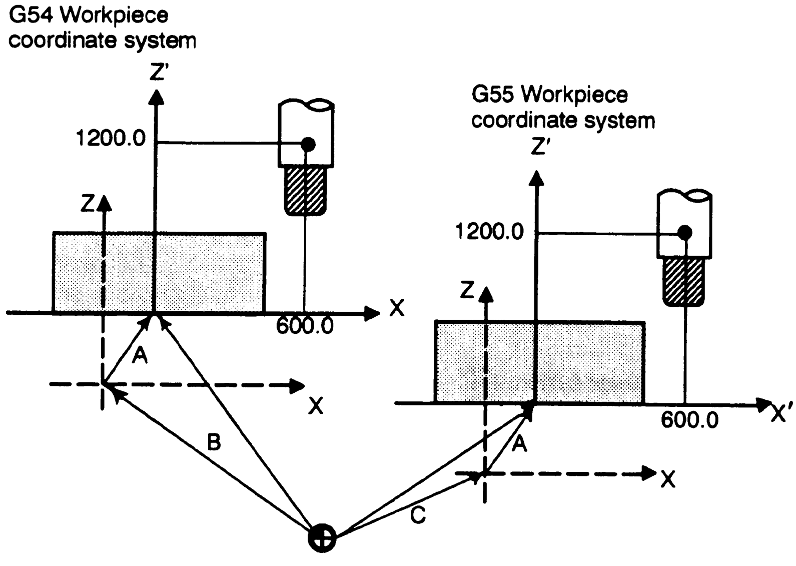
Note

When a coordinate system is set with G92 after an external workpiece zero point offset value is set, the coordinate system is not affected by the external workpiece zero point offset value. When G92X100.0Z80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Z = 80.0 is set.

Examples



If G92X100Y100; is commanded when the tool is positioned at (200, 160) in G54 mode, workpiece coordinate system 1 (X' – Y') shifted by vector A is created.



X' – Z' --- New workpiece coordinate system
X – Z ---- Original workpiece coordinate system
A : Offset value created by G92
B : Workpiece zero point offset value in G54
C : Workpiece zero point offset value in G55

Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black circle on the tool (figure at the left) is at (600.0,12000.0) can be set with the following command if the relative relationship between the G54 workpiece coordinate system and G55 workpiece coordinate system is set correctly:G92X600.0Z1200.0;Also, suppose that pallets are loaded at two different positions. If the relative relationship of the coordinate systems of the pallets at the two positions is correctly set by handling the coordinate systems as the G54 workpiece coordinate system and G55 workpiece coordinate system, a coordinate system shift with G92 in one pallet causes the same coordinate system shift in the other pallet. This means that workpieces on two pallets can be machined with the same program just by specifying G54 or G55.

7.3 LOCAL COORDINATE SYSTEM

When a program is created in a workpiece coordinate system, a child workpiece coordinate system may be set for easier programming. Such a child coordinate system is referred to as a local coordinate system.

Format

G52 IP _; Setting the local coordinate system

......

G52 IP 0 ; Canceling of the local coordinate system

IP _ : Origin of the local coordinate system

Explanations

By specifying G52 IP_;, a local coordinate system can be set in all the workpiece coordinate systems (G54 to G59). The origin of each local coordinate system is set at the position specified by IP_ in the workpiece coordinate system.
When a local coordinate system is set, the move commands in absolute mode (G90), which is subsequently commanded, are the coordinate values in the local coordinate system. The local coordinate system can be changed by specifying the G52 command with the zero point of a new local coordinate system in the workpiece coordinate system.
To cancel the local coordinate system and specify the coordinate value in the workpiece coordinate system, match the zero point of the local coordinate system with that of the workpiece coordinate system.

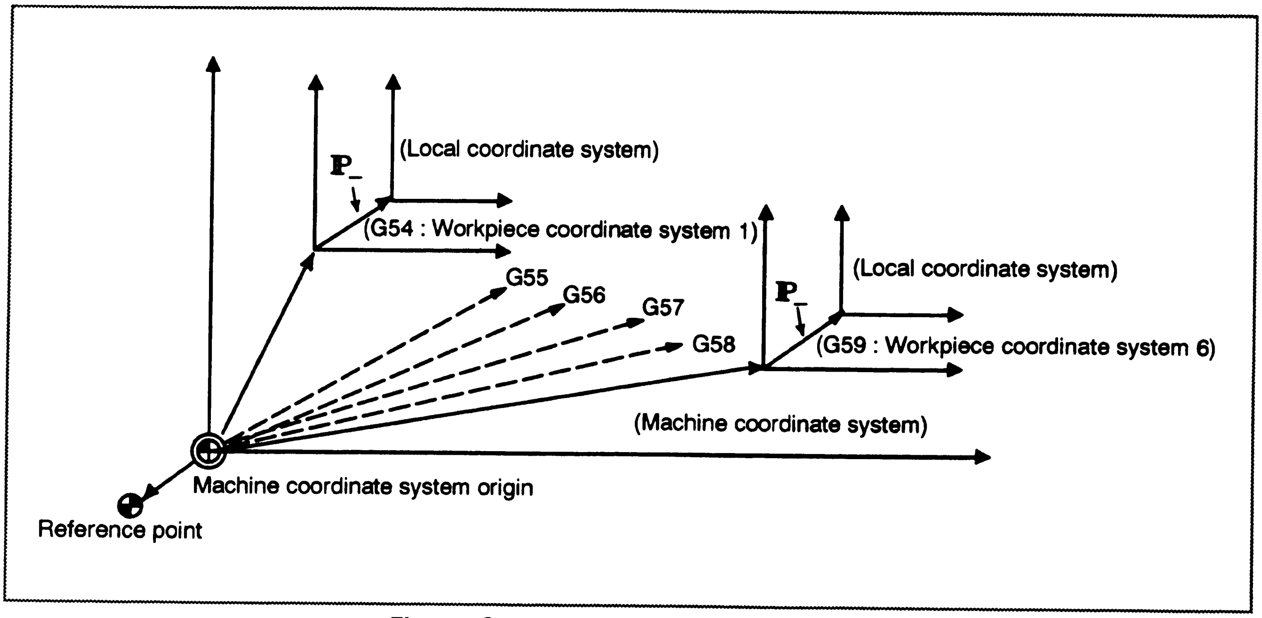


Fig. 7.3 Setting the local coordinate system

Notes

1. When an axis returns to the reference point by the manual reference point return function, the zero point of the local coordinate system of the axis matches that of the work coordinate system. The same is true when the following command is issued:

   G52α0;

   α:Axis which returns to the reference point
2. The local coordinate system setting does not change the workpiece and machine coordinate systems.
3. Whether the local coordinate system is canceled at reset depends on the parameter setting. The local coordinate system is canceled when either CLR, bit 6 of parameter No. 3402 or RLC, bit 3 or parameter No. 1202 is set to 1.
4. The local coordinate system is canceled when a workpiece coordinate system is established according to the specification of G92.
5. G52 cancels the offset temporarily in cutter compensation.
6. Command a move command immediately after the G52 block in the absolute mode.

7.4 PLANE SELECTION

Select the planes for circular interpolation, cutter compensation, and drilling by G–code.
The following table lists G–codes and the planes selected by them.

Explanations

Table 7.4 Plane selected by G code

G code	Selected plane	Xp	Yp	Zp
G17	Xp Yp plane	X–axis or an axis parallel to it	Y–axis or an axis parallel to it	Z–axis or an axis parallel to it
G18	Zp Xp plane			
G19	Yp Zp plane			

The plane is unchanged in the block in which G17, G18 or G19 is not commanded.
When the power is turned on or the CNC is reset, G17 (XY plane), G18 (ZX plane), or G19 (YZ plane) is selected by bits 1 (G18) and 2 (G19) of parameter 3402.
The movement instruction is irrelevant to the plane selection.

Examples

Plane selection when the X–axis is parallel with the U–axis.

G17X_Y_	XY plane,
G18X_Z_	ZX plane
X_Y_	Plane is unchanged (ZX plane)
G17	XY plane
G18	ZX plane
G18Y_ ;	ZX plane, Y axis moves regardless without any relation to the plane.

8 COORDINATE VALUE AND DIMENSION

This chapter contains the following topics.

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)
8.2 INCH/METRIC CONVERSION (G20, G21)
8.3 DECIMAL POINT PROGRAMMING

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

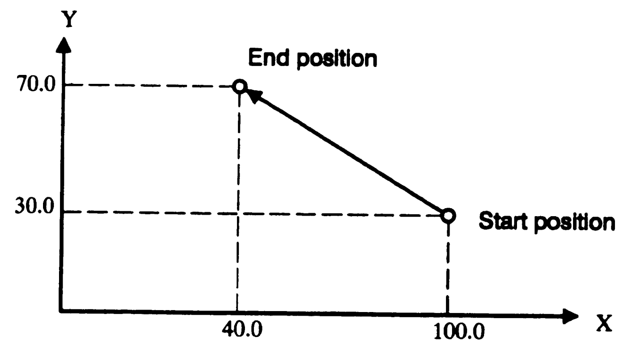
There are two ways to command travels of the tool; the absolute command, and the incremental command. In the absolute command, coordinate value of the end position is programmed; in the incremental command, move distance of the position itself is programmed. G90 and G91 are used to command absolute or incremental command, respectively.

Format

Absolute command	**G90 P_ ;**
Incremental command	**G91 P_ ;**

Examples

G90 X40.0 Y70.0 ; **Absolute command**
G91 X–60.0 Y40.0 ; **Incremental command**



8.2 INCH/METRIC CONVERSION(G20,G21)

Either inch or metric input can be selected by G code.

Format

G20 ;	Inch input
G21 ;	mm input

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. After the G code for inch/metric conversion is specified, the unit of input data is switched to the least inch or metric input increment of increment system IS–B (Section 2.3). The unit systems for the following values are changed after inch/metric conversion:

– Feedrate commanded by F code
– Positional command
– Work zero point offset value
– Tool compensation value
– Unit of scale for manual pulse generator
– Movement distance in incremental feed
– Some parameters

When the power is turned on, the G code is the same as that held before the power was turned off.

Notes

1. G20 and G21 must not be switched during a program.
2. When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
3. When switching inch input (G20) to metric input (G21) and vice versa, the tool compensation value must be re-set according to the least input increment.
   However, when bit 0 (OIM) of parameter 5006 is 1, tool compensation values are automatically converted and need not be re-set.
4. When the G28 command is specified first after inch input is switched to metric input or rice versa, movement from the intermediate point is the same as for manual return to the reference position. In this case, the tool moves in the direction for reference return, as specified with bit 5 (ZMI) of parameter No. 1006.
5. The inch and metric input can also be switched by setting of the data setting (See III–11.4.2).

8.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Y, Z, U, V, W, A, B, C, I, J, K, Q, R, and F.

Explanations

There are two types of decimal point notation: calculator–type notation and standard notation.
When calculator–type decimal notation is used, a value without decimal point is considered to be specified in mm or inch. When standard decimal notation is used, such a value is considered to be specified in least input increments.Select either calculator–type or standard decimal notation by using the DPI bit (bit 0 of parameter 3401).Values can be specified both with and without decimal point in a single program.

Examples

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001 mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

Notes

1. In a single block, specify a G code before entering a value. The position of decimal point may depend on the command.
   Examples:
   G20; Input in inches
   X1.0 G04; X1.0 is considered to be a distance and processed as X10000. This command is equivalent to G04 X10000. The tool dwells for 10 seconds.
   G04 X1.0; Equivalent to G04 X1000. The tool dwells for one second.
2. Fractions less than the least input increment are truncated.
   Examples:
   X1.2345; Truncated to X1.234 when the least input increment is 0.001 mm.
   Processed as X1.2345 when the least input increment is 0.0001 inch.
3. When more than eight digits are specified, an alarm occurs. If a value is entered with a decimal point, the number of digits is also checked after the value is converted to an integer according to the least input increment.
   Examples:
   X1.23456789; Alarm 003 occurs because more than eight digits are specified.
   X123456.7; If the least input increment is 0.001 mm, the value is converted to integer 123456700. Because the integer has more than eight digits, an alarm occurs.

9 SPINDLE SPEED FUNCTION (S FUNCTION)

The spindle speed can be controlled by specifying a value following address S.
This chapter contains the following topics.

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5–DIGIT COMMAND)

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE

By specifying numerical value following address S, a binary code signal and a strobe signal are a block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the number of digits in an S code or the execution order when a move command and an S code command are in the same block.

9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5–DIGIT COMMAND)

The spindle speed can be specified directly by address S followed by a Max. five–digit value (rpm). The unit for specifying the spindle speed may vary depending on the machine tool builder. Refer to the appropriate manual provided by the machine tool builder for details.

10 TOOL FUNCTION (T FUNCTION)

10.1 TOOL SELECTION FUNCTION

By specifying an up to 8–digit numerical value following address T, tools can be selected on the machine.
One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commandable with address T and the correspondence between the T codes and machine operations.
When a move command and a T code are specified in the same block, the commands are executed in one of the following two ways:

(i) Simultaneous execution of the move command and T function commands.

(ii) Executing T function commands upon completion of move command execution.
The selection of either (i) or (ii) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

11 AUXILIARY FUNCTION

General

There is the following function: miscellaneous function (M code) for specifying spindle start, spindle stop program end, and so on.
When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:

i) Simultaneous execution of the move command and miscellaneous function commands.

ii) Executing miscellaneous function commands upon completion of move command execution.
The selection of either sequence depends on the machine tool builder's specification. Refer to the manual issued by the machine tool builder for details.

11.1 AUXILIARY FUNCTION (M FUNCTION)

When a numeral is specified following address M, code signal and a strobe signal are sent to the machine. The machine uses these signals to turn on or off its functions.
Usually, only one M code can be specified in one block. In some cases, however, up to three M codes can be specified for some types of machine tools.
Which M code corresponds to which machine function is determined by the machine tool builder.
The machine processes all operations specified by M codes except those specified by M98 or M99. Refer to the machine tool builder's instruction manual for details.

Explanations

The following M codes have special meanings.

- **M02,M03 (End of program)**

This indicates the end of the main program
Automatic operation is stopped and the CNC unit is reset.
This differs with the machine tool builder.
After a block specifying the end of the program is executed, control returns to the start of the program.
Bit 5 of parameter 3404 (M02) can be used to disable M02 from returning control to the start of the program.
Bit4 of parameter 3404 (M30) can be used to disable M30 from returning control to the start of the program.

- **M00 (Program stop)**

Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation. This differs with the machine tool builder.

- **M01 (Optional stop)**

Similarly to M00, automatic operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel has been pressed.

- **M98 (Calling of sub–program)**

This code is used to call a subprogram. The code and strobe signals are not sent. See the subprogram **section 12.3** for details .

- **M99 (End of subprogram)**

This code indicates the end of a subprogram.
M99 execution returns control to the main program. The code and strobe signals are not sent. See the subprogram **section 12.3** for details.

> **Note**
> The block subsequent to M00, M01, M02, or M30 is not read in advance (no buffering) Similarly, ten M codes which do not buffer can be set by parameters (Nos. 3411 to 3420). Refer to the machine tool builder's instruction manual for these M codes.

11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

In general, only one M code can be specified in a block. However, up to three M codes can be specified at once in a block by setting bit 7 (M3B) of parameter No. 3404 to 1. Up to three M codes specified in a block are simultaneously output to the machine. This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining.

Explanations

CNC allows up to three M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.

M00, M01, M02, M30, M98, M99, or M198 must not be specified together with another M code.

Some M codes other than M00, M01, M02, M30, M98, M99, and M198 cannot be specified together with other M codes; each of those M codes must be specified in a single block.

Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks. Meanwhile, multiple of M codes that direct the CNC only to send the M codes themselves (without performing internal operations) can be specified in a single block.

Examples

One M command in a single block	Multiple M commands in a single block
M40 ; M50 ; M60 ; G28G91X0Y0Z0 ; : : :	M40M50M60 ; G28G91X0Y0Z0 ; : : :

12 PROGRAM CONFIGURATION

General

• Main program and subprogram

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

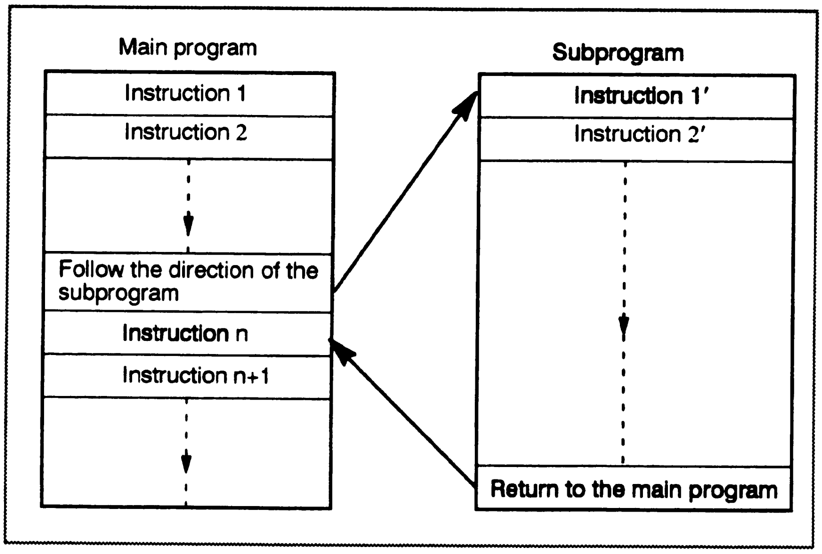


Fig. 12 (a) Main program and Subprogram

The CNC memory can hold up to 63 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine. See Chapter 10 in OPERATION for the methods of registering and selecting programs.

- **Program components**

A program consists of the following components:

Table 12(a) Program components

Components	Descriptions
Tape start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Tape end	Symbol indicating the end of a program file

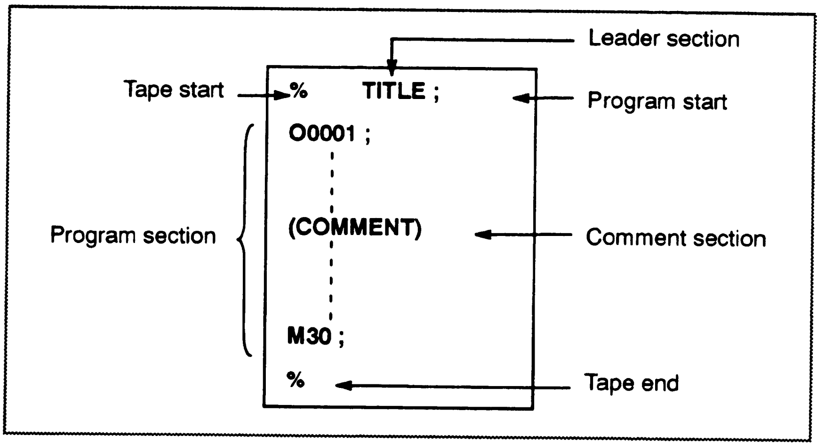


Fig. 12(b) Program configuration

- **Program section configuration**

A program section consists of several blocks. A program section starts with a program number and ends with a program end code.

Program section configuration	Program section
Program number	O0001 ;
Block 1	N1 G91 G00 X120.0 Y80.0 ;
Block 2	N2 G43 Z–32.0 H01 ;
:	:
Block n	Nn Z0 ;
Program end	M30 ;

A block contains information necessary for machining, such as a move command or coolant on/off command. Specifying a slash (/) at the start of a block disables the execution of some blocks (see “optional block skip” in Section 12.2).

12.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See Section 12.2 for a program section.

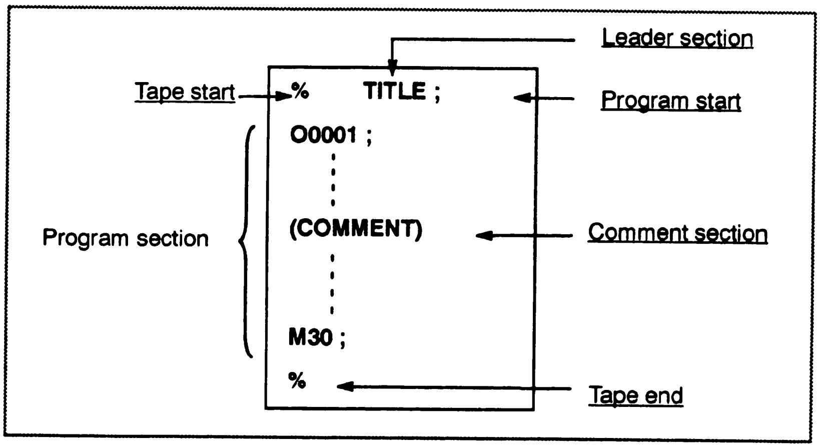


Fig. 12.1(a) Program configuration

Explanations

• Tape start

The tape start indicates the start of a file that contains CNC programs.
The mark is not required when programs are entered using SYSTEM P or ordinary personal computers. The mark is not displayed on the CRT display screen.
However, if the file is output,the mark is automatically output at the start of the file.

Table 12.1(a) Code of a tape start

Name	ISO code	EIA code	Notation in this manual
Tape start	%	ER	%

• Leader section

Data entered before the programs in a file constitutes a leader section.
When machining is started, the label skip state is usually set by turning on the power or resetting the system. In the label skip state, all information is ignored until the first end–of–block code is read. When a file is read into the CNC unit from an I/O device, leader sections are skipped by the label skip function.
A leader section generally contains information such as a file header. When a leader section is skipped, even a TV parity check is not made. So a leader section can contain any codes except the EOB code.

• Program start

The program start code is to be entered immediately after a leader section, that is, immediately before a program section.
This code indicates the start of a program, and is always required to disable the label skip function.
With SYSTEM P or ordinary personal computers, this code can be entered by pressing the return key.

Table 12.1(b) Code of a program start

Name	ISO code	EIA code	Notation in this manual
Program start	LF	CR	;

Note

If one file contains multiple programs, the EOB code for label skip operation must not appear before a second or subsequent program number.

- **Comment section**

Any information enclosed by the control–out and control–in codes is regarded as a comment.
The user can enter a header, comments, directions to the operator, etc. in a comment section using the EOB code or any other code.

Table 12.1 (c) Codes of a control–in and a control–out

Name	ISO code	EIA code	Notation in this manual	Meaning
Control–out	(	2–4–5	(	Start of comment section
Control–in	)	2–4–7	)	End of comment section

When a program is read into memory for memory operation, comment sections, if any, are not ignored but are also read into memory. Note, however, that codes other than those listed in the code table in Appendix A are ignored, and thus are not read into memory.
When a program in this memory is output to an external device (see III–8), the comments in the program are also output.
When a program is displayed on the screen, its comment sections are also displayed. However, those codes that were ignored when read into memory are not output or displayed.
When memory operation or DNC operation is performed, all comment sections are ignored.

Notes

1. If a long comment section appears in the middle of a program section, a move along an axis may be suspended for a long time because of such a comment section. So a comment section should be placed where movement suspension may occur or no movement is involved.
2. If only a control–in code is read with no matching control–out code, the read control–in code is ignored.

- **Tape end**

A tape end is to be placed at the end of a file containing NC programs. If programs are entered using the automatic programming system, the mark need not be entered.
The mark is not displayed on the CRT display screen. However, when a file is output, the mark is automatically output at the end of the file.
If an attempt is made to execute % when M02 or M03 is not placed at the end of the program, the alarm (No. 5010) is occurred.

Table 12.1(d) Code of a tape end

Name	ISO code	EIA code	Notation in this manual
Tape end	%	ER	%

12.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See Section 12.1 for program components other than program sections.

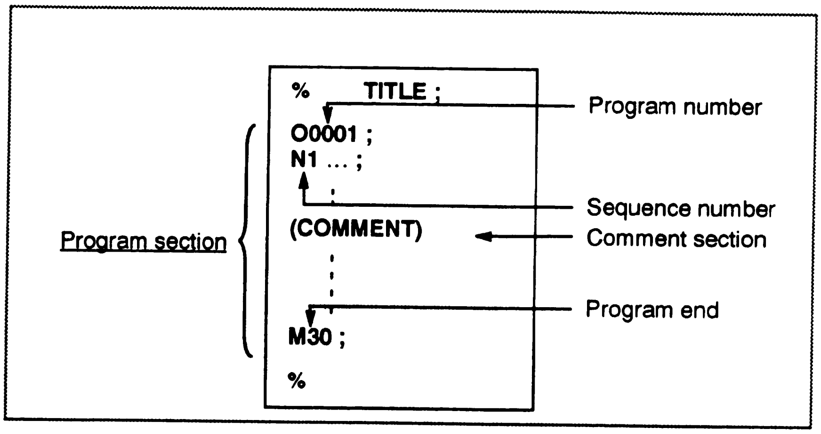


Fig. 12.2(a) Program configuration

- **Program number**

A program number consisting of address O followed by a four–digit number is assigned to each program at the beginning registered in memory to identify the program.
In ISO code, the colon (:) can be used instead of O.
When no program number is specified at the start of a program, the sequence number (N....) at the start of the program is regarded as its program number. If a five–digit sequence number is used, the lower four digits are registered as a program number. If the lower four digits are all 0, the program number registered immediately before added to 1 is registered as a program number. Note, however, that N0 cannot be used for a program number.

Note
Program numbers 8000 to 9999 may be used by machine tool builders, and the user may not be able to use these numbers.

- **Sequence number and block**

A program consists of several commands. One command unit is called a block. One block is separated from another with an end of block code (EOB).

Table 12.2(a) EOB code

Name	ISO code	EIA code	Notation in this manual
End of block (EOB)	LF	CR	;

At the head of a block, a sequence number consisting of address N followed by a number not longer than five digits (1 to 99999) can be placed. Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps (for example, when a new tool is used by tool replacement, and machining proceeds to a new surface with table indexing.)

<u>N300</u> X200.0 Z300.0 ; A sequence number is underlined.

Fig. 12.2(b) Sequence number and block (example)

> **Note**
> N0 must not be used for the reason of file compatibility with other CNC systems.
> Program number 0 cannot be used. So 0 must not be used for a sequence number regarded as a program number.

- **TV check (Vertical parity check along tape)**

A parity check is made for a block on input tape vertically. If the number of characters in one block (starting with the code immediately after an EOB and ending with the next EOB) is odd, an alarm (No.002) is output. No TV check is made only for those parts that are skipped by the label skip function. Select by setting parameter CTV (No. 0100#1), whether a comment section enclosed in parenthesis is counted to the number of characters in TV check or not. The TV check function can be enabled or disabled by setting on the MDI unit (See Subsec. 11.4.3 in Part III.).

- **Block configuration (word and address)**

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus sign (–) may be prefixed to a number.)
Word = Address + number (Example : X–1000)
For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address. Table 12.2 (b) indicates the usable addresses and their meanings.
The same address may have different meanings, depending on the preparatory function specification.

Table 12.2(b) Major functions and addresses

Function	Address	Meaning
Program number	O (1)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)
Dimension word	X, Y, Z	Coordinate axis move command
	I, J, K	Coordinate of the arc center
	R	Arc radius
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
Offset number	D, H	Offset number
Dwell	P, X	Dwell time
Program number designation	P	Subprogram number
Number of repetitions	P	Number of subprogram repetitions
Parameter	P, Q	Canned cycle parameter

> **Note**
> In ISO code, the colon (:) can also be used as the address of a program number.

N_	G_	X_ Y_	F_	S_	T_	M_	;
Sequence number	Preparatory function	Dimension word	Feed–function	Spindle speed function	Tool function	Miscellaneous function	

Fig. 12.2 (c) 1 block (example)

- **Major addresses and ranges of command values**

Major addresses and the ranges of values specified for the addresses are shown below. Note that these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example, the CNC allows a tool to traverse up to about 100 m (in millimeter input) along the X axis.
However, an actual stroke along the X axis may be limited to 2 m for a specific machine tool.
Similarly, the CNC may be able to control a cutting federate of up to 240 m/min, but the machine tool may not allow more than 3 m/min. When developing a program, the user should carefully read the manuals of the machine tool as well as this manual to be familiar with the restrictions on programming.

Table 12.2(c) Major addresses and ranges of command values

Function	Address	Input in mm	Input in inch
Program number	O [1]	1–9999	1–9999
Sequence number	N	1–99999	1–99999
Preparatory function	G	0–99	0–99
Dimension word	X, Y, Z, I, J, K, R	±99999.999mm	±9999.9999inch
Feed per minute	F	1–240000mm/min	0.01–9600.00 inch/min
Spindle speed function	S	0–20000	0–20000
Tool function	T	0–99999999	0–99999999
Auxiliary function	M	0–99999999	0–99999999
Offset number	H, D	0–32	0–32
Dwell	X, P	0–99999.999s	0–99999.999s
Designation of a program number	P	1–9999	1–9999
Number of subprogram repetitions	P	1–999	1–999

Note
In ISO code, the colon (:) can also be used as the address of a program number.

• Optional block skip

When a slash (/) is specified at the head of a block, and optional block skip switch on the machine operator panel is set to on, the information contained in the block for which / corresponding to switch number is specified is ignored in tape operation or memory operation.
When optional block skip switch is set to off, the information contained in the block for which / is specified is valid. This means that the operator can determine whether to skip the block containing /.
This function is ignored when programs are loaded into memory. Blocks containing /n are also stored in memory, regardless of how the optional block skip switch is set.
Programs held in memory can be output, regardless of how the optional block skip switches are set.
Optional block skip is effective even during sequence number search operation.

Notes

1. ***Position of a slash***
   A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.
2. ***TV and TH check***
   When an optional block skip switch is on. TH and TV checks are made for the skipped portions in the same way as when the optional block skip switch is off.
3. ***Disabling an optional block skip switch***
   Optional block skip operation is processed when blocks are read from memory or tape into a buffer. Even if a switch is set to on after blocks are read into a buffer, the blocks already read are not ignored.

• Program end

The end of a program is indicated by programming one of the following codes at the end of the program:

Table 12.2(d) Code of a program end

Code	Meaning usage
M02	For main program
M30	
M99	For subprogram

If one of the program end codes is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. When the subprogram end code is executed, control returns to the program that called the subprogram.

Note
A block containing an optional block skip code such as /M02 ; , /M30 ; , or /M99 ; is not regarded as the end of a program, if the optional block skip switch on the machine operator's panel is set to on.
(See Section 13.2 for optional block skip.)

12.3 SUBPROGRAM

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.
A subprogram can be called from the main program.
A called subprogram can also call another subprogram.

Format

• Subprogram configuration

One subprogram

```
O □□□□ ;    Subprogram number
 :          (or the colon (:) optionally in the case of ISO)
 :
M99 ;       Program end
```

M99 need not constitute a separate block as indicated below.

Example) **X100.0 Y100.0 M99 ;**

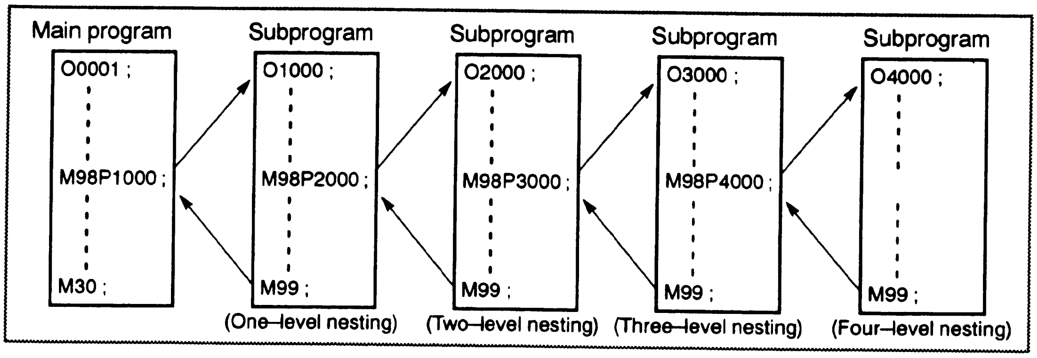
• Subprogram call

```
M98 P ○○○  ○○○○ ;
      ↑     ↑
      Number of times the subprogram is called repeatedly
            Subprogram number
```

When no repetition data is specified, the subprogram is called just once.

Explanations

When the main program calls a subprogram, it is regarded as a one–level subprogram call. Thus, subprogram calls can be nested up to four levels as shown below.



A single call command can repeatedly call a subprogram up to 9999 times. For compatibility with automatic programming systems, in the first block, Nxxxx can be used instead of a subprogram number that follows O (or :). A sequence number after N is registered as a subprogram number.

• Reference

See Chapter 10 in Part III for the method of registering a subprogram.

Notes
1. The M98 and M99 code or strobe signals are not output to the machine tool.
2. If the subprogram number specified by address P cannot be found, an alarm (No. 078) is output.

Examples

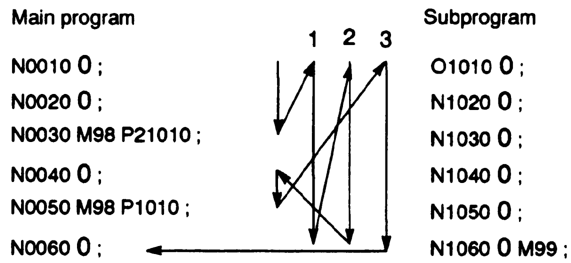
☆ **M98 P51002 ;**

This command specifies "Call the subprogram (number 1002) five times in succession." A subprogram call command (M98P_) can be specified in the same block as a move command.

☆ **X1000.0 M98 P1200 ;**

This example calls the subprogram (number 1200) after an X movement.

☆ Execution sequence of subprograms called from a main program



A subprogram can call another subprogram in the same way as a main program calls a subprogram.

Special Usage

- **Specifying the sequence number for the return destination in the main program**

If P is used to specify a sequence number when a subprogram is terminated, control does not return to the block after the calling block, but returns to the block with the sequence number specified by P. Note, however, that P is ignored if the main program is operating in a mode other than memory operation mode.
This method consumes a much longer time than the normal return method to return to the main program.

Main program	Subprogram
N0010 ... ;	O1010 ... ;
N0020 ... ;	N1020 ... ;
N0030 M98 P1010 ;	N1030 ... ;
N0040 ... ;	N1040 ... ;
N0050 ... ;	N1050 ... ;
N0060 ... ;	N1060 M99 P0060 ;

- **Using M99 in the main program**

If M99 is executed in a main program, control returns to the start of the main program. For example, M99 can be executed by placing /M99 ; at an appropriate location of the main program and setting the optional block skip function to off when executing the main program. When M99 is executed, control returns to the start of the main program, then execution is repeated starting at the head of the main program.
Execution is repeated while the optional block skip function is set to off.
If the optional block skip function is set to on, the /M99 ; block is skipped ; control is passed to the next block for continued execution.
If/M99P<u>n</u> ; is specified, control returns not to the start of the main program, but to sequence number n. In this case, a longer time is required to return to sequence number n.

```
                          N0010 ... ;
                          N0020 ... ;
                          N0030 ... ;
Optional block skip       N0040 ... ;
OFF                       N0050 ... ;
                        / N0060 M99 P0030 ;     Optional block skip
                          N0070 ... ;           ON
                          N0080 M02 ;
```

• Using a subprogram only

A subprogram can be executed just like a main program by searching for the start of the subprogram with the MDI.
(See Section 9.4 in Part III for information about search operation.)
In this case, if a block containing M99 is executed, control returns to the start of the subprogram for repeated execution. If a block containing M99P<u>n</u> is executed, control returns to the block with sequence number n in the subprogram for repeated execution. To terminate this program, a block containing /M02 ; or /M30 ; must be placed at an appropriate location, and the optional block switch must be set to off ; this switch is to be set to on first.

```
  N1010 ... ;
  N1020 ... ;
  N1030 ... ;
/ N1040 M02 ;
  N1050 M99 P1020 ;
```

Optional block skip ON

13 FUNCTIONS TO SIMPLIFY PROGRAMMING

General

This chapter explains the following items:

13.1 CANNED CYCLE
13.2 RIGID TAPPING
13.3 EXTERNAL MOTION FUNCTION (G81)

13.1 CANNED CYCLE

Canned cycles make it easier for the programmer to create programs. With a canned cycle, a frequently–used machining operation can be specified in a single block with a G function; without canned cycles, normally more than one block is required. In addition, the use of canned cycles can shorten the program to save memory.
Table 13.1 (a) lists canned cycles.

Table13.1(a) Canned cycles

G code	Drilling(–Z direction)	Operation at the bottom of a hole	Retraction(+Z direction)	Application
G73	Intermittent feed	–	Rapid traverse	High–speed peck drilling cycle
G74	Feed	Dwell→Spindle CW	Feed	Left–hand tapping cycle
G76	Feed	Oriented spindle stop	Rapid traverse	Fine boring cycle
G80	–	–	–	Cancel
G81	Feed	–	Rapid traverse	Drilling cycle, spot drilling cycle
G82	Feed	Dwell	Rapid traverse	Drilling cycle, counter boring cycle
G83	Intermittent feed	–	Rapid traverse	Peck drilling cycle
G84	Feed	Dwell→Spindle CCW	Feed	Tapping cycle
G85	Feed	–	Feed	Boring cycle
G86	Feed	Spindle stop	Rapid traverse	Boring cycle
G87	Feed	Spindle CW	Rapid traverse	Back boring cycle
G88	Feed	Dwell→spindle stop	Manual	Boring cycle
G89	Feed	Dwell	Feed	Boring cycle

Explanations

A canned cycle consists of a sequence of six operations (Fig. 13.1 (a))

Operation 1 ---- Positioning of axes X and Y
(including also another axis)
Operation 2 ---- Rapid traverse up to point R level
Operation 3 ---- Hole machining
Operation 4 ---- Operation at the bottom of a hole
Operation 5 ---- Retraction to point R level
Operation 6 ---- Rapid traverse up to the initial point

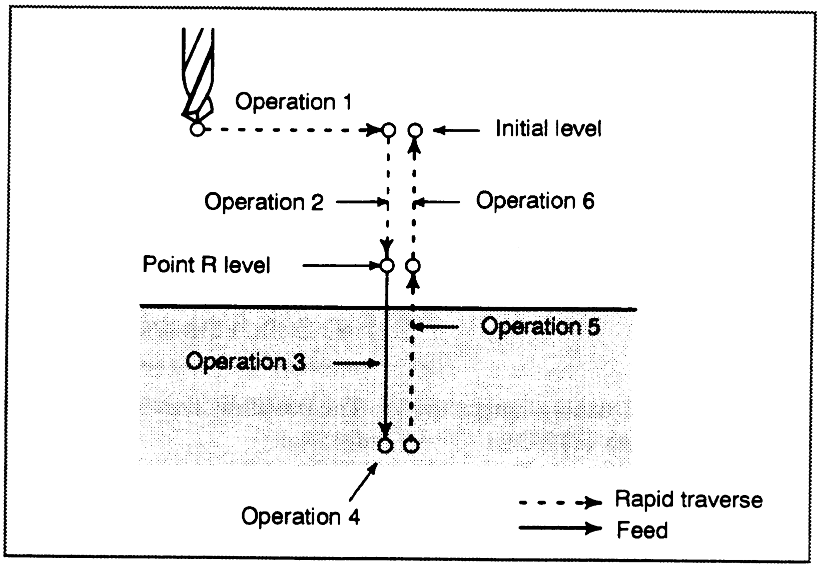


Fig. 13.1 (a) Canned cycle operation sequence

- **Positioning plane**

The positioning plane is determined by plane selection code G17, G18, or G19.
The positioning axis is an axis other than the drilling axis.

- **Drilling axis**

Although canned cycles include tapping and boring cycles as well as drilling cycles, in this chapter, only the term drilling will be used to refer to operations implemented with canned cycles.
The drilling axis is an axis (X, Y, or Z) not used to define the positioning plane.

Table13.1(b) Positioning plane and drilling axis

G code	Positioning plane	Drilling axis
G17	X–Y plane	Z
G18	Z–X plane	Y
G19	Y–Z plane	X

Examples

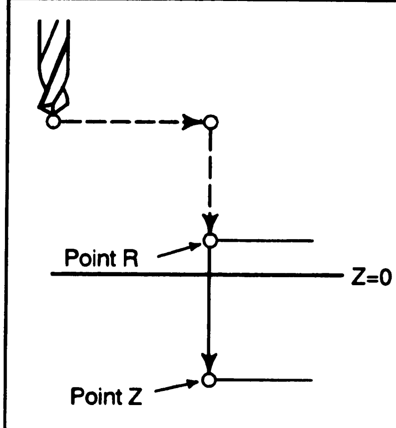
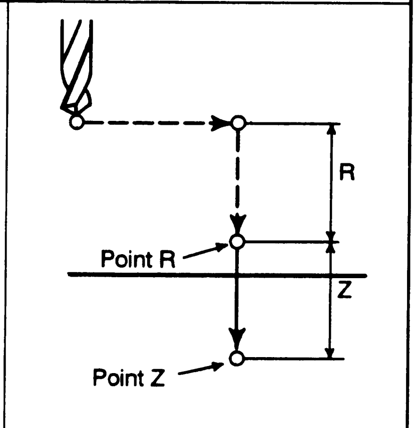
G17 to G19 may be specified in a block in which any of G73 to G89 is not specified.

Notes

1. A parameter FXY (No. 6200 #0) can be set to the Z axis always used as the drilling axis. When FXY=0, the Z axis is always the drilling axis.
2. Switch the drilling axis after canceling a canned cycle.

- **Travel distance along the drilling axis G90/G91**

The travel distance along the drilling axis varies for G90 and G91 as follows:

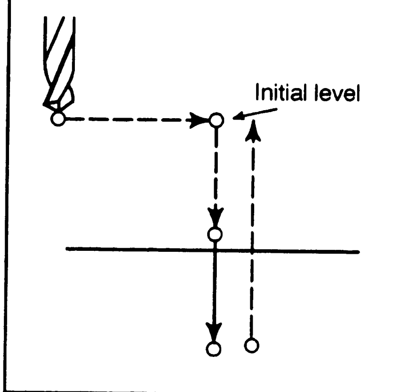
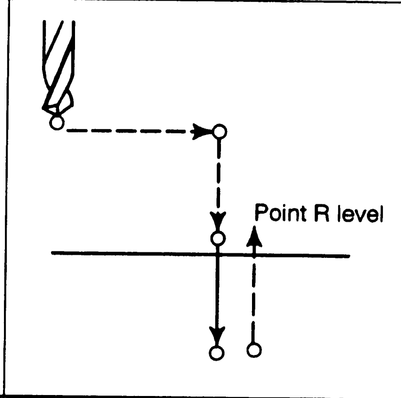
G90 (Absolute Command)	G91 (Incremental Command)
	

- **Drilling mode**

G73, G74, G76, and G81 to G89 are modal G codes and remain in effect until canceled. When in effect, the current state is the drilling mode. Once drilling data is specified in the drilling mode, the data is retained until modified or canceled.
Specify all necessary drilling data at the beginning of canned cycles; when canned cycles are being performed, specify data modifications only.

• Return point level G98/G99

When the tool reaches the bottom of a hole, the tool may be returned to point R or to the initial level. These operations are specified with G98 and G99. The following illustrates how the tool moves when G98 or G99 is specified. Generally, G99 is used for the first drilling operation and G98 is used for the last drilling operation.
The initial level does not change even when drilling is performed in the G99 mode.

G98(Return to initial level)	G99(Return to point R level)
	

• Repeat

To repeat drilling for equally–spaced holes, specify the number of repeats in K_.
K is effective only within the block where it is specified.
Specify the first hole position in incremental mode (G91).
If it is specified in absolute mode (G90), drilling is repeated at the same position.

Number of repeats K	The maximum command value = 9999

If K0 is specified, drilling data is stored, but drilling is not performed.

• Cancel

To cancel a canned cycle, use G80 or a group 01 G code.

Group 01 G codes
G00 : Positioning (rapid traverse)
G01 : Linear interpolation
G02 : Circular interpolation or helical interpolation (CW)
G03 : Circular interpolation or helical interpolation (CCW)

• Symbols in figures

Subsequent sections explain the individual canned cycles. Figures in these explanations use the following symbols:

- - -→	Positioning (rapid traverse G00)
——→	Cutting feed (linear interpolation G01)
~~→	Manual feed
OSS	Oriented spindle stop (The spindle stops at a fixed rotation position)
⇨	Shift (rapid traverse G00)
P	Dwell

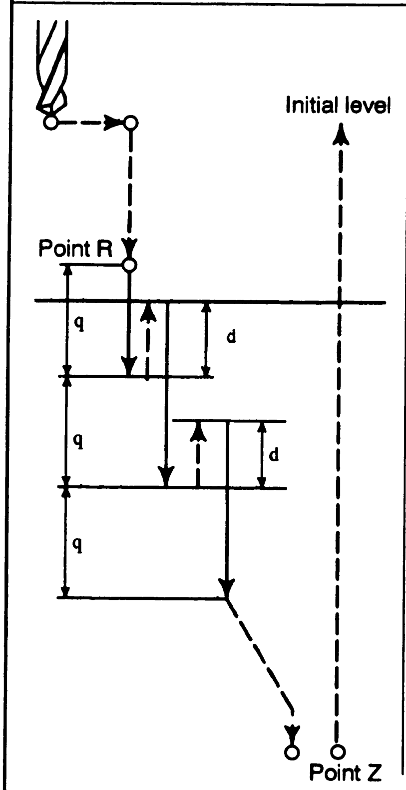
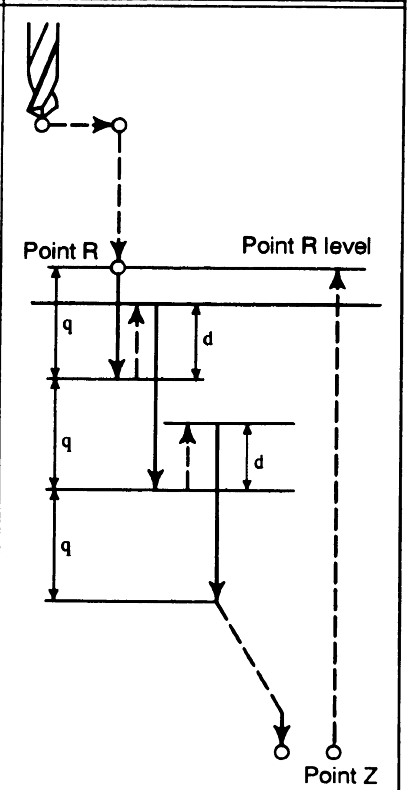
13.1.1 High-speed Peck Drilling Cycle (G73)

This cycle performs high-speed peck drilling. It performs intermittent cutting feed to the bottom of a hole while removing chips from the hole.

Format

G73 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
Q_	:	Depth of cut for each cutting feed
F_	:	Cutting feedrate
K_	:	Number of repeats

G73 (G98)	G73 (G99)
	

Explanations

The high–speed peck drilling cycle performs intermittent feeding along the Z–axis. When this cycle is used, chips can be removed from the hole easily, and a smaller value can be set for retraction. This allows, drilling to be performed efficiently. Set the clearance, d, in parameter 5114. The tool is retracted in rapid traverse.
Before specifying G73, rotate the spindle using a miscellaneous function (M code).
When the G73 code and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, R, drilling is not performed.

• Q/R

Specify Q and R in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.

Examples

M3 S2000 ; Cause the spindle to start rotating.
G90 G99 G73 X300. Y–250. Z–150. R–100. Q15. F120. ;
Position, drill hole 1, then return to point R.
Y–550. ; Position, drill hole 2, then return to point R.
Y–750. ; Position, drill hole 3, then return to point R.
X1000. ; Position, drill hole 4, then return to point R.
Y–550. ; Position, drill hole 5, then return to point R.
G98 Y–750. ; Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ; Return to the reference position return
M5 ; Cause the spindle to stop rotating.

13.1.2 Left–handed Tapping Cycle (G74)

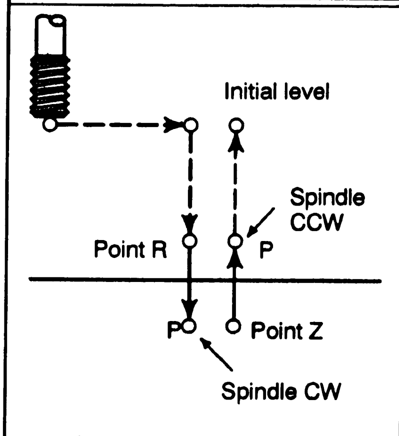
This cycle performs left–handed tapping. In the left–handed tapping cycle, when the bottom of the hole has been reached, the spindle rotates clockwise.

Format

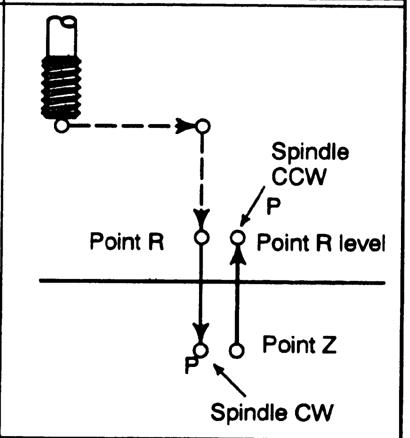
G74 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time
F_	:	Cutting feedrate
K_	:	Number of repeats

G74 (G98)



G74 (G99)



Explanations

Tapping is performed by turning the spindle counterclockwise. When the bottom of the hole has been reached, the spindle is rotated clockwise for retraction. This creates a reverse thread.

Feedrate overrides are ignored during left–handed tapping. A feed hold does not stop the machine until the return operation is completed.

Before specifying G74, use a miscellaneous function (M code) to rotate the spindle counterclockwise.

When the G74 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

- **Axis switching**

Before the drilling axis can be changed, the canned cycle must be canceled.

- **Drilling**

In a block that does not contain X, Y, Z, or R, drilling is not performed.

- **R**

Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

- **Cancel**

Do not specify a group 01 G code (G00 to G03) and G74 in the same block. If they are specified together, G74 is canceled.

Examples

M4 S100 ;	Cause the spindle to start rotating.
G90 G99 G74 X300. Y–250. Z–150. R–120. F120. ;	
	Position, tapping hole 1, then return to point R.
Y–550. ;	Position, tapping hole 2, then return to point R.
Y–750. ;	Position, tapping hole 3, then return to point R.
X1000. ;	Position, tapping hole 4, then return to point R.
Y–550. ;	Position, tapping hole 5, then return to point R.
G98 Y–750. ;	Position, tapping hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

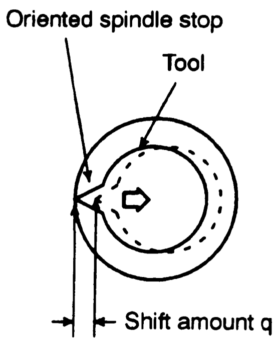
13.1.3 Fine Boring Cycle (G76)

The fine boring cycle bores a hole precisely. When the bottom of the hole has been reached, the spindle stops, and the tool is moved away from the machined surface of the workpiece and retracted.

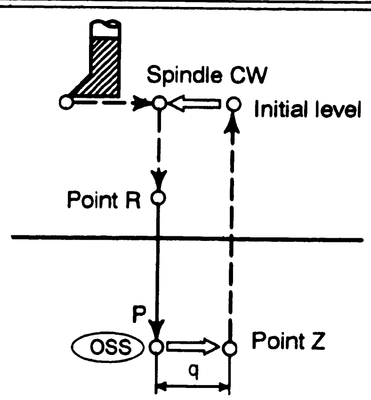
Format

G76 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

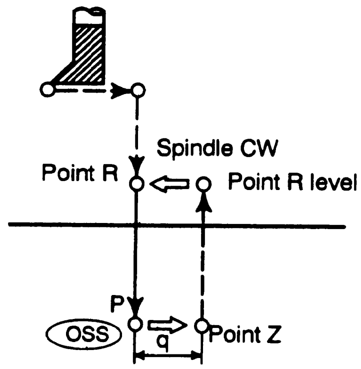
X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
Q_	:	Shift amount at the bottom of a hole
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feedrate
K_	:	Number of repeats



G76 (G98)



G76 (G99)



Note
Q (shift at the bottom of a hole) is a modal value retained within canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

Explanations

When the bottom of the hole has been reached, the spindle is stopped at the fixed rotation position, and the tool is moved in the direction opposite to the tool tip and retracted. This ensures that the machined surface is not damaged and enables precise and efficient boring to be performed.
Before specifying G76, use a miscellaneous function (M code) to rotate the spindle.
When the G76 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

- **Axis switching**

Before the drilling axis can be changed, the canned cycle must be canceled.

- **Boring**

In a block that does not contain X, Y, Z, R, boring is not performed.

- **Q/R**

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in bits 4 (RD1) and 5 (RD2) of parameter 5101. Specify Q and R in a block that performs boring. If they are specified in a block that does not perform boring, they are not stored as modal data.

- **Cancel**

Do not specify a group 01 G code (G00 to G03) and G76 in the same block. If they are specified together, G76 is canceled.

Examples

M3 S500 ;	Cause the spindle to start rotating.
G90 G99 G76 X300. Y–250. R.	Position, bore hole 1, then return to point
Z–150. R–120. Q5.	Orient at the bottom of the hole, then shift by 5 mm.
P1000 F120. ;	Stop at the bottom of the hole for 1 s.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

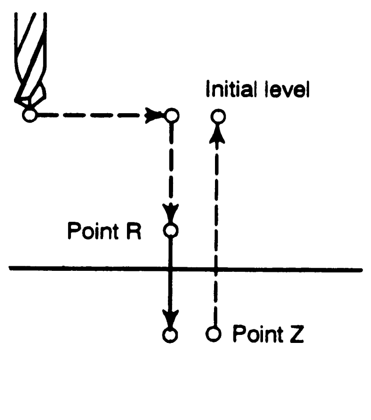
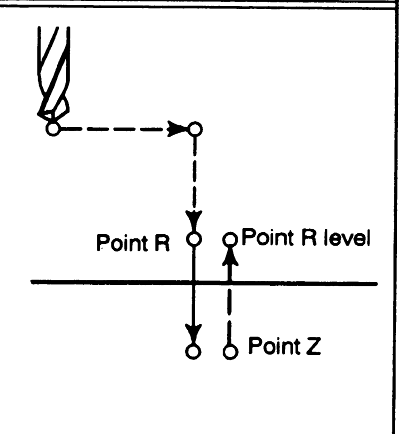
13.1.4 Drilling Cycle, Spot Drilling (G81)

This cycle is used for normal drilling. Cutting feed is performed to the bottom of the hole. The tool is then retracted from the bottom of the hole in rapid traverse.

Format

G81 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
F_	:	Cutting feedrate
K_	:	Number of repeats

G81 (G98)	G81 (G99)
	

Explanations

After positioning along the X– and Y–axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
The tool is then retracted in rapid traverse.
Before specifying G81, use a miscellaneous function (M code) to rotate the spindle.
When the G81 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is performed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, drilling is not performed.

• R

Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G81 in the same block. If they are specified together, G81 is canceled.

Examples

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G81 X300. Y–250. Z–150. R–100. F120. ;	
	Position, drill hole 1, then return to point R.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

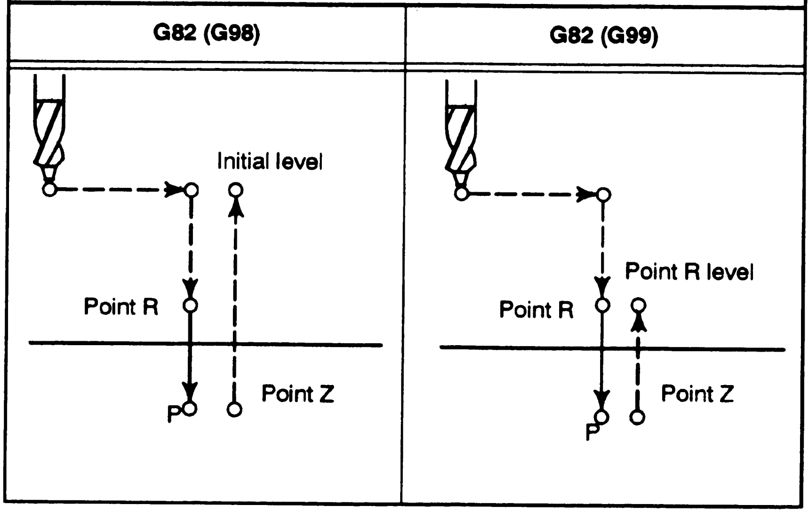
13.1.5 Drilling Cycle Counter Boring Cycle (G82)

This cycle is used for normal drilling.
Cutting feed is performed to the bottom of the hole. At the bottom, a dwell is performed, then the tool is retracted in rapid traverse.
This cycle is used to drill holes more accurately with respect to depth.

Format

G82 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feed rate
K_	:	Number of repeats



Explanations

After positioning along the X– and Y–axes, rapid traverse is performed to point R.
Drilling is then performed from point R to point Z.
When the bottom of the hole has been reached, a dwell is performed. The tool is then retracted in rapid traverse.
Before specifying G82, use a miscellaneous function (M code) to rotate the spindle.
When the G82 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, or R, drilling is not performed.

• R

Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G82 in the same block. If they are specified together, G82 is canceled.

Examples

```
M3 S2000 ;                 Cause the spindle to start rotating.
G90 G99 G82 X300. Y–250. Z–150. R–100. P1000 F120. ;
                           Position, drill hole 2, and dwell for 1 s at the
                           bottom of the hole, then return to point R.
Y–550. ;                   Position, drill hole 2, then return to point R.
Y–750. ;                   Position, drill hole 3, then return to point R.
X1000. ;                   Position, drill hole 4, then return to point R.
Y–550. ;                   Position, drill hole 5, then return to point R.
G98 Y–750. ;               Position, drill hole 6, then return to the initial
                           level.
G80 G28 G91 X0 Y0 Z0 ;     Return to the reference position return
M5 ;                       Cause the spindle to stop rotating.
```

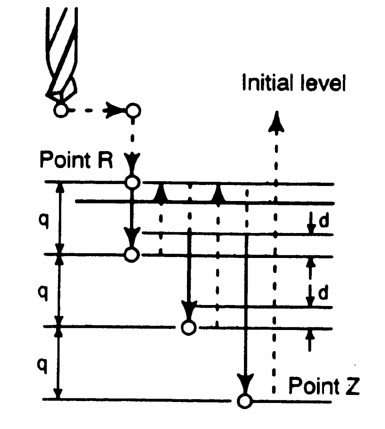
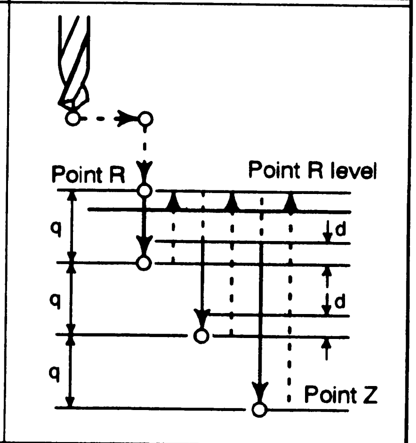
13.1.6 Peck Drilling Cycle (G83)

This cycle performs peck drilling.
It performs intermittent cutting feed to the bottom of a hole while removing shavings from the hole.

Format

G83 X_ Y_ Z_ R_ Q_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
Q_	:	Depth of cut for each cutting feed
F_	:	Cutting feedrate
K_	:	Number of repeats

G83 (G98)	G83 (G99)
	

Explanations

Q represents the depth of cut for each cutting feed. It must always be specified as an incremental value.
In the second and subsequent cutting feeds, rapid traverse is performed up to a point a specified distance d ahead of the point where the previous drilling ended, and cutting feed is performed again. Set d in parameter No. 5115.
Be sure to specify a positive value in Q. Negative values are ignored.
Before specifying G83, use a miscellaneous function (M code) to rotate the spindle.
When the G83 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, or R, drilling is not performed.

• Q/R

Specify Q and R in blocks that perform drilling. If they are specified in a block that does not perform drilling, they cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G83 in the same block. If they are specified together, G83 is canceled.

Examples

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G83 X300. Y–250. Z–150. R–100. Q15. F120. ;	Position, drill hole 1, then return to point R.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

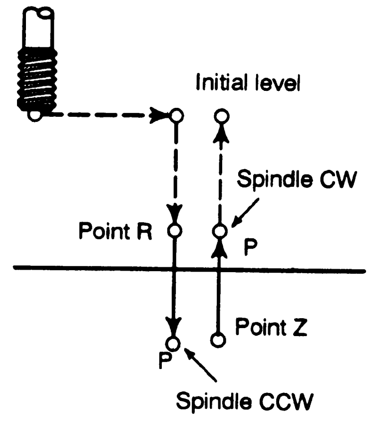
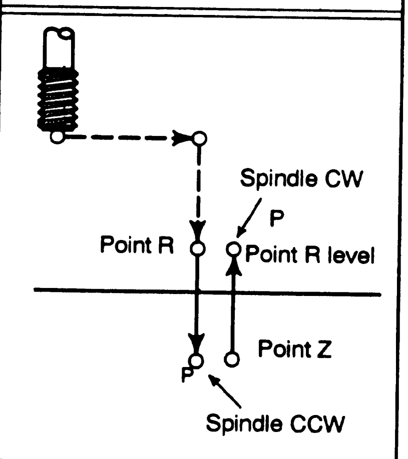
13.1.7 Tapping Cycle (G84)

This cycle performs tapping.
In this tapping cycle, when the bottom of the hole has been reached, the spindle is rotated in the reverse direction.

Format

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time
F_	:	Cutting feedrate
K_	:	Number of repents

G84 (G98)	G84 (G99)
	

Explanations

Tapping is performed by rotating the spindle clockwise. When the bottom of the hole has been reached, the spindle is rotated in the reverse direction for retraction. This operation creates threads.
Feedrate overrides are ignored during tapping. A feed hold does not stop the machine until the return operation is completed.
Before specifying G84, use a miscellaneous function (M code) to rotate the spindle.
When the G84 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When the K is used to specify number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, or R, drilling is not performed.

• R

Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G84 in the same block. If they are specified together, G84 is canceled.

Examples

```
M3 S100 ;                Cause the spindle to start rotating.
G90 G99 G84 X300. Y–250. Z–150. R–120. P300 F120. ;
                         Position, drill hole 1, then return to point R.
Y–550. ;                 Position, drill hole 2, then return to point R.
Y–750. ;                 Position, drill hole 3, then return to point R.
X1000. ;                 Position, drill hole 4, then return to point R.
Y–550. ;                 Position, drill hole 5, then return to point R.
G98 Y–750. ;             Position, drill hole 6, then return to the initial
                         level.
G80 G28 G91 X0 Y0 Z0 ;   Return to the reference position return
M5 ;                     Cause the spindle to stop rotating.
```

13.1.8 Boring Cycle (G85)

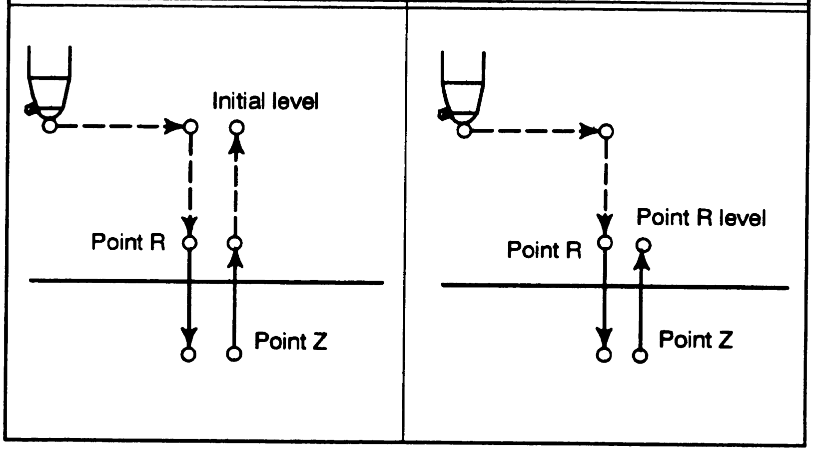
This cycle is used to bore a hole.

Format

G85 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole
R_	: The distance from the initial level to point R level
F_	: Cutting feed rate
K_	: Number of repeats

G85 (G98)	G85 (G99)



Explanations

After positioning along the X– and Y– axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
When point Z has been reached, cutting feed is performed to return to point R.
Before specifying G85, use a miscellaneous function (M code) to rotate the spindle.
When the G85 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, or R, drilling is not performed.

• R

Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G85 in the same block. If they are specified together, G85 is canceled.

Examples

```
M3 S100 ;                Cause the spindle to start rotating.
G90 G99 G85 X300. Y–250. Z–150. R–120. F120. ;
                         Position, drill hole 1, then return to point R.
Y–550. ;                 Position, drill hole 2, then return to point R.
Y–750. ;                 Position, drill hole 3, then return to point R.
X1000. ;                 Position, drill hole 4, then return to point R.
Y–550. ;                 Position, drill hole 5, then return to point R.
G98 Y–750. ;             Position, drill hole 6, then return to the initial
                         level.
G80 G28 G91 X0 Y0 Z0 ;   Return to the reference position return
M5 ;                     Cause the spindle to stop rotating.
```

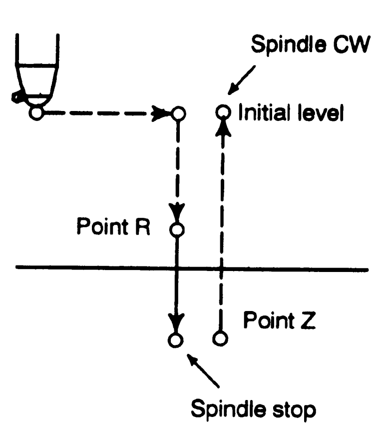
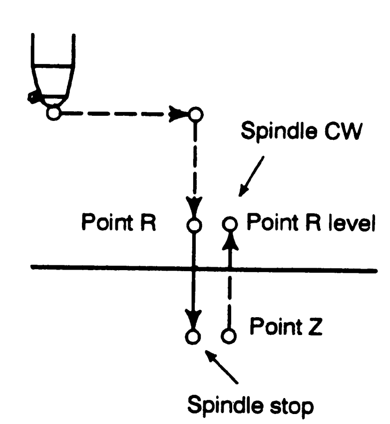
13.1.9 Boring Cycle (G86)

This cycle is used to bore a hole.

Format

G86 X_ Y_ Z_ R_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from point R to the bottom of the hole
R_ : The distance from the initial level to point R level
F_ : Cutting feed rate
K_ : Number of repeats

G86 (G98)	G86 (G99)
	

Explanations

After positioning along the X– and Y–axes, rapid traverse is performed to point R.
Drilling is performed from point R to point Z.
When the spindle is stopped at the bottom of the hole, the tool is retracted in rapid traverse.
Before specifying G86, use a miscellaneous function (M code) to rotate the spindle.
When the G86 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, or R, drilling is not performed.

• R

Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G86 in the same block. If they are specified together, G86 is canceled.

Examples

M3 S2000 ;	Cause the spindle to start rotating.
G90 G99 G86 X300. Y–250. Z–150. R–100. F120. ;	
	Position, drill hole 1, then return to point R.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

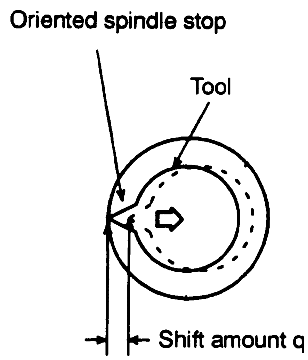
13.1.10 Boring Cycle Back Boring Cycle (G87)

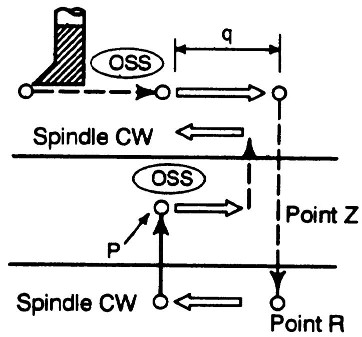
This cycle performs accurate boring.

Format

G87 X_ Y_ Z_ R_ Q_ P_ F_ K_ ;

X_ Y_ : Hole position data
Z_ : The distance from the bottom of the hole to point Z
R_ : The distance from the initial level to point R (the bottom of the hole) level
Q_ : Tool shift amount
P_ : Dwell time
F_ : Cutting feed rate
K_ : Number of repeats



G87 (G98)	G87 (G99)
	Not used

Note
Q (shift at the bottom of a hole) is a modal value retained in canned cycles. It must be specified carefully because it is also used as the depth of cut for G73 and G83.

Explanations

After positioning along the X– and Y–axes, the spindle is stopped at the fixed rotation position. The tool is moved in the direction opposite to the tool tip, positioning (rapid traverse) is performed to the bottom of the hole (point R).
The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise. Boring is performed in the positive direction along the Z–axis until point Z is reached.
At point Z, the spindle is stopped at the fixed rotation position again, the tool is shifted in the direction opposite to the tool tip, then the tool is returned to the initial level. The tool is then shifted in the direction of the tool tip and the spindle is rotated clockwise to proceed to the next block operation.
Before specifying G87, use a miscellaneous function (M code) to rotate the spindle.
When the G87 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation.
The system then proceeds to the next drilling operation. When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Restrictions

- **Axis switching**

Before the drilling axis can be changed, the canned cycle must be canceled.

- **Boring**

In a block that does not contain X, Y, Z, or R, boring is not performed.

- **Q/R**

Be sure to specify a positive value in Q. If Q is specified with a negative value, the sign is ignored. Set the direction of shift in bits 4 (RD1) and 5 (RD2) of parameter 5101. Specify Q and R in a block that performs boring. If they are specified in a block that does not perform boring, they are not stored as modal data.

- **Cancel**

Do not specify a group 01 G code (G00 to G03) and G76 in the same block. If they are specified together, G76 is canceled.

Examples

```
M3 S500 ;                  Cause the spindle to start rotating.
G90 G87 X300. Y–250.       Position, bore hole 1.
    Z–150. R–120. Q5.      Orient at the initial level, then shift by 5 mm.
P1000 F120. ;              Stop at point Z for 1 s.
Y–550. ;                   Position, drill hole 2.
Y–750. ;                   Position, drill hole 3.
X1000. ;                   Position, drill hole 4.
Y–550. ;                   Position, drill hole 5.
Y–750. ;                   Position, drill hole 6
G80 G28 G91 X0 Y0 Z0 ;     Return to the reference position return
M5 ;                       Cause the spindle to stop rotating.
```

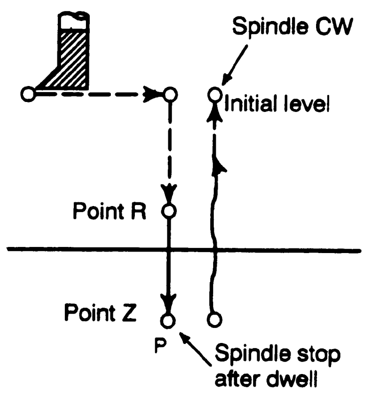
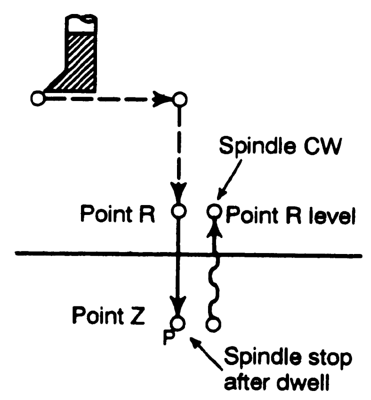
13.1.11 Boring Cycle (G88)

This cycle is used to bore a hole.

Format

G88 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feed rate
K_	:	Number of repeats

G88 (G98)	G88 (G99)
	

Explanations

After positioning along the X– and Y–axes, rapid traverse is performed to point R. Boring is performed from point R to point Z. When boring is completed, a dwell is performed, then the spindle is stopped. The tool is manually retracted from the bottom of the hole (point Z) to point R. At point R, the spindle is rotated clockwise, and rapid traverse is performed to the initial level.

Before specifying G88, use a miscellaneous function (M code) to rotate the spindle.

When the G88 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.

When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.

When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, or R, drilling is not performed.

• R

Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G88 in the same block. If they are specified together, G88 is canceled.

Examples

```
M3 S2000 ;                Cause the spindle to start rotating.
G90 G99 G88 X300. Y–250. Z–150. R–100. P1000 F120. ;
                          Position, drill hole 1, return to point R
                          then stop at the bottom of the hole for 1 s.
Y–550. ;                  Position, drill hole 2, then return to point R.
Y–750. ;                  Position, drill hole 3, then return to point R.
X1000. ;                  Position, drill hole 4, then return to point R.
Y–550. ;                  Position, drill hole 5, then return to point R.
G98 Y–750. ;              Position, drill hole 6, then return to the initial
                          level.
G80 G28 G91 X0 Y0 Z0 ;    Return to the reference position return
M5 ;                      Cause the spindle to stop rotating.
```

13.1.12 Boring Cycle (G89)

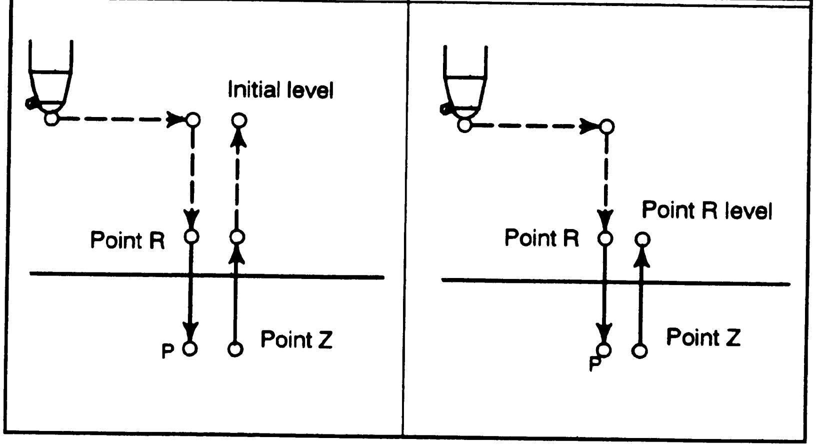
This cycle is used to bore a hole.

Format

G89 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of a hole
F_	:	Cutting feed rate
K_	:	Number of repeats

G89 (G98)	G89 (G99)



Explanations

This cycle is almost the same as G85. The difference is that this cycle performs a dwell at the bottom of the hole.
Before specifying G89, use a miscellaneous function (M code) to rotate the spindle.
When the G89 command and an M code are specified in the same block, the M code is executed at the time of the first positioning operation. The system then proceeds to the next drilling operation.
When K is used to specify the number of repeats, the M code is executed for the first hole only; for the second and subsequent holes, the M code is not executed.
When a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled.

• Drilling

In a block that does not contain X, Y, Z, or R, drilling is not performed.

• R

Specify R in blocks that perform drilling. If it is specified in a block that does not perform drilling, it cannot be stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.

Examples

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G89 X300. Y–250. Z–150. R–120. P1000 F120. ;	
	Position, drill hole 1, return to point R then stop at the bottom of the hole for 1 s.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return
M5 ;	Cause the spindle to stop rotating.

13.1.13 Canned Cycle Cancel (G80)

G80 cancels canned cycles.

Format

```
G80 ;
```

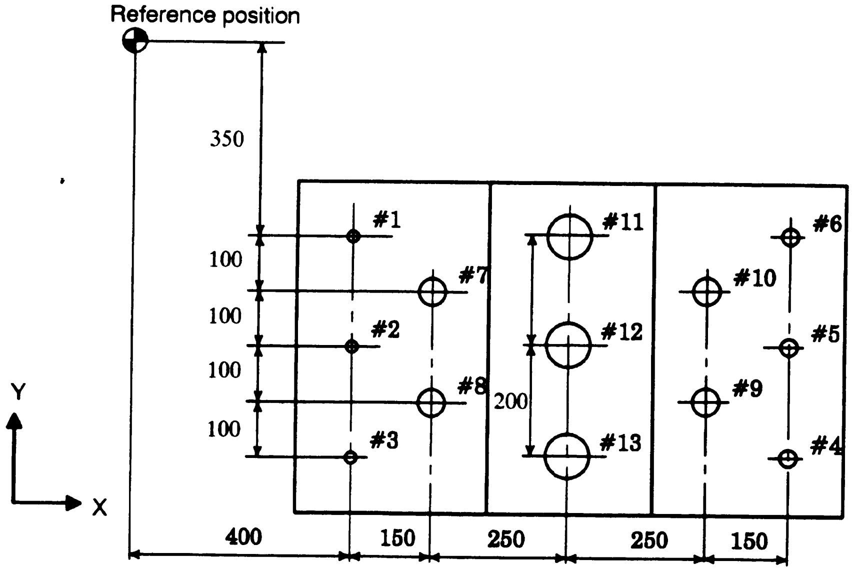
Explanations

All canned cycles are canceled to perform normal operation. Point R and point Z are cleared. This means that R = 0 and Z = 0 in incremental mode. Other drilling data is also canceled (cleared).

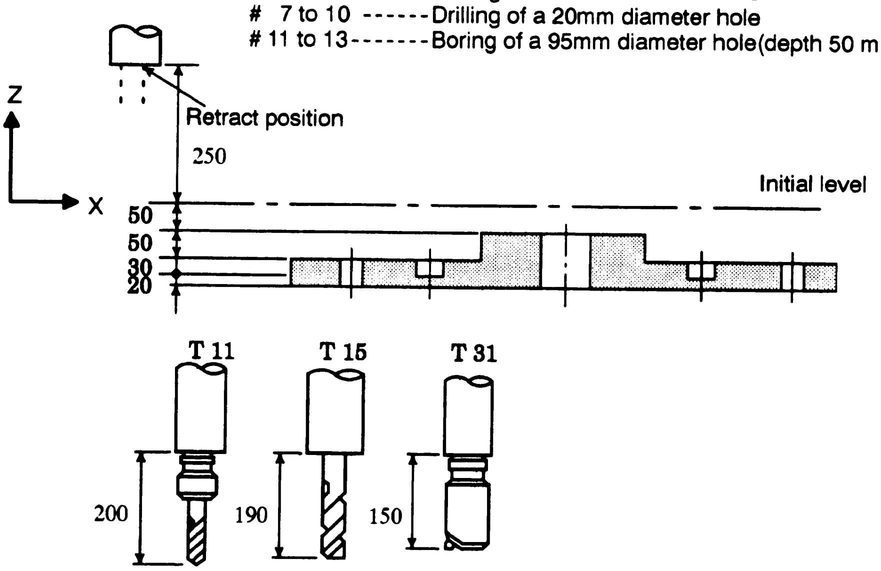
Examples

M3 S100 ;	Cause the spindle to start rotating.
G90 G99 G88 X300. Y–250. Z–150. R–120. F120. ;	
	Position, drill hole 1, then return to point R.
Y–550. ;	Position, drill hole 2, then return to point R.
Y–750. ;	Position, drill hole 3, then return to point R.
X1000. ;	Position, drill hole 4, then return to point R.
Y–550. ;	Position, drill hole 5, then return to point R.
G98 Y–750. ;	Position, drill hole 6, then return to the initial level.
G80 G28 G91 X0 Y0 Z0 ;	Return to the reference position return, canned cycle cancel
M5 ;	Cause the spindle to stop rotating.

Program example using tool length offset and canned cycles



1 to 6-------Drilling of a 10mm diameter hole
7 to 10 ------Drilling of a 20mm diameter hole
11 to 13-------Boring of a 95mm diameter hole(depth 50 mm)



Offset value +200.0 is set in offset No.11, +190.0 is set in offset No.15, and +150.0 is set in offset No.31

Program example

;		
N001	G92X0Y0Z0;	Coordinate setting at reference position
N002	G90 G00 Z250.0 T11 M6;	Tool change
N003	G43 Z0 H11;	Initial level, tool length offset
N004	S30 M3	Spindle start
N005	G99 G81X400.0 R Y–350.0	
	Z–153,0R–97.0 F120;	Positioning, then #1 drilling
N006	Y–550.0;	Positioning, then #2 drilling and point R level return
N007	G98Y–750.0;	Positioning, then #3 drilling and initial level return
N008	G99X1200.0;	Positioning, then #4 drilling and point R level return
N009	Y–550.0;	Positioning, then #5 drilling and point R level return
N010	G98Y–350.0;	Positioning, then #6 drilling and initial level return
N011	G00X0Y0M5;	Reference position return, spindle stop
N012	G49Z250.0T15M6;	Tool length offset cancel, tool change
N013	G43Z0H15;	Initial level, tool length offset
N014	S20M3;	Spindle start
N015	G99G82X550.0Y–450.0	Positioning, then #7 drilling, point R level return
	Z–130.0R–97.0P300F70;	
N016	G98Y–650.0;	Positioning, then #8 drilling, initial level return
N017	G99X1050.0;	Positioning, then #9 drilling, point R level return
N018	G98Y–450.0;	Positioning, then #10 drilling, initial level return
N019	G00X0Y0M5;	Reference position return, spindle stop
N020	G49Z250.0T31M6;	Tool length offset cancel, tool change
N021	G43Z0H31;	Initial level, tool length offset
N022	S10M3;	Spindle start
N023	G85G99X800.0Y–350.0	Positioning, then #11 drilling, point R level return
	Z–153.0R47.0F50;	
N024	G91Y–200.0K2;	Positioning, then #12, 13 drilling. point R level return
N025	G28X0Y0M5;	Reference position return, spindle stop
N026	G49Z0;	Tool length offset cancel
N027	M0;	Program stop

13.2 RIGID TAPPING

The tapping cycle (G84) and left–handed tapping cycle (G74) may be performed in standard mode or rigid tapping mode.

In standard mode, the spindle is rotated and stopped along with a movement along the tapping axis using miscellaneous functions M03 (rotating the spindle clockwise), M04 (rotating the spindle counterclockwise), and M05 (stopping the spindle) to perform tapping.In rigid mode, tapping is performed by controlling the spindle motor as if it were a servo motor and by interpolating between the tapping axis and spindle.

When tapping is performed in rigid mode, the spindle rotates one turn every time a certain feed (thread lead) which takes place along the tapping axis. This operation does not vary even during acceleration or deceleration.

Rigid mode eliminates the need to use a floating tap required in the standard tapping mode, thus allowing faster and more precise tapping.

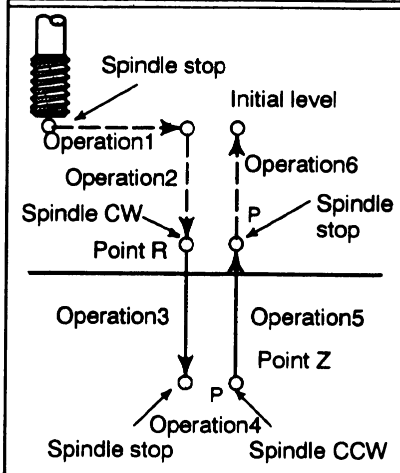
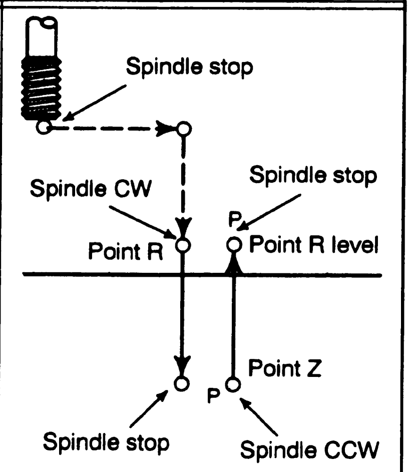
13.2.1 Rigid Tapping (G84)

When the spindle motor is controlled in rigid mode as if it were a servo motor, a tapping cycle can be sped up.

Format

G84 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole and the position of the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of the hole and at point R when a return is made
F_	:	Cutting feedrate
K_	:	Number of repeats

G84(G98)	G84(G99)
	

Explanations

After positioning along the X– and Y–axes, rapid traverse is performed to point R.

Tapping is performed from point R to point Z. When tapping is completed, the spindle is stopped and a dwell is performed. Then the spindle is then rotated in the reverse direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed.

While tapping is being performed, the feedrate override and spindle override are assumed to be 100%.

However, the speed for extraction (operation 5) can be overridden by up to 200% depending on the setting at bit 4 (DOV) of parameter 5200 and parameter 5211.

- **Rigid mode**

Rigid mode can be specified using any of the following methods:

·**Specify M29 S***** before a tapping command.**
·**Specify M29 S***** in a block which contains a tapping command.**
·**Specify G84 for rigid tapping (parameter No. 5200 #0 (G84) = 1).**

• Thread lead

In feed–per–minute mode, the thread lead is obtained from the expression, feedrate × spindle speed.

• Tool length offset

If a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm (No. 206) is issued.

• S command

If a speed higher than the maximum speed for the gear being used is specified, alarm (No. 200) is issued.

• F command

If a value exceeding the upper limit of cutting feedrate is specified, alarm (No. 011) is issued.

• Unit of F

	Metric input	Inch input	Remarks
Unit of F	1 mm/min	0.01 inch/min	Decimal point programming allowed

• M29

If an S command and axis movement are specified between M29 and G84, alarm (No. 203) is issued. If M29 is specified in a tapping cycle, alarm (No. 204) is issued.

• R

Specify R in a block that performs drilling. If R is specified in a non–drilling block, it is not stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.

Examples

Z–axis feedrate 1000 mm/min
Spindle speed 1000 rpm
Thread lead 1.0 mm

G94 ;	Specify a feed–per–minute command.
G00 X120.0 Y100.0 ;	Positioning
M29 S1000 ;	Rigid mode specification
G84 Z–100.0 R–20.0 F1000 ;	Rigid tapping

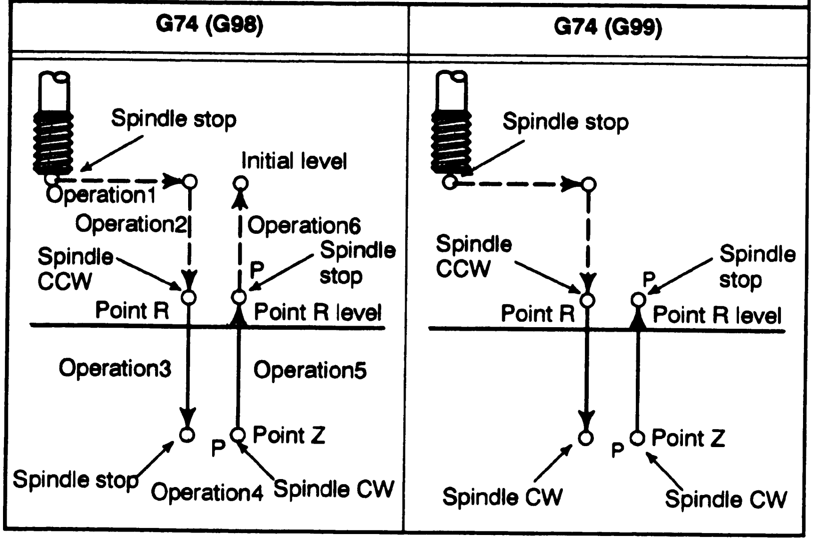
13.2.2 Left–handed Rigid Tapping Cycle (G74)

When the spindle motor is controlled in rigid mode as if it were a servo motor, tapping cycles can be sped up.

Format

G74 X_ Y_ Z_ R_ P_ F_ K_ ;

X_ Y_	:	Hole position data
Z_	:	The distance from point R to the bottom of the hole and the position of the bottom of the hole
R_	:	The distance from the initial level to point R level
P_	:	Dwell time at the bottom of the hole and at point R when return is made.
F_	:	Cutting feedrate
K_	:	Number of repeats



Explanations

After positioning along the X– and Y–axes, rapid traverse is performed to point R.
Tapping is performed from point R to point Z. When tapping is completed, the spindle is stopped and a dwell is performed. Then the spindle is then rotated in the normal direction, the tool is retracted to point R, then the spindle is stopped. Rapid traverse to initial level is then performed.
While tapping is being performed, the feedrate override and spindle override are assumed to be 100%.
However, the speed for extraction (operation 5) can be overridden by up to 200% depending on the setting at bit 4 (DOV) of parameter 5200 and parameter 5211.

• Rigid mode

Rigid mode can be specified using any of the following methods:
·Specify M29 S*** before a tapping command.**
·Specify M29 S*** in a block which contains a tapping command.**
·Specify G84 for rigid tapping. (parameter No. 5200#0 = 1).

• Thread lead

In feed–per–minute mode, the thread lead is obtained from the expression, feedrate × spindle speed.

• Tool length offset

If a tool length offset (G43, G44, or G49) is specified in the canned cycle, the offset is applied at the time of positioning to point R.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm (No. 206) is issued.

• S command

Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm (No. 200).

• F command

Specifying a value that exceeds the upper limit of cutting feedrate causes alarm (No. 011).

• Unit of F

	Metric input	Inch input	Remarks
Unit of F	1 mm/min	0.01 inch/min	Decimal point programming allowed

• M29

Specifying an S command or axis movement between M29 and G84 causes alarm (No. 203).
Then, specifying M29 in the tapping cycle causes alarm (No. 204).

• R

Specify R in a block that performs drilling. If R is specified in a non–drilling block, it ss not stored as modal data.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.

Examples

Z–axis feedrate 1000 mm/min
Spindle speed 1000 rpm
Thread lead 1.0 mm

G94 ;	Specify a feed–per–minute command.
G00 X120.0 Y100.0 ;	Positioning
M29 S1000 ;	Rigid mode specification
G84 Z–100.0 R–20.0 F1000 ;	Rigid tapping

13.2.3 Peck Rigid Tapping Cycle (G84 or G74)

Tapping a deep hole in rigid tapping mode may be difficult due to chips sticking to the tool or increased cutting resistance. In such cases, the peck rigid tapping cycle is useful.
In this cycle, cutting is performed several times until the bottom of the hole is reached. Two peck tapping cycles are available: High–speed peck tapping cycle and standard peck tapping cycle. These cycles are selected using the PCP bit (bit 5) of parameter 5200.

Format

G84 (or G74) X_ Y_ Z_ R_ P_ Q_ F_ K_ ;

X_ Y_	: Hole position data
Z_	: The distance from point R to the bottom of the hole and the position of the bottom of the hole
R_	: The distance from the initial level to point R level
P_	: Dwell time at the bottom of the hole and at point R when a return is made
Q_	: Depth of cut for each cutting feed
F_	: The cutting feedrate
K_	: Number of repeats

G84, G74 (G98) | G84, G74 (G99)

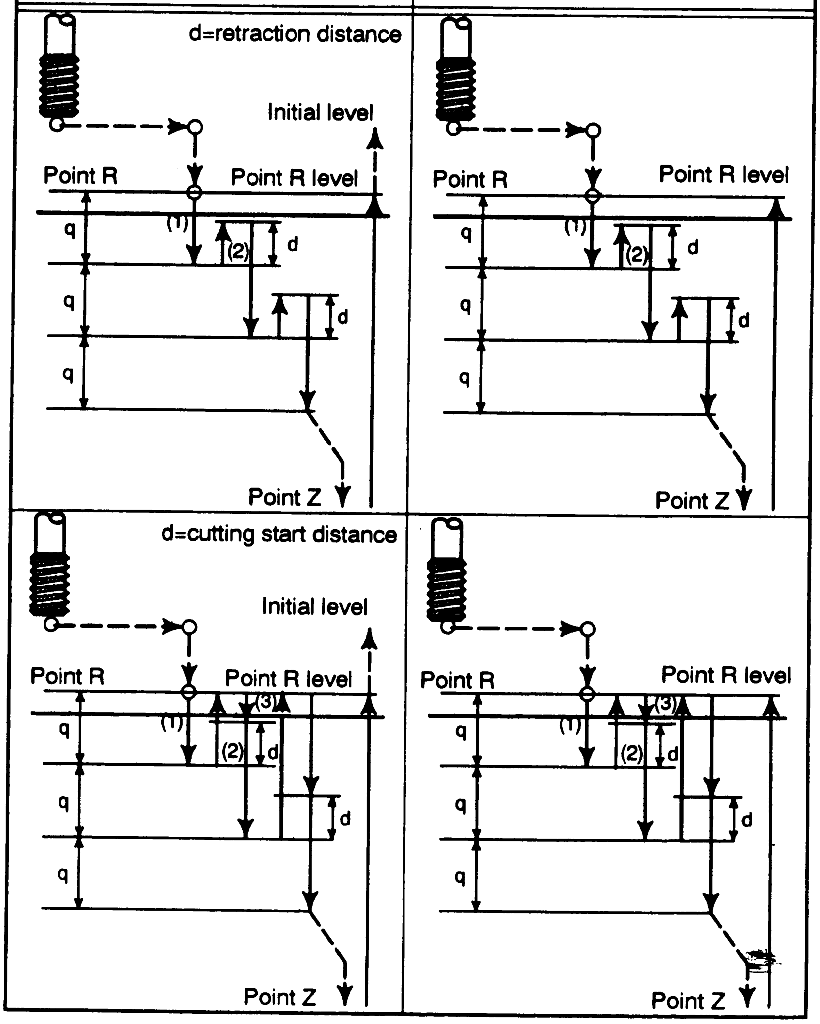
·High–speed peck tapping cycle
(Parameter PCP(No.5200#5=0))

(1) The tool operates at a normal cutting feedrate. The normal time constant is used.
(2) Retraction can be overridden. The retraction time constant is used.

·Peck tapping cycle
(Parameter PCP(No.5200#5=1))

(1) The tool operates at a normal cutting feedrate. The normal time constant is used.
(2) Retraction can be overridden. The retraction time constant is used.
(3) Retraction can be overridden. The normal time constant is used.

During a rigid tapping cycle, inposition check is performed at the end of each operation of (1) and (2) in the peck tapping cycle.



Explanations

• High–speed peck tapping cycle

After positioning along the X– and Y–axes, rapid traverse is performed to point R. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then the tool is retracted by distance d. The DOV bit (bit 4) of parameter 5200 specifies whether retraction can be overridden or not. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.
Set the retraction distance, d, in parameter 5213.

• Peck tapping cycle

After positioning along the X– and Y–axes, rapid traverse is performed to point R level. From point R, cutting is performed with depth Q (depth of cut for each cutting feed), then a return is performed to point R. The DOV bit (bit 4) of parameter 5200 specifies whether the retraction can be overridden or not. Rapid traverse is performed from point R to a position distance d from the end point of the last cutting, which is where cutting is restarted. For this rapid traverse, the specification of the DOV bit (bit 4) of parameter 5200 is also valid. When point Z has been reached, the spindle is stopped, then rotated in the reverse direction for retraction.
Set d (distance to the point at which cutting is started) in parameter 5213.

Limitations

• Axis switching

Before the drilling axis can be changed, the canned cycle must be canceled. If the drilling axis is changed in rigid mode, alarm (No. 206) is issued.

• S command

Specifying a rotation speed exceeding the maximum speed for the gear used causes alarm (No. 200).

• F command

Specifying a value that exceeds the upper limit of cutting feedrate causes alarm (No. 011).

• Unit of F

	Metric input	Inch input	Remarks
Unit of F	1 mm/min	0.01 inch/min	Decimal point programming allowed

• M29

Specifying an S command or axis movement between M29 and G84 causes alarm (No. 203).
Then, specifying M29 in the tapping cycle causes alarm (No. 204).

• Q/R

Specify Q and R in a block that performs drilling. If they are specified in a block that does not perform drilling, they are not stored as modal data. When Q0 is specified, the peck rigid tapping cycle is not performed.

• Cancel

Do not specify a group 01 G code (G00 to G03) and G73 in the same block. If they are specified together, G73 is canceled.

13.2.4 Canned Cycle Cancel (G80)

The rigid tapping canned cycle is canceled. For how to cancel this cycle, see Section 13.1.13.

13.3 EXTERNAL MOTION FUNCTION (G81)

Upon completion of positioning in each block in the program, an external operation function signal can be output to allow the machine to perform specific operation.
Concerning this operation, refer to the manual supplied by the machine tool builder.

Format

G81 IP_ ; (IP_ Axis move command)

Explanations

Every time positioning for the IP_ move command is completed, the CNC sends a external operation function signal to the machine. An external operation signal is output for each positioning operation until canceled by G80 or a group 01 G code.

Restrictions

- **A block without X or Y axis**

No external operation signals are output during execution of a block that contains neither X nor Y.

- **Reference to canned cycle G81**

G81 can also be used for a drilling canned cycle (see 13.1.4). Whether G81 is to be used for an external motion function or for a drilling canned cycle is specified with [EXEC], bit 1 of parameter No. 5101.

14 COMPENSATION FUNCTION

General

This chapter describes the following compensation functions:

TOOL LENGTH OFFSET (G43, G44, G49) Sec.14.1
CUTTER COMPENSATION C (G40–G42) Sec.14.2, 14.3
TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10) Sec.14.4
SCALING (G50, G51) Sec.14.5
COORDINATE SYSTEM ROTATION (G68, G69) Sec.14.6

14.1 TOOL LENGTH OFFSET (G43,G44,G49)

This function can be used by setting the difference between the tool length assumed during programming and the actual tool length of the tool used into the offset memory. It is possible to compensate the difference without changing the program.

Specify the direction of offset with G43 or G44. Select a tool length offset value from the offset memory by entering the corresponding address and number (H code).

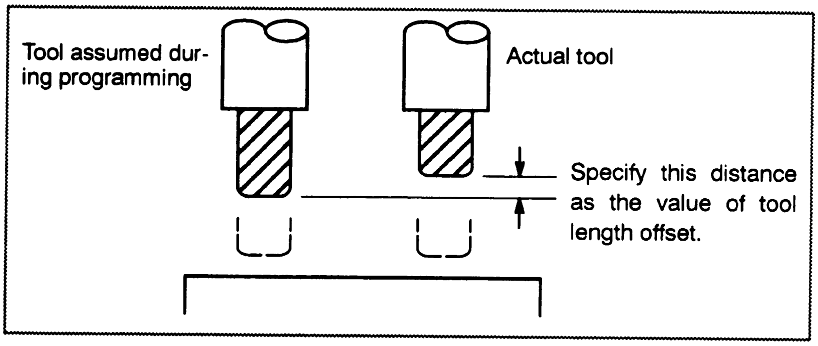


Fig14.1(a) Tool length offset

The following three methods of tool length offset can be used, depending on the axis along which tool length offset can be made.

·**Tool length offset A**

Compensates for the difference in tool length along the Z–axis.

·**Tool length offset B**

Compensates for the difference in tool length along the X–,Y–, or Z–axis.

·**Tool length offset C**

Compensates for the difference in tool length along a specified axis.

Format

<table>
<tr><td>Tool length offset A</td><td>G43 Z_ H_ ;
G44 Z_ H_ ;</td><td rowspan="4">Explanation of each address
G43 : Positive offset
G44 : Negative offset
G17 : XY plane selection
G18 : ZX plane selection
G19 : YZ plane selection
α : Address of a specified axis
H : Address for specifying the tool length offset value</td></tr>
<tr><td>Tool length offset B</td><td>G17 G43 Z_ H_ ;
G17 G44 Z_ H_ ;
G18 G43 Y_ H_ ;
G18 G44 Y_ H_ ;
G19 G43 X_ H_ ;
G19 G44 X_ H_ ;</td></tr>
<tr><td>Tool length offset C</td><td>G43 α_ H_ ;
G44 α_ H_ ;</td></tr>
<tr><td>Tool length offset cancel</td><td>G49 ; or H0 ;</td></tr>
</table>

Explanations

• Selection of tool length offset

Select tool length offset A, B, or C, by setting bits 0 and 1 of parameter No. 5001.

• Direction of the offset

When G43 is specified, the tool length offset value (stored in offset memory) specified with the H code is added to the coordinates of the end position specified by a command in the program. When G44 is specified, the same value is subtracted from the coordinates of the end position. The resulting coordinates indicate the end position after compensation, regardless of whether the absolute or incremental mode is selected.
If movement along an axis is not specified, the system assumes that a move command that causes no movement is specified. When a positive value is specified for tool length offset with G43, the tool is moved accordingly in the positive direction. When a positive value is specified with G44, the tool is moved accordingly in the negative direction. When a negative value is specified, the tool is moved in the opposite direction. G43 and G44 are modal G codes. They are valid until another G code belonging to the same group is used.

• Specification of the tool length offset value

The tool length offset value assigned to the number (offset number) specified in the H code is selected from offset memory and added to or subtracted from the coordinates specified by a command in the program. The tool length offset value may be set in the offset memory through the CRT/MDI panel.
The range of values that can be set as the tool length offset value is as follows.

	Metric input	Inch input
Tool length offset value	0A±999.999mm	0A±99.9999inch

Notes

1. The tool length offset value corresponding to offset No. 0 , that is, H0 always means 0. It is impossible to set any other tool length offset value to H0.
2. When the tool length offset value is changed due to a change of the offset number, the offset value changes to the new tool length offset value, the new tool length offset value is not added to the old tool length offset value.
   H1 : tool length offset value 20.0
   H2 : tool length offset value 30.0
   G90 G43 Z100.0 H1 ; Z will move to 120.0
   G90 G43 Z100.0 H2 ; Z will move to 130.0
3. When the tool length offset is used, specify the tool length offset with H code and the cutter compensation with D code by a parameter OFH (No. 5001#2) set to 0.

- **Performing tool length offset along two or more axes**

Tool length offset B can be executed along two or more axes when the axes are specified in two or more blocks.

Offset in X and Y axes.

G19 G43 H _ ; Offset in X axis
G18 G43 H _ ; Offset in Y axis
(Offsets in X and Y axes are performed)

If the TAL bit (bit 3 of parameter No. 5001) is set to 1, an alarm will not occur even when tool length offset C is executed along two or more axes at the same time.

- **Tool length offset cancel**

To cancel tool length offset, specify G49 or H0. After G49 or H0 is specified, the system immediately cancels the offset mode.

- **G28 in tool length offset mode**

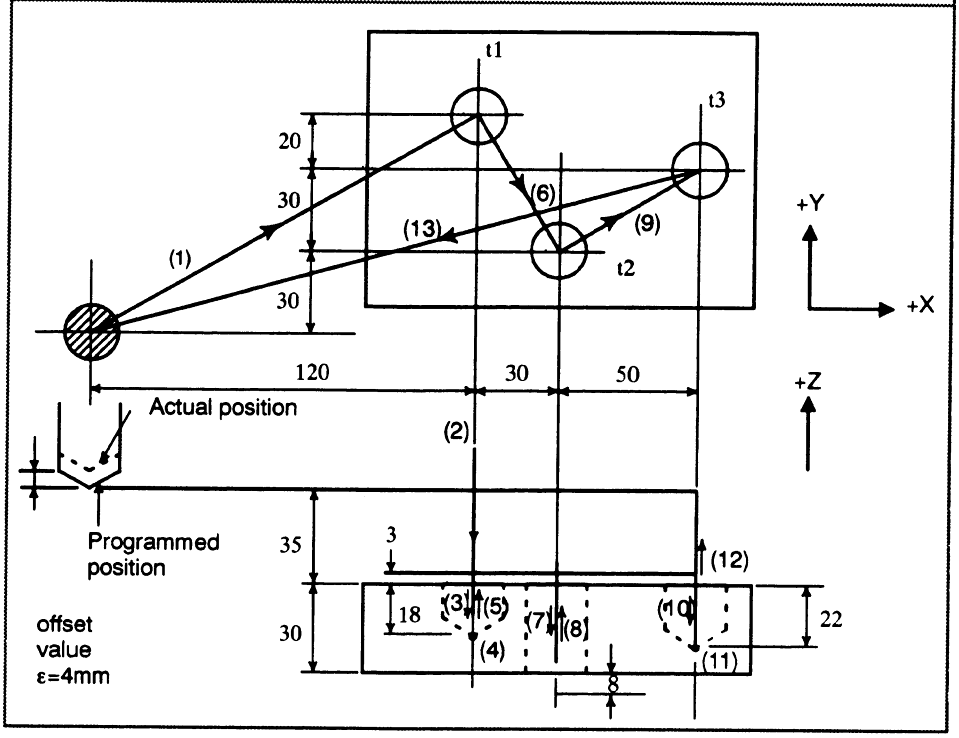
When G28 (automatic reference position return) is executed in tool length offset mode, the tool length offset vector is canceled.
In tool length offset A and B, the canceled tool length offset vector is recovered in the next block if the EVO bit (bit 6 of parameter No. 5001) is set to 1.

Note

· After tool length offset B is executed along two or more axes, offset along all the axes is canceled by specifying G49. If H0 is specified, only offset along an axis perpendicular to the specified plane is canceled.

Examples

Tool length offset (In boring holes No.1, 2, and 3)



·**Program**

H1=–4.0 (Tool length offset value)

N1 G91 G00 X120.0 Y80.0 ; . . . . (1)
N2 G43 Z–32.0 H1 ; . . . . (2)
N3 G01 Z–21.0 F1000 ; . . . . (3)
N4 G04 P2000 ; . . . . (4)
N5 G00 Z21.0 ; . . . . (5)
N6 X30.0 Y–50.0 ; . . . . (6)
N7 G01 Z–41.0 ; . . . . (7)
N8 G00 Z41.0 ; . . . . (8)
N9 X50.0 Y30.0 ; . . . . (9)
N10 G01 Z–25.0 ; . . . . (10)
N11 G04 P2000 ; . . . . (11)
N12 G00 Z57.0 H0 ; . . . . (12)
N13 X–200.0 Y–60.0 ; . . . . (13)
N14 M2 ;

14.2 OVERVIEW OF CUTTER COMPENSATION C (G40 – G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 14.2 (a)).
To make an offset as large as the radius of the tool, CNC first creates an offset vector with a length equal to the radius of the tool (start–up). The offset vector is perpendicular to the tool path. The tail of the vector is on the workpiece side and the head positions to the center of the tool.
If a linear interpolation or circular interpolation command is specified after start–up, the tool path can be shifted by the length of the offset vector during machining.
To return the tool to the start position at the end of machining, cancel the cutter compensation mode.

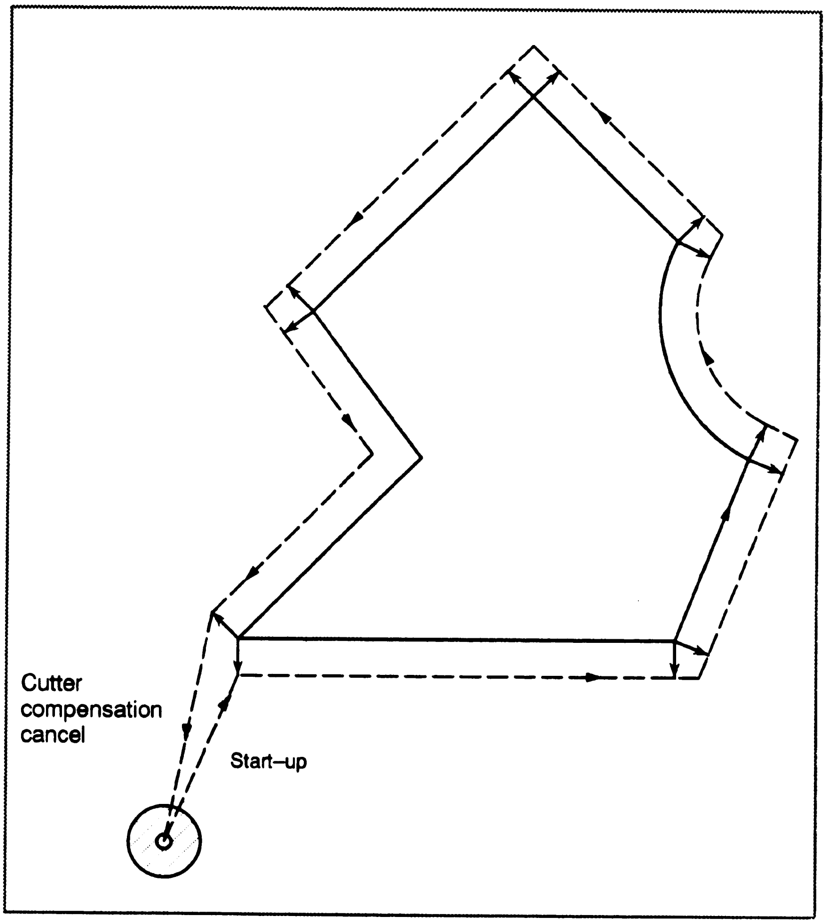


Fig. 14.2 (a) Outline of Cutter Compensation C

Format

·Start up (Tool compensation start)

G00(or G01)G41(or G42) IP_ D_ ;

G41 : Cutter compensation left (Group07)
G42 : Cutter compensation right (Group07)
IP_ : Command for axis movement
D_ : Code for specifying as the cutter compensation value(1–2digits) (D code)

·Cutter compensation cancel (offset mode cancel)

G40 IP_ ;

G40 : Cutter compensation cancel(Group 07) (Offset mode cancel)
IP_ : Command for axis movement

·Selection of the offset plane

Offset plane	Command for plane selection	IP_
XY	**G17 ;**	X_Y_
ZX	**G18 ;**	X_Z_
YZ	**G19 ;**	Y_Z_

Explanations

• Offset cancel mode

At the beginning when power is applied the control is in the cancel mode. In the cancel mode, the vector is always 0, and the tool center path coincides with the programmed path.

• Start Up

When a cutter compensation command (G41 or G42, nonzero dimension words in the offset plane, and D code other than D0) is specified in the offset cancel mode, the CNC enters the offset mode.
Moving the tool with this command is called start–up.
Specify positioning (G00) or linear interpolation (G01) for start–up. If circular interpolation (G02, G03) is specified, alarm 34 occurs.
When processing the start–up block and subsequent blocks, the CNC prereads two blocks. The second preread block is not indicated.

• Offset mode

In the offset mode, compensation is accomplished by positioning (G00), linear interpolation (G01), or circular interpolation (G02, G03). If two or more blocks that do not move the tool (miscellaneous function, dwell, etc.) are processed in the offset mode, the tool will make either an excessive or insufficient cut. If the offset plane is switched in the offset mode, alarm 37 occurs and the tool is stopped.

• Offset mode cancel

In the offset mode, when a block which satisfies any one of the following conditions is executed, the equipment enters the offset cancel mode, and the action of this block is called the offset cancel.

1. **G40 has been commanded.**
2. **0 has been commanded as the offset number for cutter compensation.**

When performing offset cancel, circular arc commands (G02 and G03) are not available. If a circular arc is commanded, an alarm (No. 034) is generated and the tool stops.
In the offset cancel, the control executes the instructions in that block and the block in the cutter compensation buffer. In the meantime, in the case of a single block mode, after reading one block, the control executes it and stops. By pushing the cycle start button once more, one block is executed without reading the next block.
Then the control is in the cancel mode, and normally, the block to be executed next will be stored in the buffer register and the next block is not read into the buffer for cutter compensation.

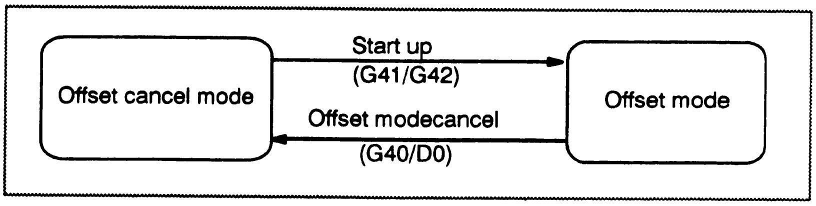


Fig. 14.2 (b) Changing the offset mode

• Change of the Cutter compensation value

In general, the cutter compensation value shall be changed in the cancel mode, when changing tools. If the cutter compensation value is changed in offset mode, the vector at the end point of the block is calculated for the new cutter compensation value.

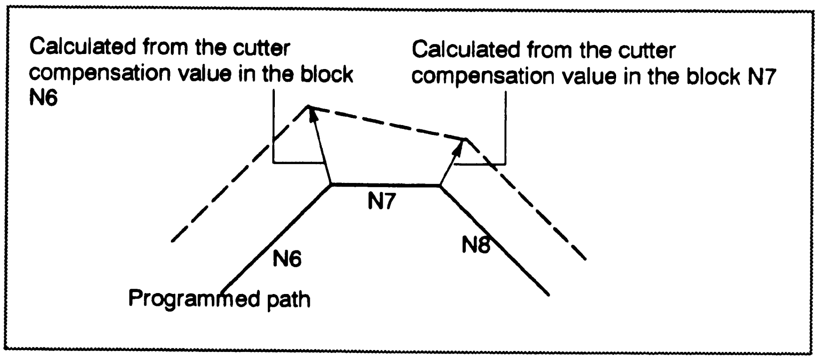


Fig. 14.2 (c) Changing the Cutter Compensation Value

- **Positive/negative cutter compensation value and tool center path**

If the offset amount is negative (–), distribution is made for a figure in which G41's and G42's are all replaced with each other on the program. Consequently, if the tool center is passing around the outside of the workpiece, it will pass around the inside, and vice versa.
The figure below shows one example. Generally, the offset amount is programmed to be positive (+).
When a tool path is programmed as in ((1)), if the offset amount is made negative (–), the tool center moves as in ((2)), and vice versa. Consequently, the same tape permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the offset amount. Applicable if start–up and cancel is A type. (See subsec. 14.3.2 and 14.3.4)

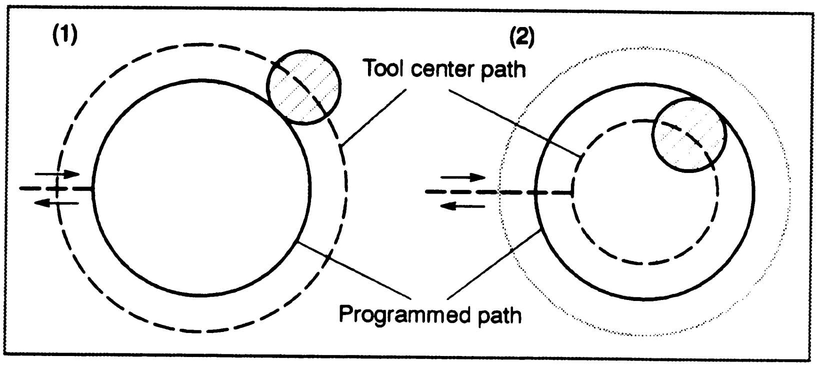


Fig. 14.2 (d) Tool Center Paths when Positive and Negative Cutter Compensation Values are Specified

- **Cutter compensation value setting**

Assign a cutter compensation values to the D codes on the CRT/MDI panel. The table below shows the range in which cutter compensation values can be specified.

	mm input	inch input
Cutter compensation value	0–±999.999mm	0–±99.9999inch

Notes
1. The cutter compensation value corresponding to offset No. 0, that is, D0 always means 0. It is impossible to set D0 to any other offset amount.
2. Cutter compensation C can be specified by H code with parameter OFH (No. 5001 #2) set to 1.

- **Offset vector**

The offset vector is the two dimensional vector that is equal to the cutter compensation value assigned by D code. It is calculated inside the control unit, and its direction is up–dated in accordance with the progress of the tool in each block.
The offset vector is deleted by reset.

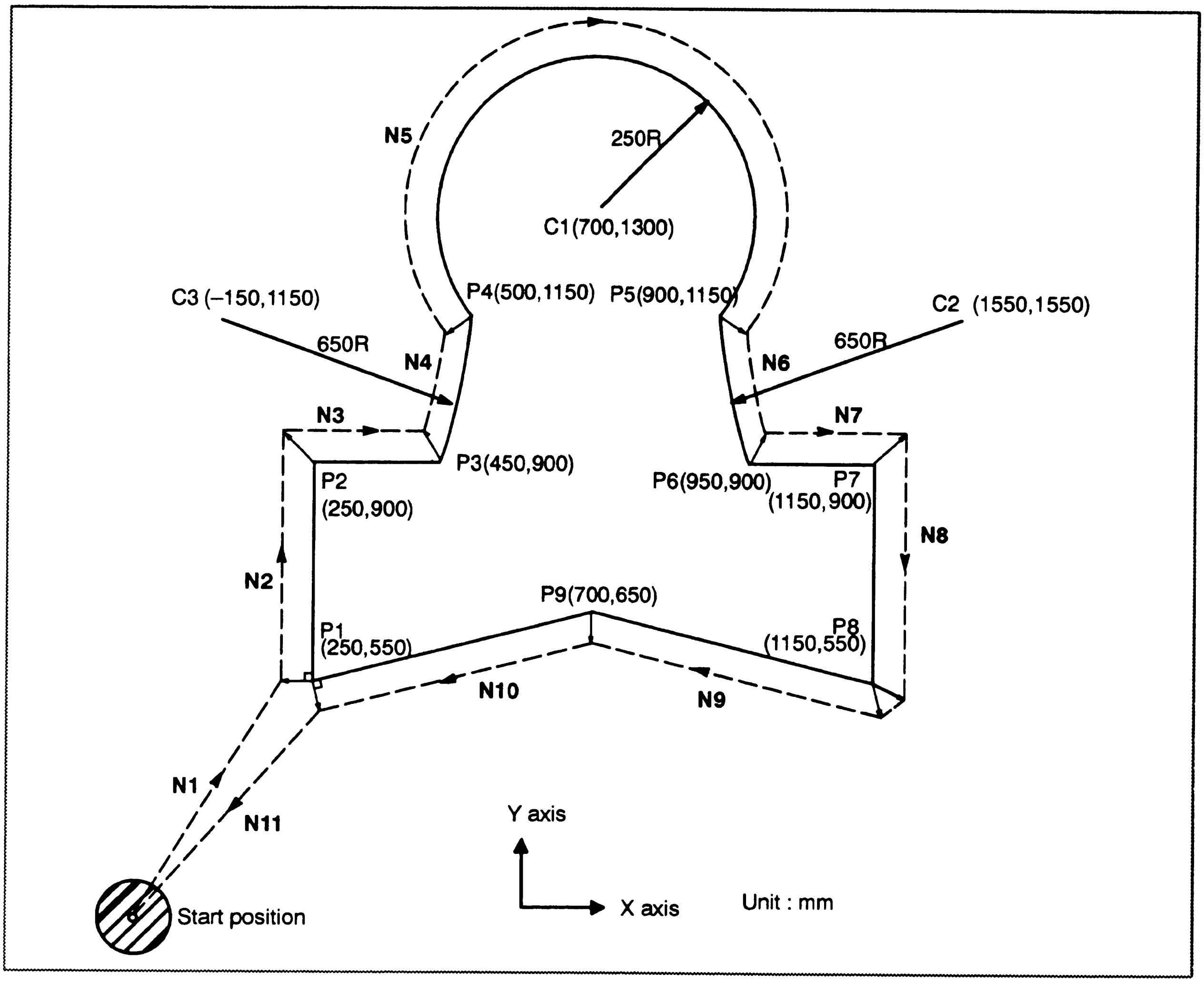
- **Specifying a cutter compensation value**

Specify a cutter compensation value with a number assigned to it. The number consists of 1 to 2 digits after address D (D code). The D code is valid until another D code is specified.

- **Plane selection and vector**

Offset calculation is carried out in the plane determined by G17, G18 and G19, (G codes for plane selection). This plane is called the offset plane. Compensation is not executed for the coordinate of a position which is not in the specified plane. The programmed values are used as they are.
In simultaneous 3 axes control, the tool path projected on the offset plane is compensated.
The offset plane is changed during the offset cancel mode. If it is performed during the offset mode, an alarm (No. 37) is displayed and the machine is stopped.

Examples



G92 X0 Y0 Z0 ;	Specifies absolute coordinates. The tool is positioned at the start position (X0, Y0, Z0).
N1 G90 G17 G00 G41 D07 X250.0 Y550.0 ;	Starts cutter compensation (start–up). The tool is shifted to the left of the programmed path by the distance specified in D07. In other words the tool path is shifted by the radius of the tool (offset mode) because D07 is set to 15 beforehand (the radius of the tool is 15 mm).
N2 G01 Y900.0 F150 ;	Specifies machining from P1 to P2.
N3 X450.0 ;	Specifies machining from P2 to P3.
N4 G03 X500.0 Y1150.0 R650.0 :	Specifies machining from P3 to P4.
N5 G02 X900.0 R–250.0 ;	Specifies machining from P4 to P5.
N6 G03 X950.0 Y900.0 R650.0 ;	Specifies machining from P5 to P6.
N7 G01 X1150.0 ;	Specifies machining from P6 to P7.
N8 Y550.0 ;	Specifies machining from P7 to P8.
N9 X700.0 Y650.0 ;	Specifies machining from P8 to P9.
N10 X250.0 Y550.0 ;	Specifies machining from P9 to P1.
N11 G00 G40 X0 Y0 ;	Cancels the offset mode. The tool is returned to the start position (X0, Y0, Z0).

14.3 DETAILS OF CUTTER COMPENSATION C

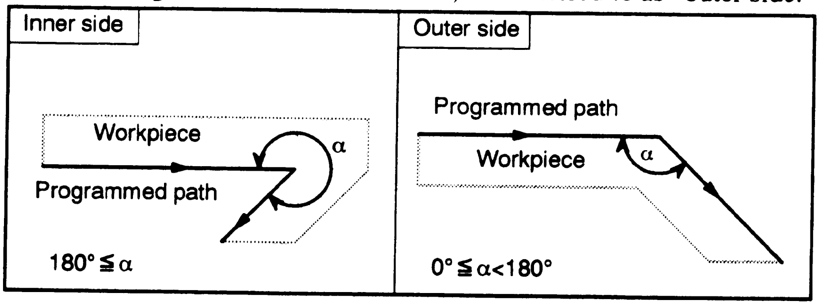
This section provides a detailed explanation of the movement of the tool for cutter compensation C outlined in Section 14.2.
This section consists of the following subsections:

14.3.1 General
14.3.2 Tool Movement in Start–up
14.3.3 Tool Movement in Offset Mode
14.3.4 Tool Movement in Offset Mode Cancel
14.3.5 Interference Check
14.3.6 Over cutting by Cutter Compensation
14.3.7 Input command from MDI

14.3.1 General

• Inner side and outer side

When an angle of intersection created by tool paths specified with move commands for two blocks is over 180°, it is referred to as “inner side.” When the angle is between 0° and 180°, it is referred to as “outer side.”



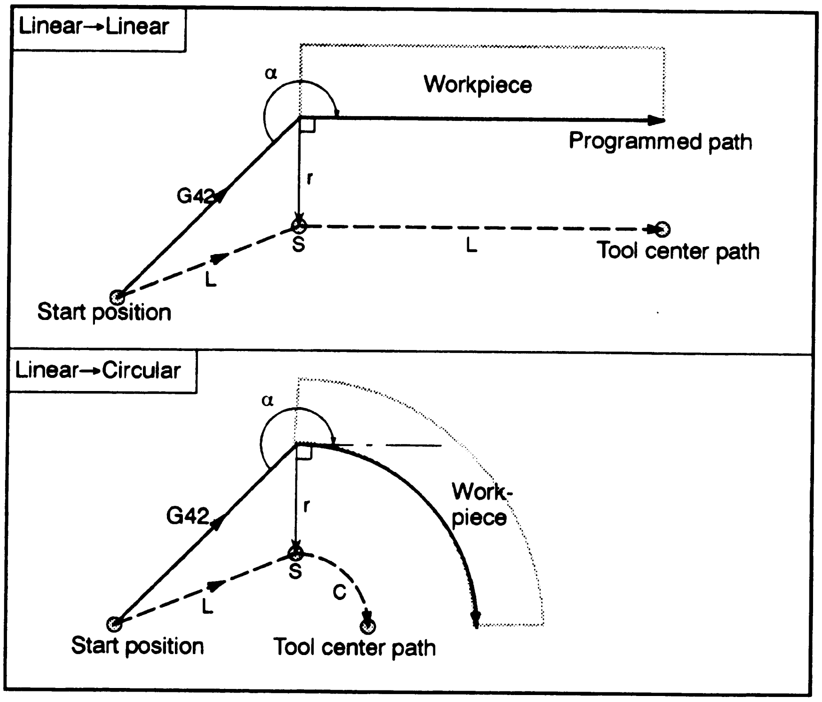
• Meaning of symbols

The following symbols are used in subsequent figures:
– *S* indicates a position at which a single block is executed once.
– *SS* indicates a position at which a single block is executed twice.
– *SSS* indicates a position at which a single block is executed three times.
– *L* indicates that the tool moves along a straight line.
– *C* indicates that the tool moves along an arc.
– *r* indicates the cutter compensation value.
– An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by *r*.
– indicates ○ the center of the tool.

14.3.2 Tool Movement in Start–up

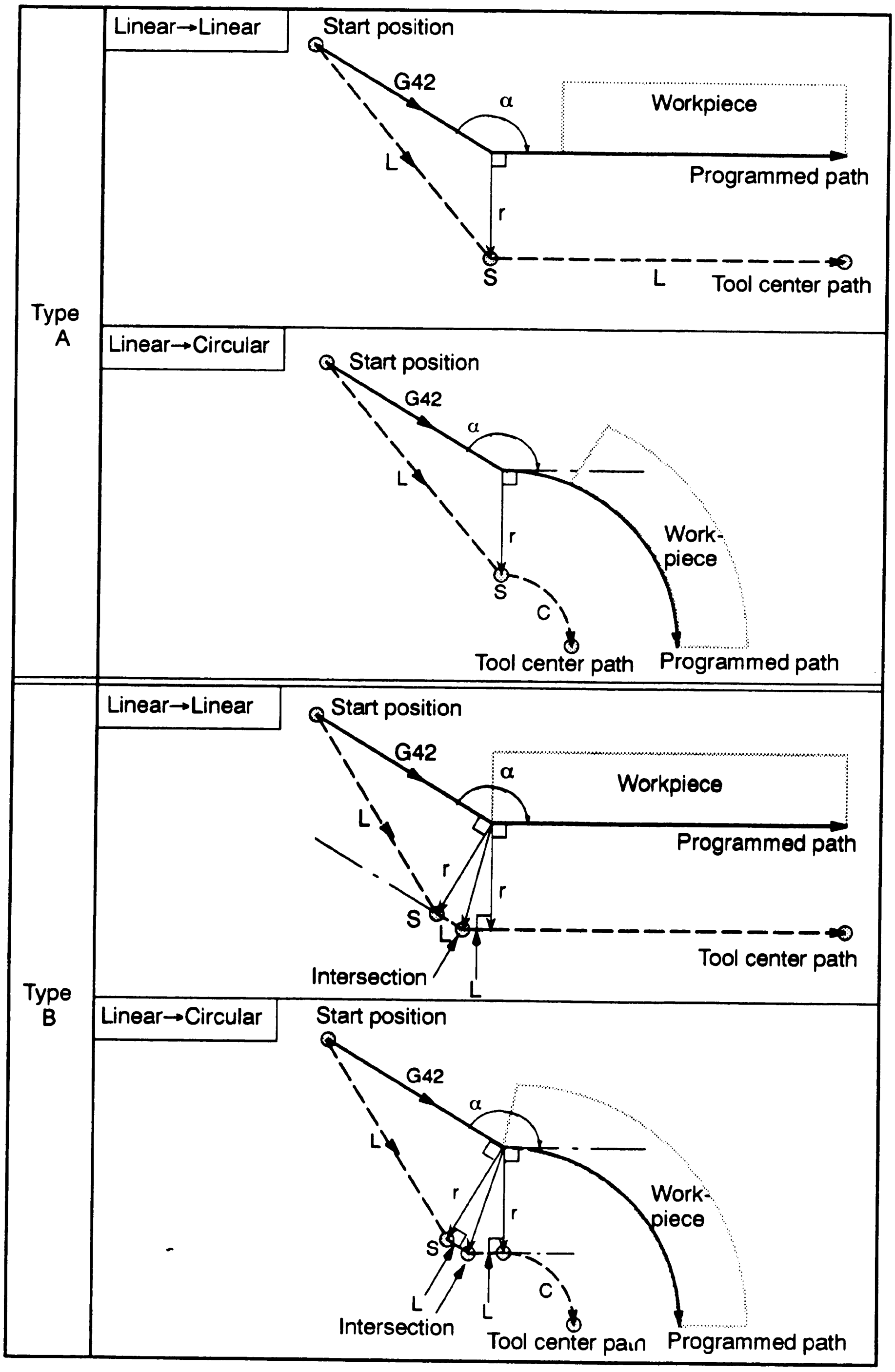
When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start–up):

Explanations



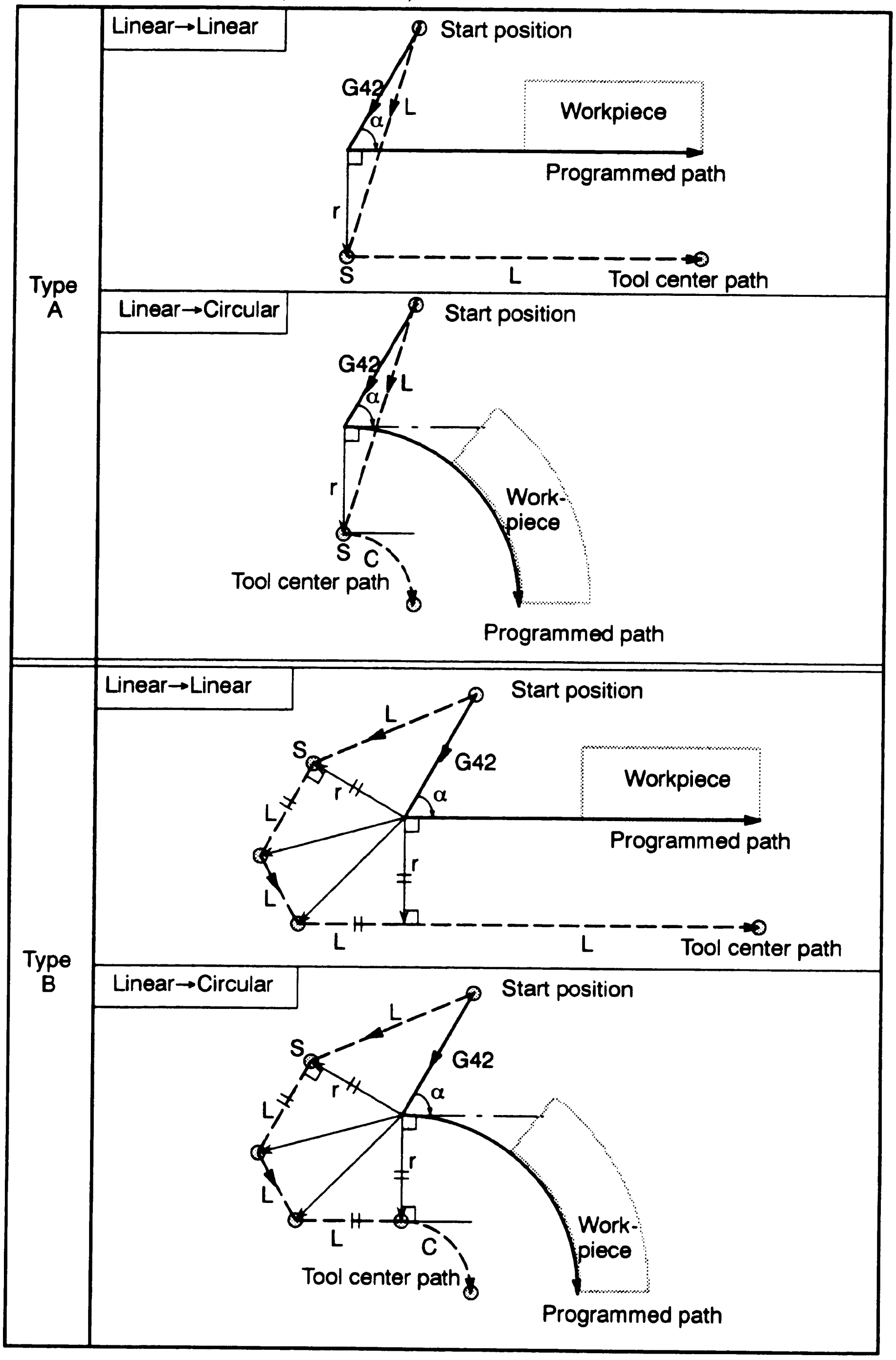
- **Tool movement around the outside of a corner at an obtuse angle (90°≦α<180°)**

Tool path in start–up has two types A and B, and they are selected by parameter SUP (No. 5003#0).

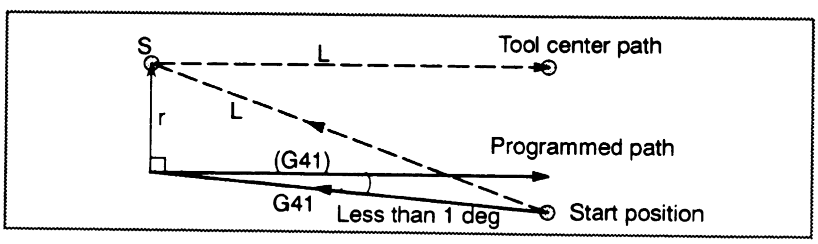


- **Tool movement around the outside of an acute angle (α<90°)**

Tool path in start–up has two types A and B, and they are selected by parameter SUP (No.5003#0).



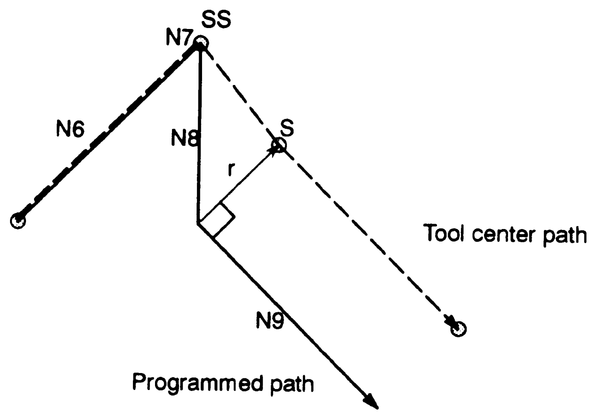
- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



- **A block without tool movement specified at start–up**

If a block without tool movement is commanded at start–up, the offset vector is not created.

```
G91 G40 ... ;
   :
N6 X100.0 Y100.0 ;
N7 G41 X0 ;
N8 Y–100.0 ;
N9 Y–100.0 X100.0 ;
```



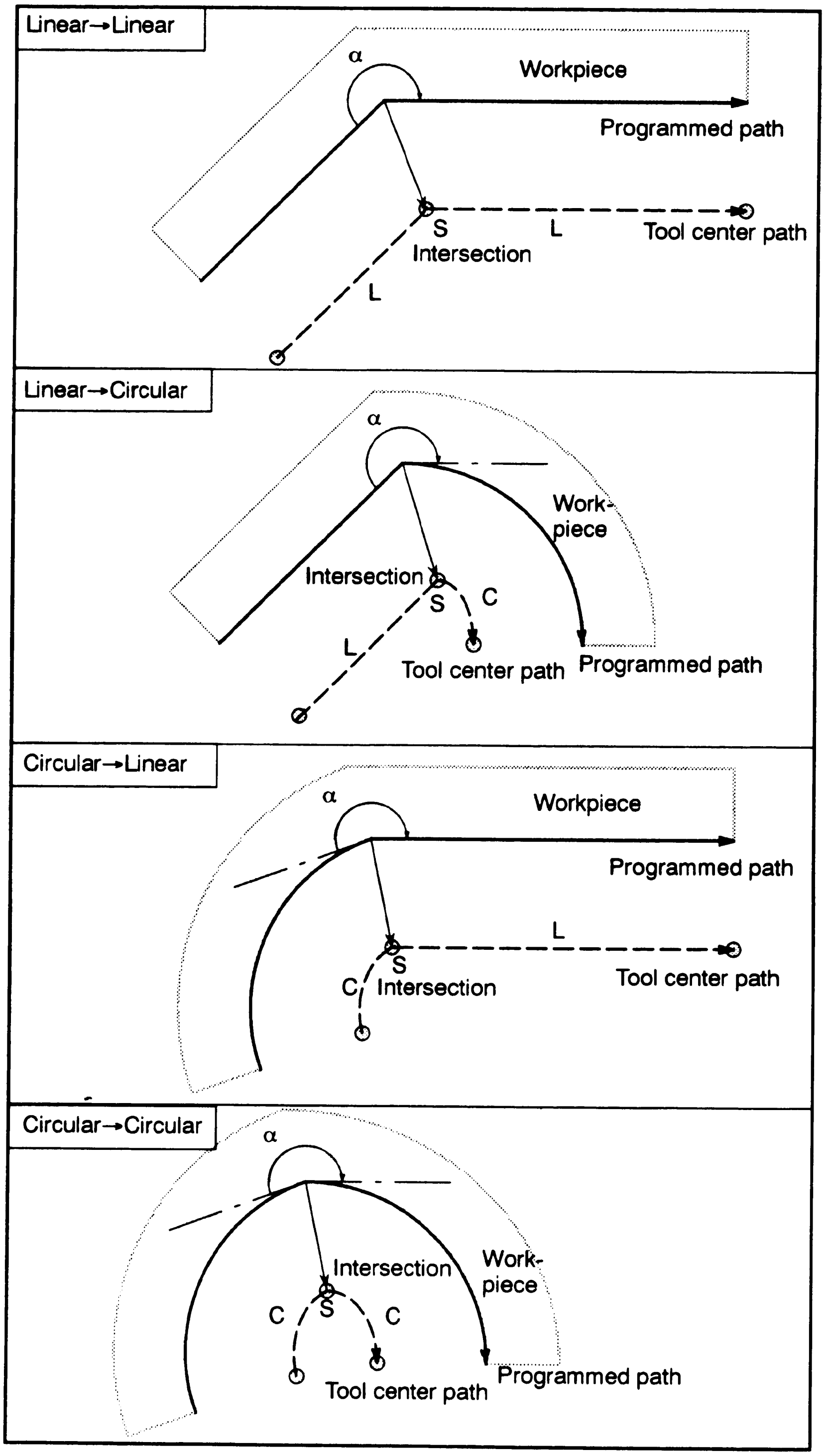
> **Note**
> For the definition of blocks that do not move the tool, see Subsection 14.3.3.

14.3.3 Tool Movement in Offset Mode

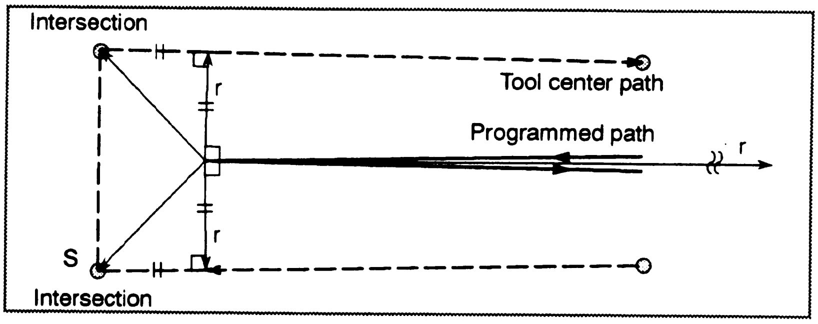
In the offset mode, the tool moves as illustrated below:

Explanations

- **Tool movement around the inside of a corner (180°≦α)**

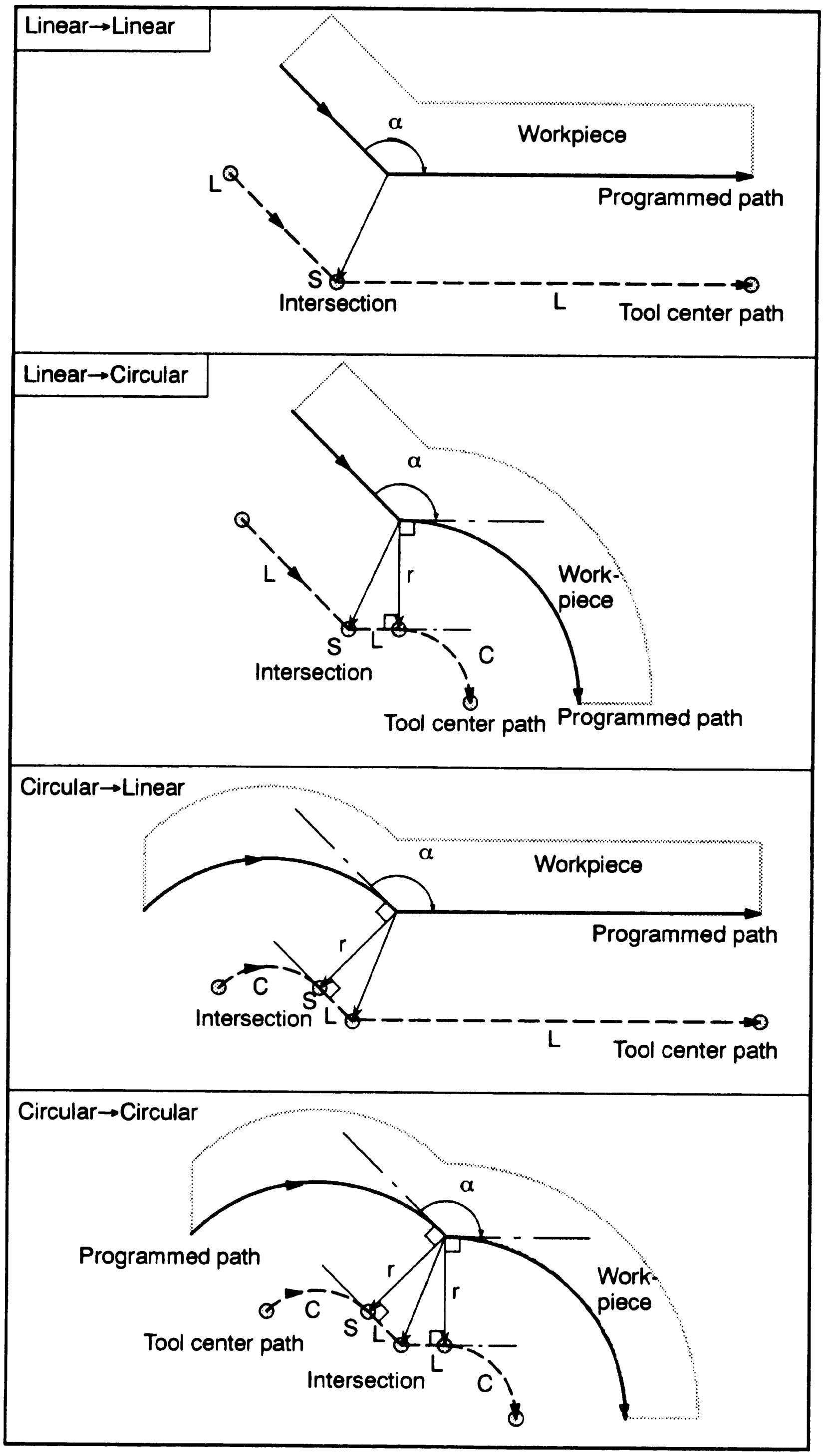


- **Tool movement around the inside (α<1°) with an abnormally long vector, linear → linear**

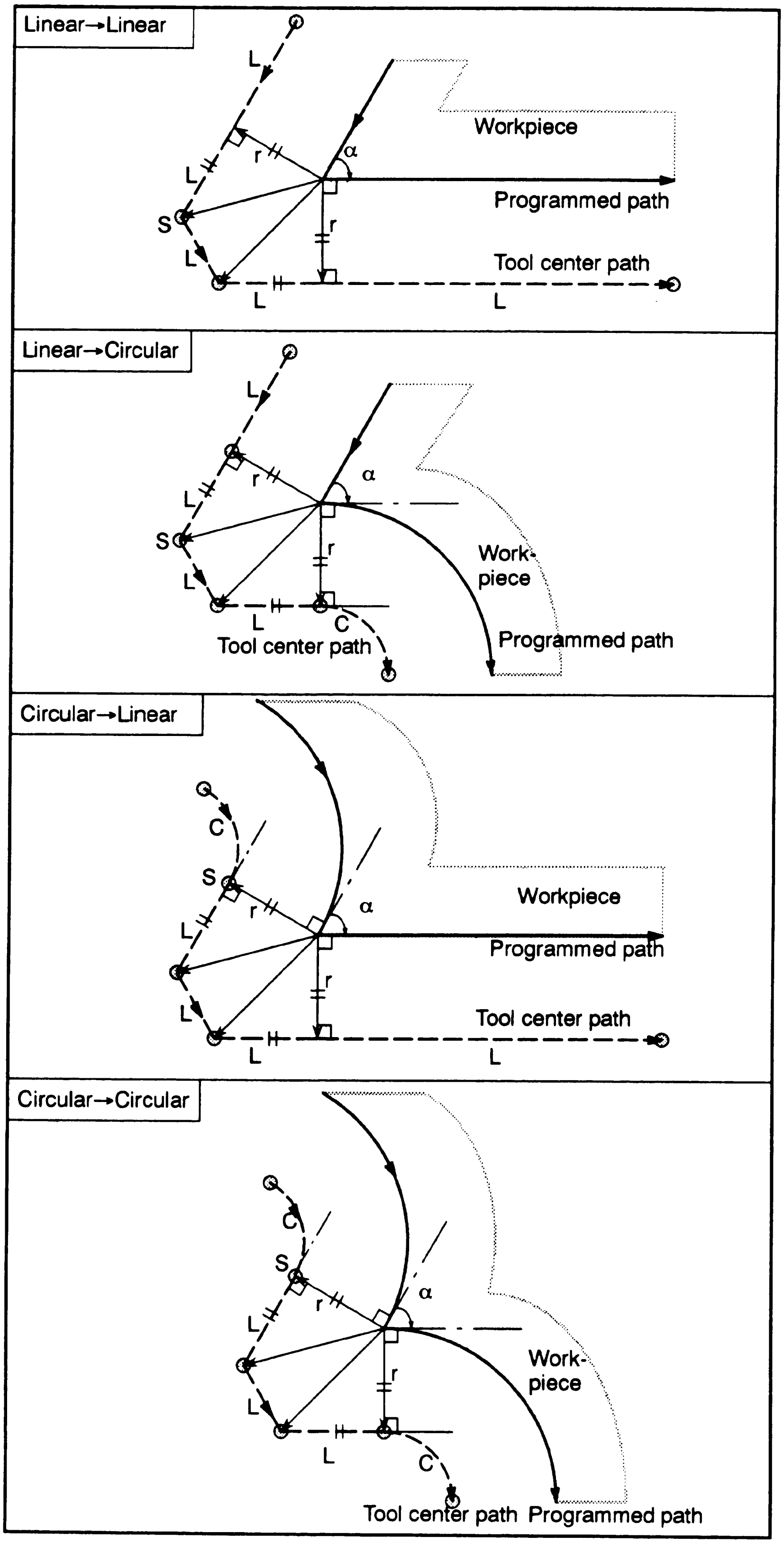


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

- **Tool movement around the outside corner at an obtuse angle (90°≦α<180°)**



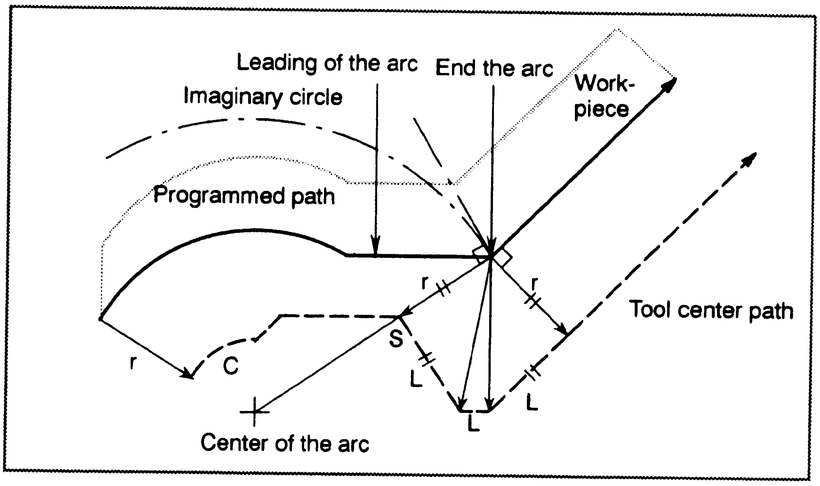
- **Tool movement around the outside corner at an acute angle (α<90°)**



- **When it is exceptional**

End position for the arc is not on the arc

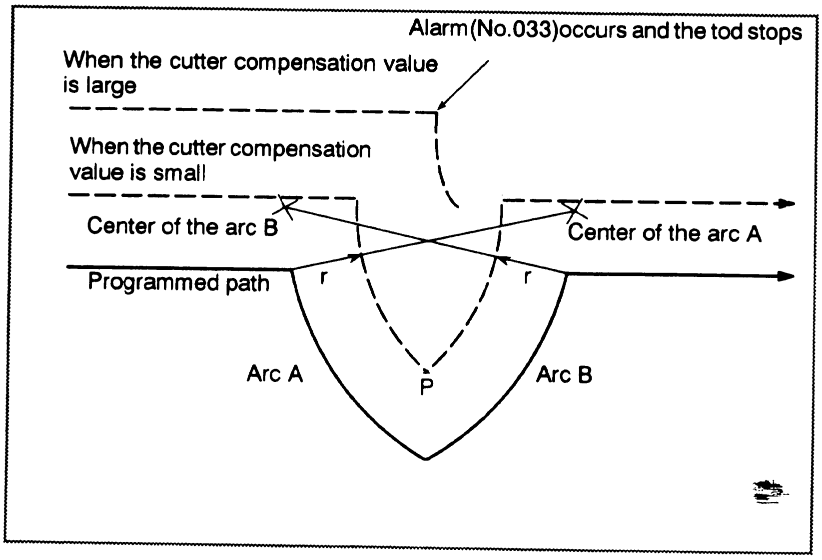
If the end of a line leading to an arc is programmed as the end of the arc by mistake as illustrated below, the system assumes that cutter compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption, the system creates a vector and carries out compensation. The resulting tool center path is different from that created by applying cutter compensation to the programmed path in which the line leading to the arc is considered straight.



The same description applies to tool movement between two circular paths.

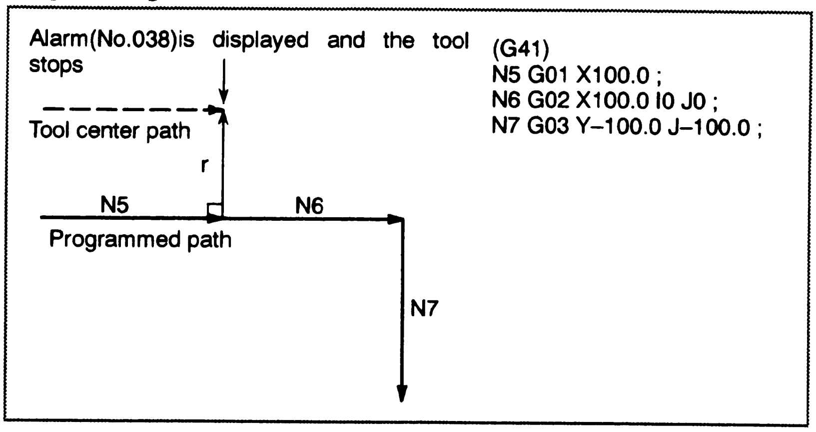
There is no inner intersection

If the cutter compensation value is sufficiently small, the two circular tool center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for cutter compensation. When this is predicted, alarm 33 occurs at the end of the previous block and the tool is stopped. In the example shown below, tool center paths along arcs A and B intersect at P when a sufficiently small value is specified for cutter compensation. If an excessively large value is specified, this intersection does not occur.



The center of the arc is identical with the start position or the end position

If the center of the arc is identical with the start position or end point, alarm (No. 038) is displayed, and the tool will stop at the end position of the preceding block.



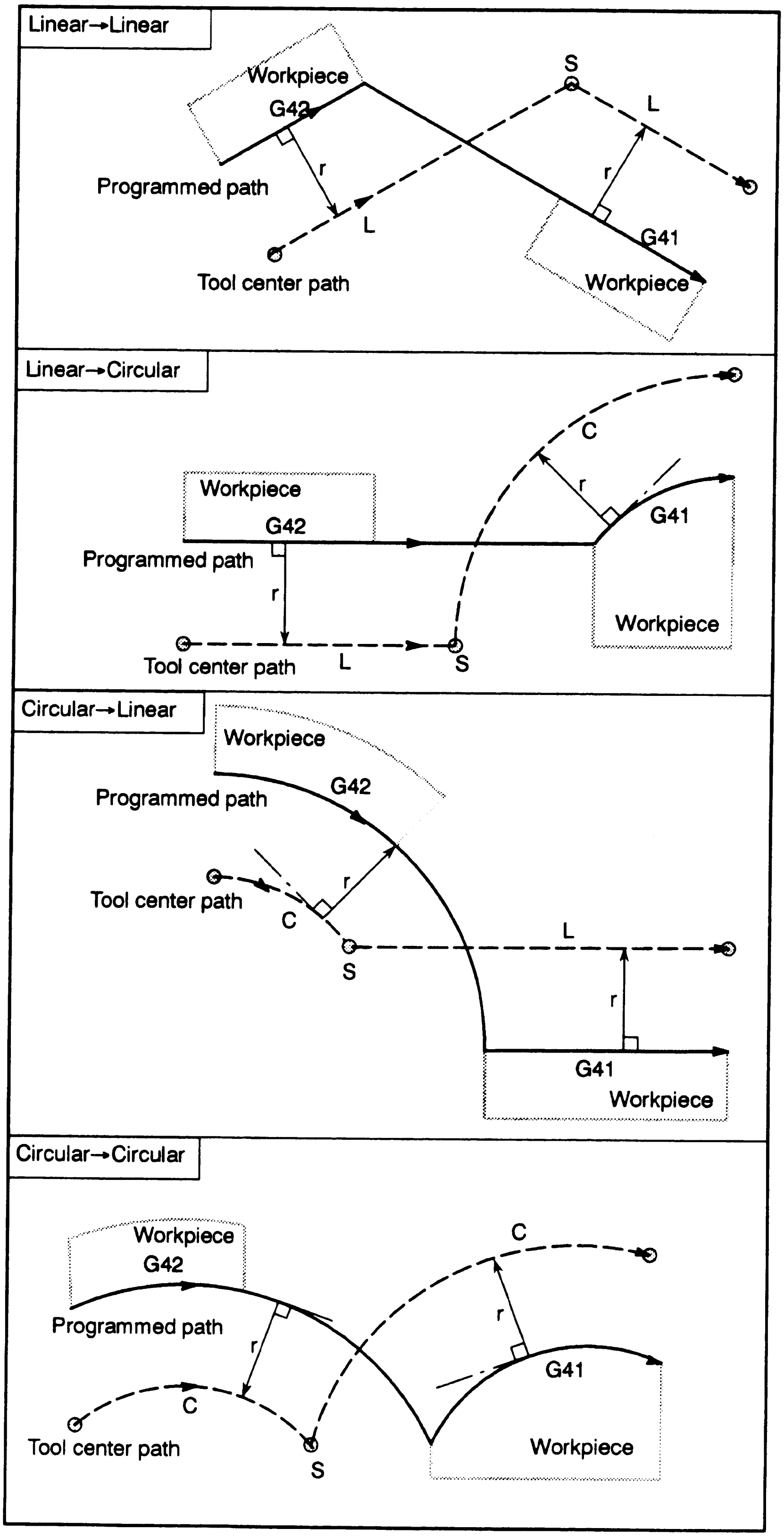
- **Change in the offset direction in the offset mode**

The offset direction is decided by G codes (G41 and G42) for cutter radius and the sign of cutter compensation value as follows.

Sign of offset amount / G code	+	–
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

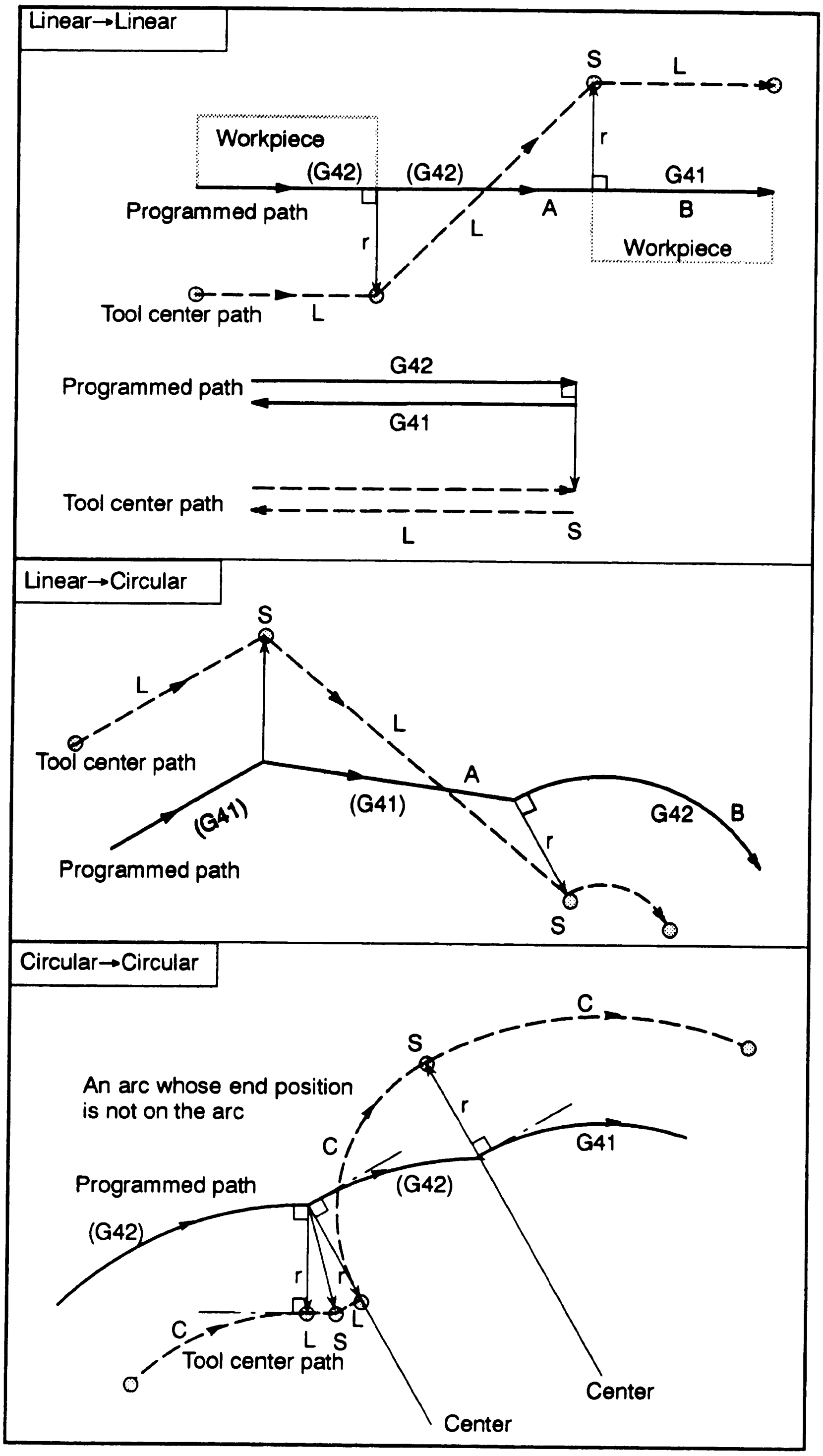
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool center path of that block and the tool center path of a preceding block. However, the change is not available in the start–up block and the block following it.

Tool center path with an intersection



Tool center path without an intersection

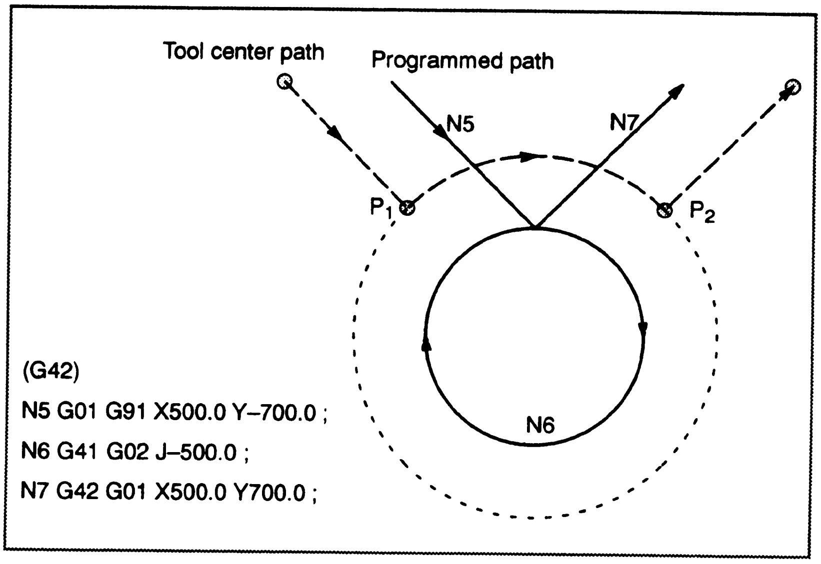
When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.



The length of tool center path larger than the circumference of a circle

Normally there is almost no possibility of generating this situation. However, when G41 and G42 are changed, or when a G40 was commanded with address I, J, and K this situation can occur.

In this case of the figure, the cutter compensation is not performed with more than one circle circumference: an arc is formed from P_1 to P_2 as shown. Depending on the circumstances, an alarm may be displayed due to the "Interference Check" described later. To execute a circle with more than one circumference, the circle must be specified in segments.

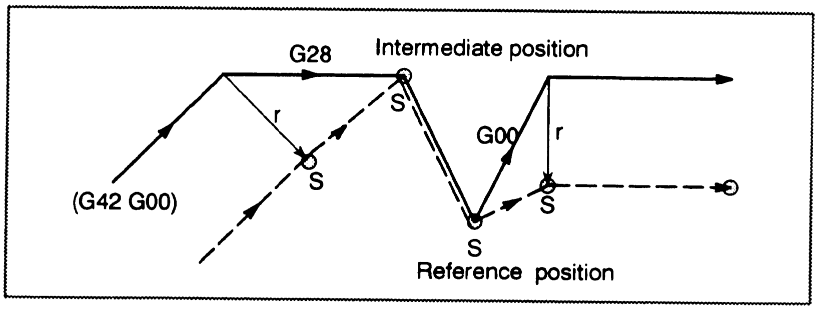


- **Temporary cutter compensation cancel**

If the following command is specified in the offset mode, the offset mode is temporarily canceled then automatically restored. The offset mode can be canceled and started as described in Subsections 15.6.2 and 15.6.4.

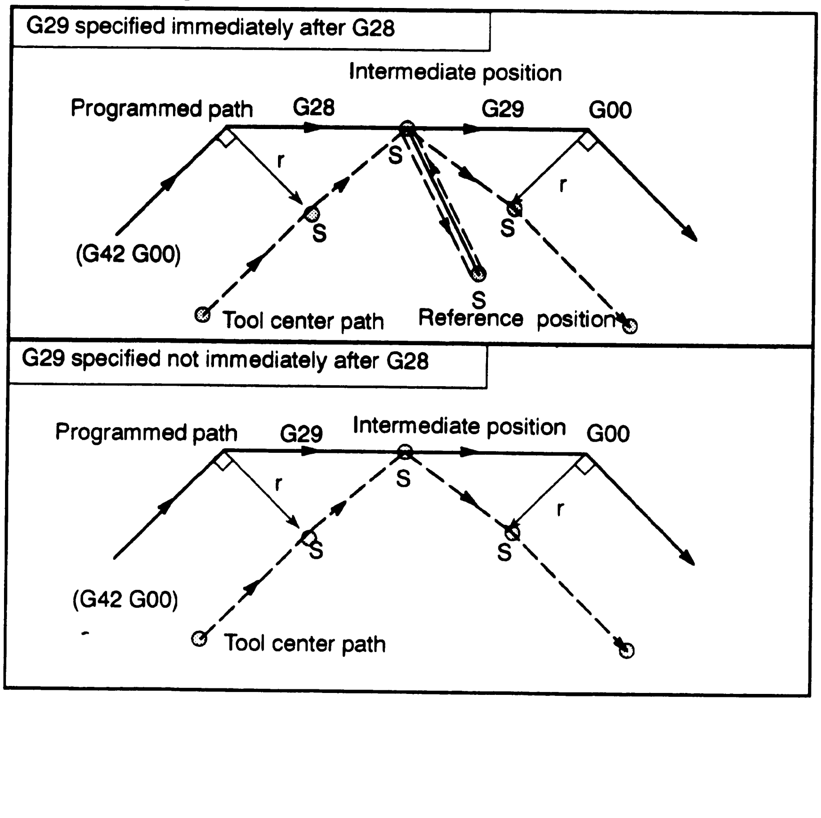
Specifying G28 (automatic return to the reference position) in the offset mode

If G28 is specified in the offset mode, the offset mode is canceled at an intermediate position. If the vector still remains after the tool is returned to the reference position, the components of the vector are reset to zero with respect to each axis along which reference position return has been made.



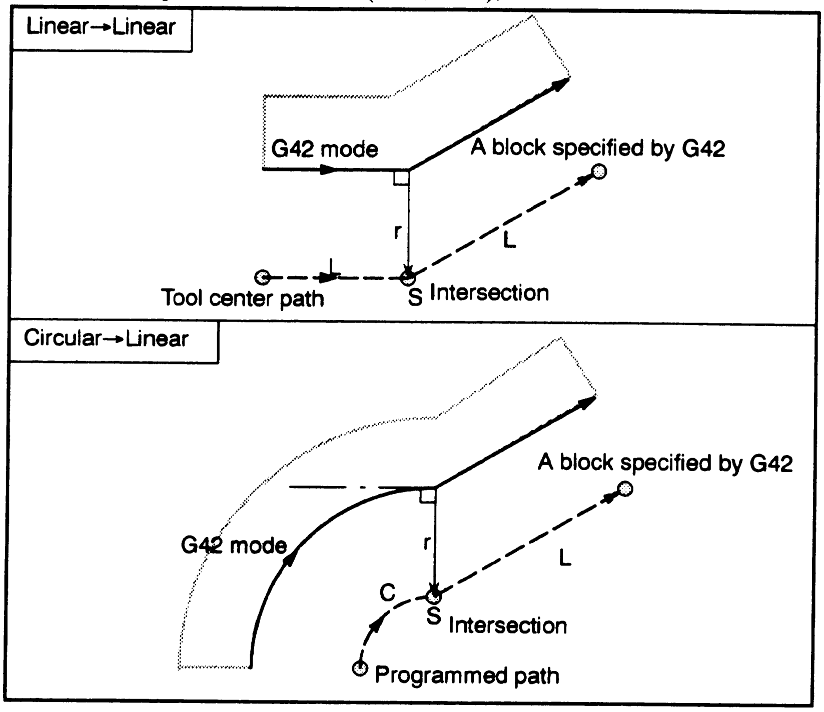
Specifying G29 (automatic return from the reference position) in the offset mode

If G29 is commanded in the offset mode, the offset will be cancelled at the intermediate point, and the offset mode will be restored automatically from the subsequent block.



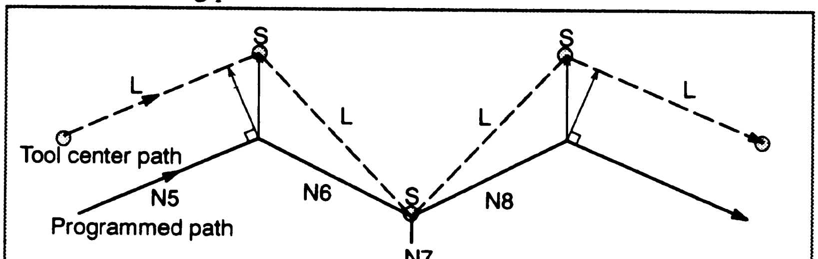
- **Cutter compensation G code in the offset mode**

The offset vector can be set to form a right angle to the moving direction in the previous block, irrespective of machining inner or outer side, by commanding the cutter compensation G code (G41, G42) in the offset mode, independently. If this code is specified in a circular command, correct circular motion will not be obtained.
When the direction of offset is expected to be changed by the command of cutter compensation G code (G41, G42), refer to Subsec.14.3.3.



- **Command cancelling the offset vector temporarily**

During offset mode, if G92 (absolute zero point programming) is commanded,the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.
In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled. Also when restored to offset mode, the tool moves directly to the intersecting point.




```
(G41)
N5 G91 G01 X300.0 Y700.0 ;
N6 X–300.0 Y600.0 ;
N7 G92 X100.0 Y200.0 ;
N8 G90 G01 X400.0 Y800.0 ;
```

- **A block without tool movement**

The following blocks have no tool movement. In these blocks, the tool will not move even if cutter compensation is effected.

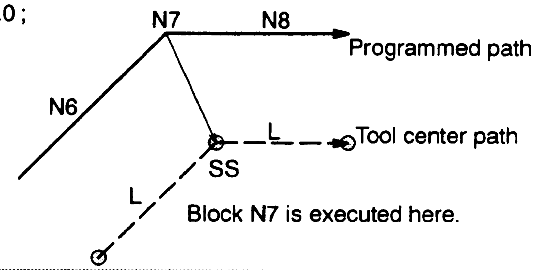
```
M05 ; . . . . . . . . .  M code output
S21 ; . . . . . . . . .  S code output
G04 X10.0 ; . . .  Dwell
G10 L11 P01 R10.0 ; Cutter compensation value setting
(G17) Z200.0 ; .  Move command not included in the
                  offset plane.
G90 ; . . . . . . . . .  G code only
G91 X0 ;Move distance is zero.
```

Commands (1) to (6) are of no movement.

A block without tool movement specified in offset mode

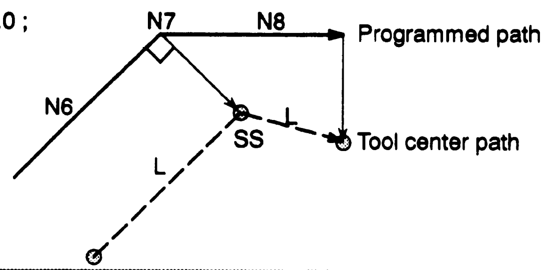
When a single block without tool movement is commanded in the offset mode, the vector and tool center path are the same as those when the block is not commanded. This block is executed at the single block stop point.

N6 G91 X100.0 Y100.0 ;
N7 G04 X100.0 ;
N8 X100.0 ;



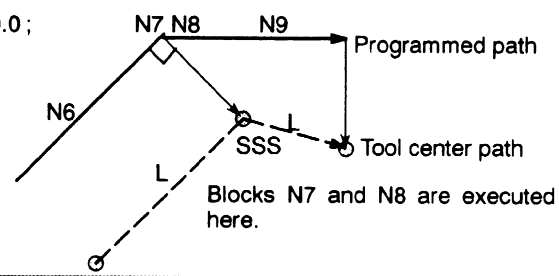
However, when the move distance is zero, even if the block is commanded singly, tool motion becomes the same as that when more than one block of without tool movement are commanded, which will be described subsequently.

N6 G91 X100.0 Y100.0 ;
N7 X0 ;
N8 X100.0 ;



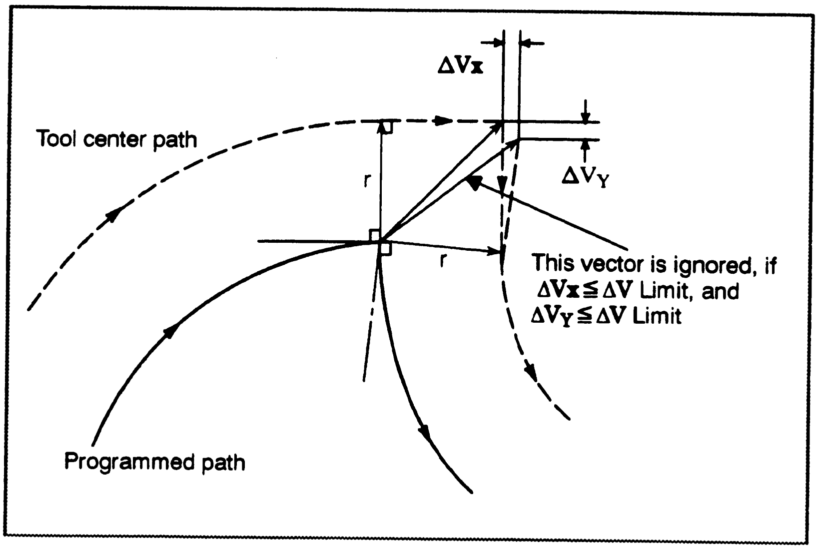
Two blocks without tool movement should not be commanded consecutively. If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier block, so overcutting may result.

N6 G91 X100.0 Y100.0 ;
N7 S21 ;
N8 G04 X10.0 ;
N9 X100.0 ;

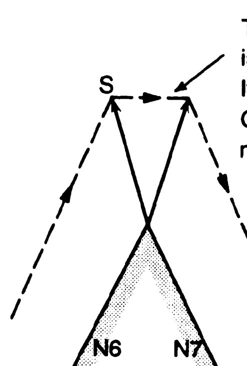


• Corner movement

When two or more vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.
If these vectors almost coincide with each other, the corner movement isn't performed and the latter vector is ignored.

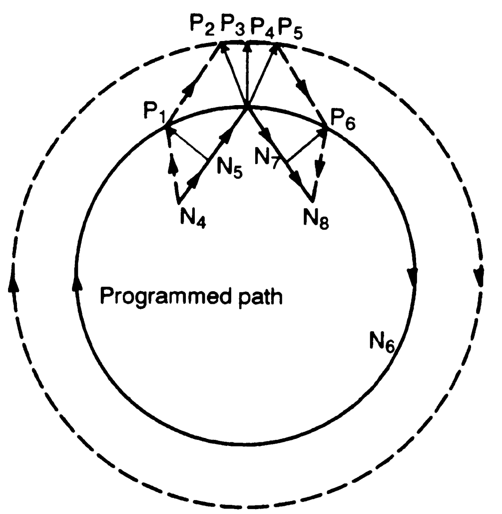


If $\Delta Vx \leqq \Delta V$limit and $\Delta Vy \leqq \Delta V$limit, the latter vector is ignored. The ΔVlimit is set in advance by parameter (No. 5010).
If these vectors do not coincide, a move is generated to turn around the corner. This move belongs to the latter block.



However, if the path of the next block is semicircular or more, the above function is not performed.
The reason for this is as follows:

```
N4 G41 G91 G01 X150.0
Y200.'0 ;
N5 X150.0 Y200.0 ;
N6 G02 J–600.0 ;
N7 G01 X150.0 Y–200.0 ;
N8 G40 X150.0 Y–200.0 ;
```



If the vector is not ignored, the tool path is as follows:
$P_1 \rightarrow P_2 \rightarrow P_3 \rightarrow$ (Circle) $\rightarrow P_4 \rightarrow P_5 \rightarrow P_6$
But if the distance between P2 and P3 is negligible, the point P3 is ignored. Therefore, the tool path is as follows:
$P_2 \rightarrow P_4$
Namely, circle cutting by the block N6 is ignored.

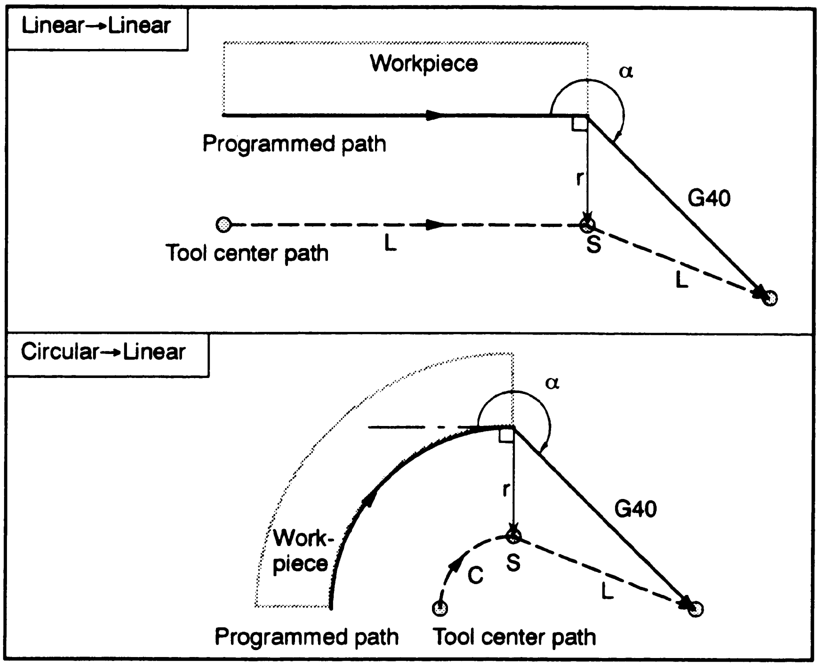
• Interruption of manual operation

For manual operation during the cutter compensation, refer to Section III–3.5, “ Manual Absolute ON and OFF.“

14.3.4 Tool Movement in Offset Mode Cancel

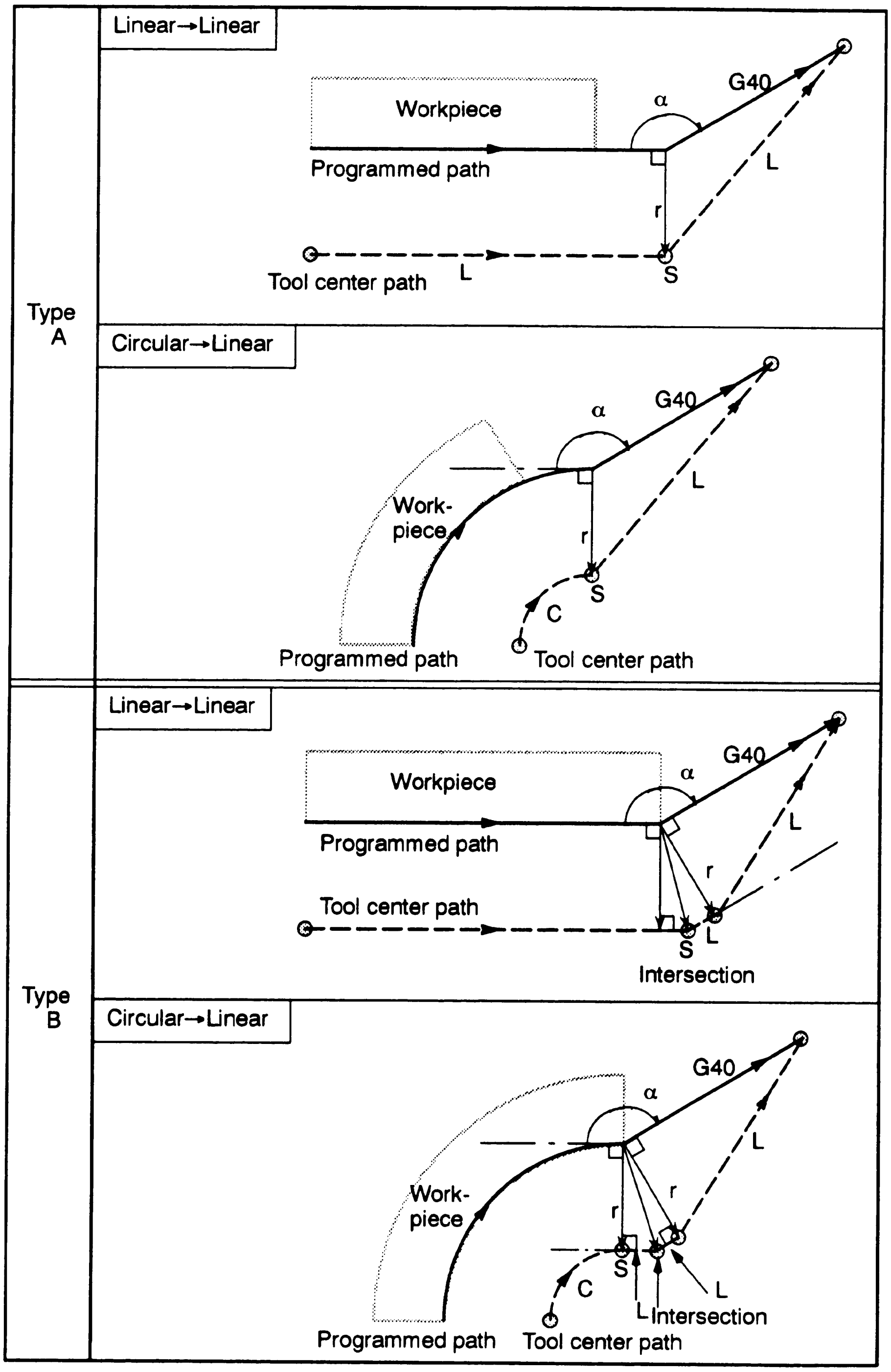
Explanations

- **Tool movement around an inside corner (180° ≦ α)**



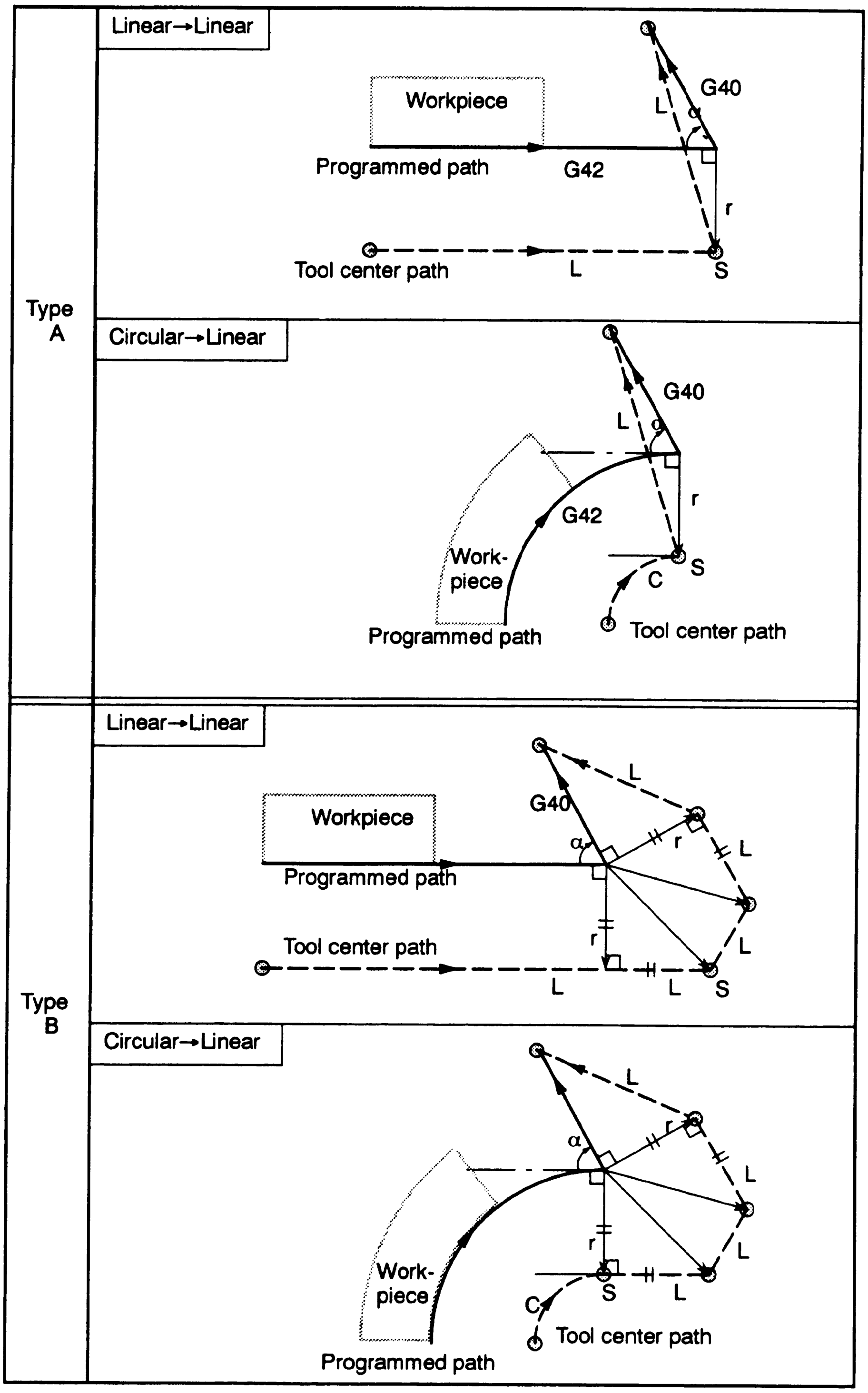
- **Tool movement around an outside corner at an obtuse angle (90° ≦ α < 180°)**

Tool path has two types, A and B; and they are selected by parameter SUP (No. 5003#0).

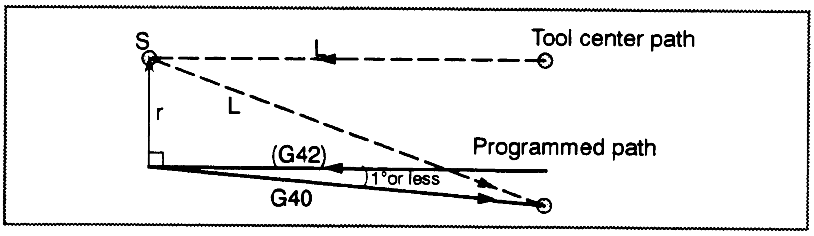


- **Tool movement around an outside corner at an acute angle (α<90°)**

Tool path has two types, A and B : and they are selected by parameter SUP (No. 5003#0)

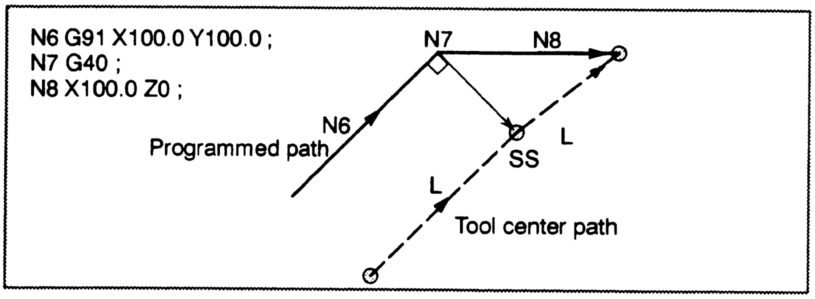


- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



- **A block without tool movement specified together with offset cancel**

When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in the earlier block, the vector is cancelled in the next move command.



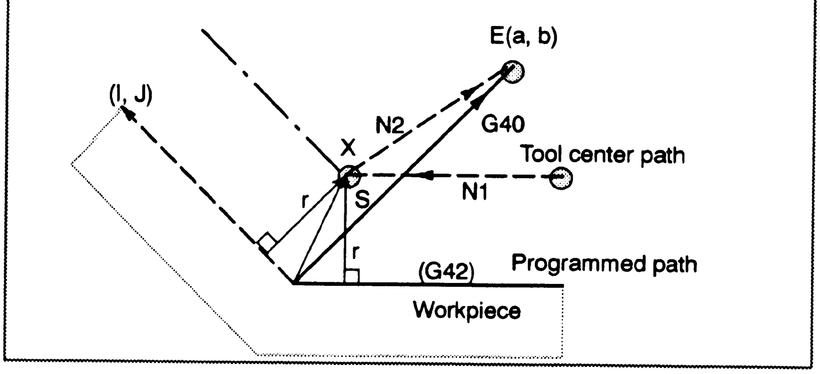
- **Block containing G40 and I_J_K_**

The previous block contains G41 or G42

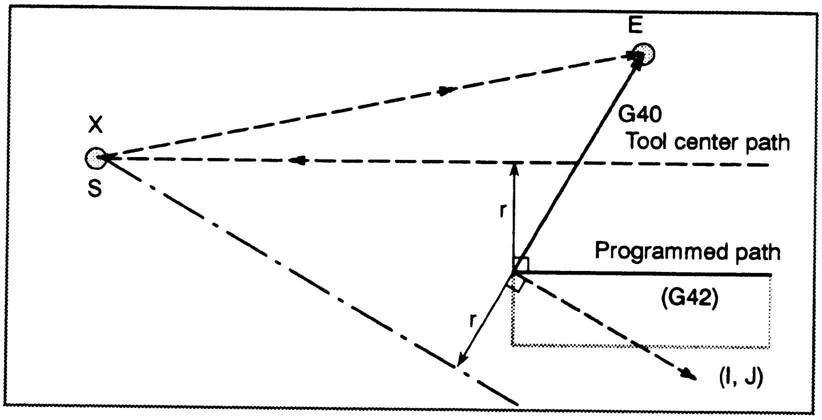
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.

N1 (G42 mode) ;
N2 G40 Xa Yb I_ J_ ;

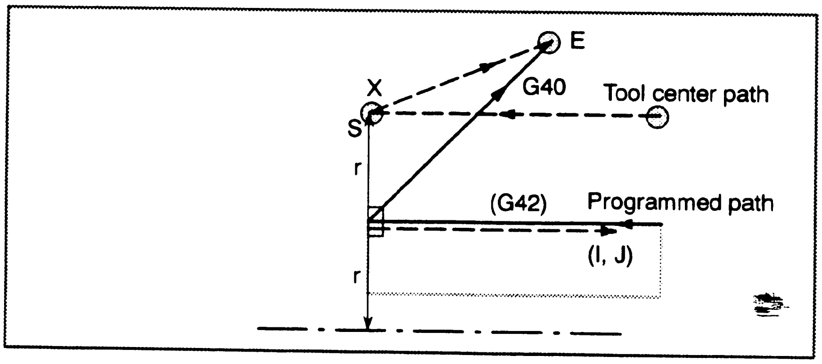
In the N1 block, the tool center moves towards X.
In the N2 block, the tool moves towards E.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified

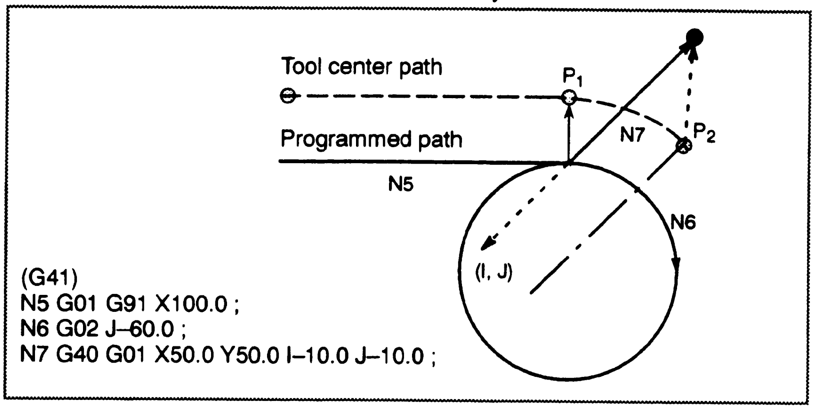


When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



The length of the tool center path larger than the circumference of a circle

In the example shown below, the tool does not trace the circle more than once. It moves along the arc from P1 to P2. The interference check function described in Subsection 14.3.5 may raise an alarm.



To make the tool trace a circle more than once, program two or more arcs.

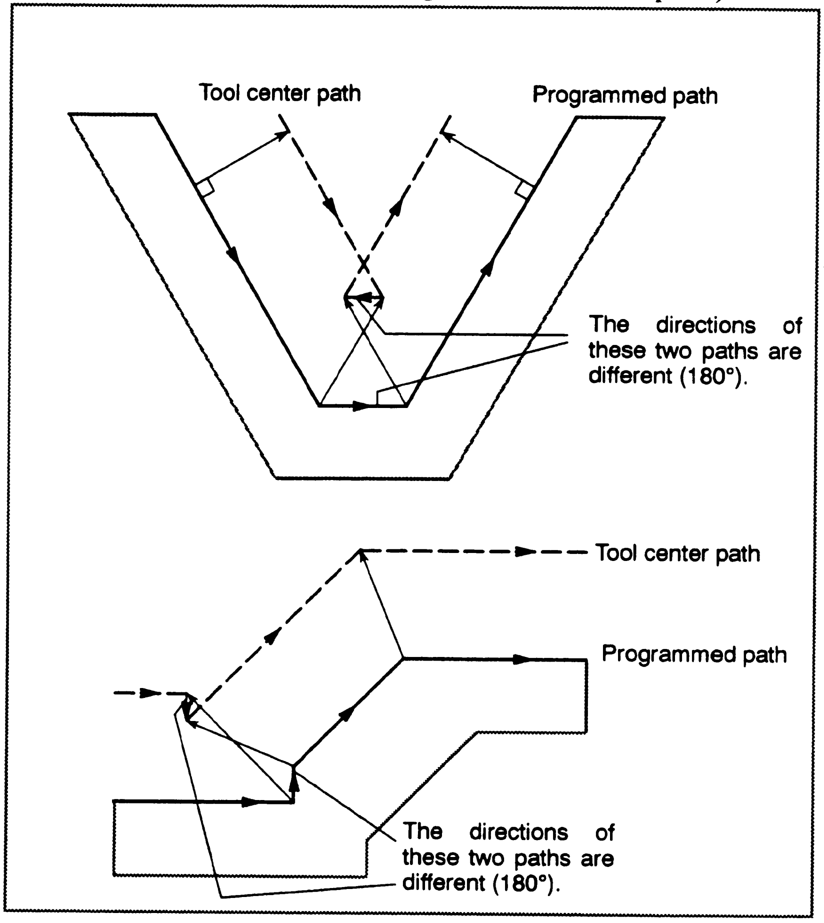
14.3.5 Interference Check

Tool overcutting is called interference. The interference check function checks for tool overcutting in advance. However, all interference cannot be checked by this function. The interference check is performed even if overcutting does not occur.

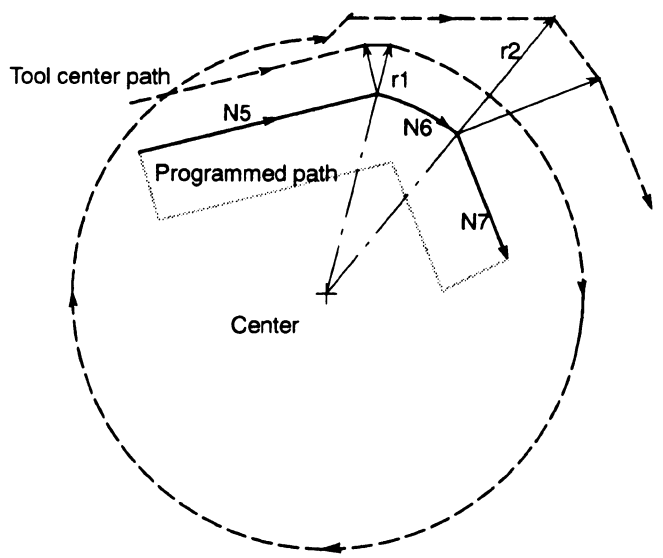
Explanations

- **Criteria for detecting interference**

(1) The direction of the tool path is different from that of the programmed path (from 90 degrees to 270 degrees between these paths).



(2) In addition to the condition (1), the angle between the start point and end point on the tool center path is quite different from that between the start point and end point on the programmed path in circular machining(more than 180 degrees).



(G41)
N5 G01 G91 X800.0 Y200.0 D1 ;
N6 G02 X320.0 Y–160.0 I–200.0 J–800.0 D2 ;
N7 G01 X200.0 Y–500.0 ;
(Tool compensation value corresponding to D1 : r_1 = 200.0)
(Tool compensation value corresponding to D2 : r_2 = 600.0)

In the above example, the arc in block N6 is placed in the one quadrant. But after cutter compensation, the arc is placed in the four quadrants.

- **Correction of interference in advance**

(1) Removal of the vector causing the interference

When cutter compensation is performed for blocks A, B and C and vectors V_1, V_2, V_3 and V_4 between blocks A and B, and V_5, V_6, V_7 and V_8 between B and C are produced, the nearest vectors are checked first. If interference occurs, they are ignored. But if the vectors to be ignored due to interference are the last vectors at the corner, they cannot be ignored.

Check between vectors V_4 and V_5

Interference —— V_4 and V_5 are ignored.

Check between V_3 and V_6

Interference —— V_3 and V_6 are ignored

Check between V_2 and V_7

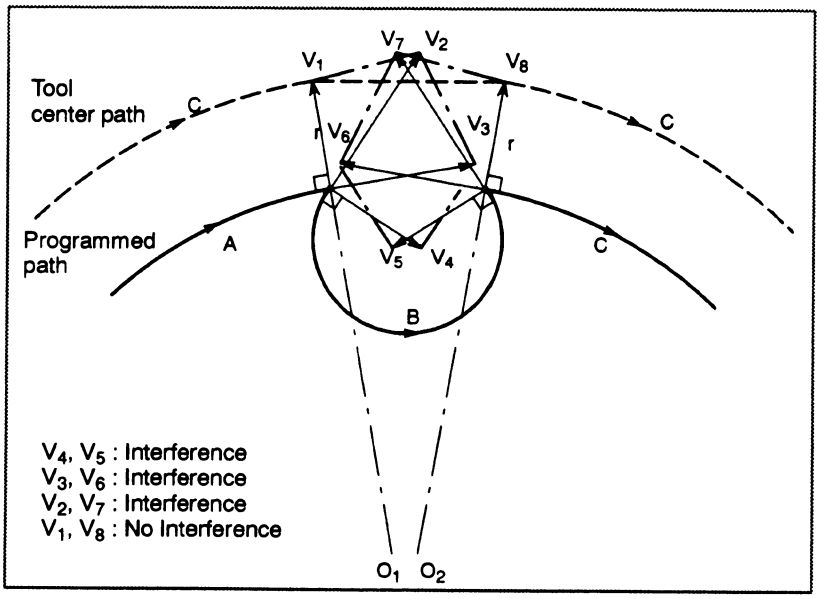
Interference —— V_2 and V_7 are Ignored

Check between V_1 and V_8

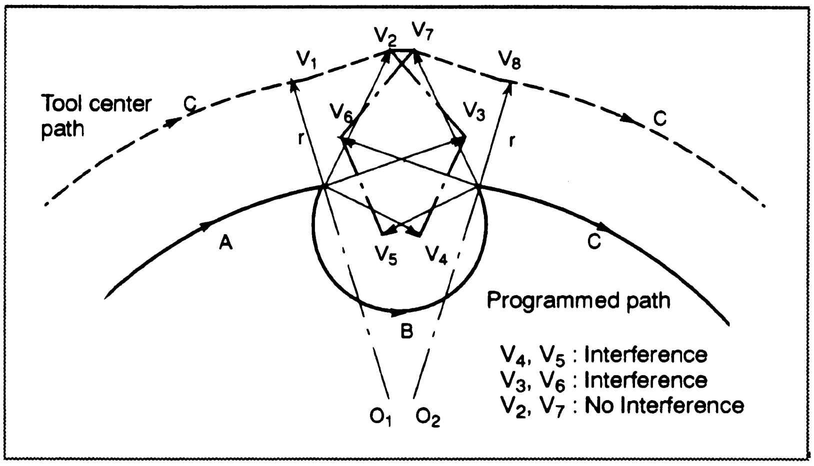
Interference —— V_1 and V_8 are cannot be ignored

If while checking, a vector without interference is detected, subsequent vectors are not checked. If block B is a circular movement, a linear movement is produced if the vectors are interfered.

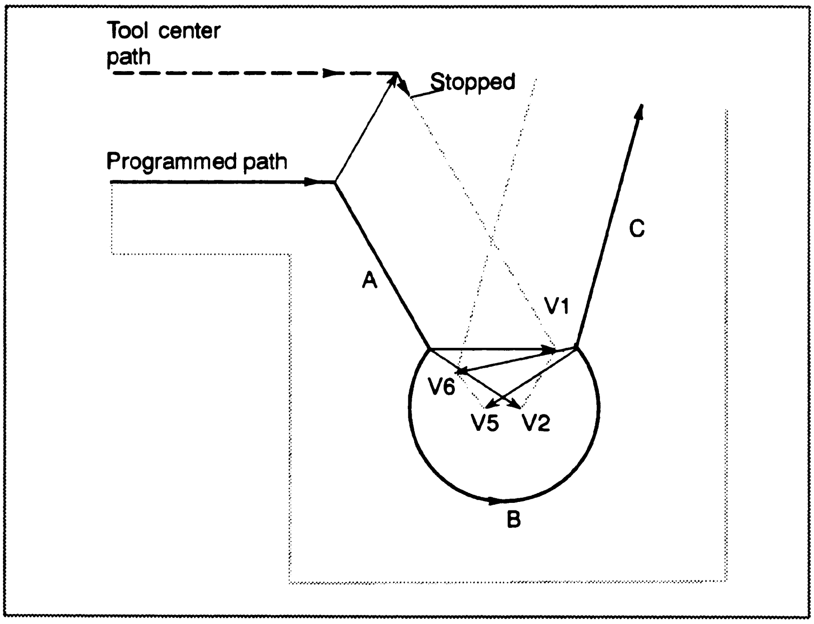
(Example 1) The tool moves linearly from V_1 to V_8



(Example 2) The tool moves linearly from V_1, V_2, V_7, to V_8



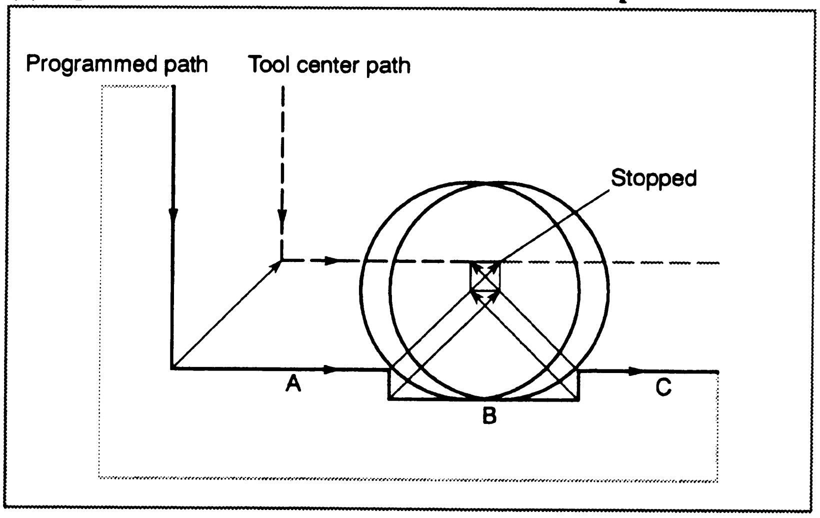
(2) If the interference occurs after correction (1), the tool is stopped with an alarm.
If the interference occurs after correction (1) or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the alarm (No.41) is displayed and the tool is stopped immediately after execution of the preceding block. If the block is executed by the single block operation, the tool is stopped at the end of the block.



After ignoring vectors V_2 and V_5 because of interference, interference also occurs between vectors V_1 and V_6. The alarm is displayed and the tool is stopped.

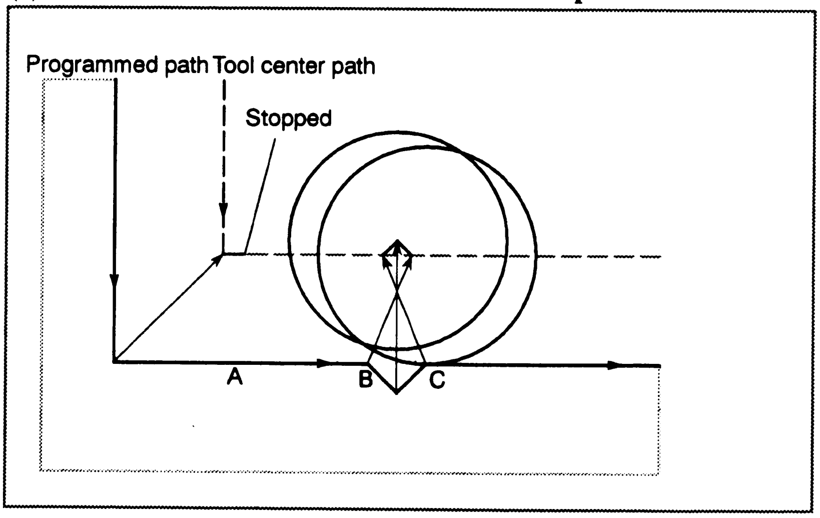
- **When interference is assumed although actual interference does not occur**

(1)Depression which is smaller than the cutter compensation value



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after cutter compensation the tool stops and an alarm is displayed.

(2)Groove which is smaller than the cutter compensation value



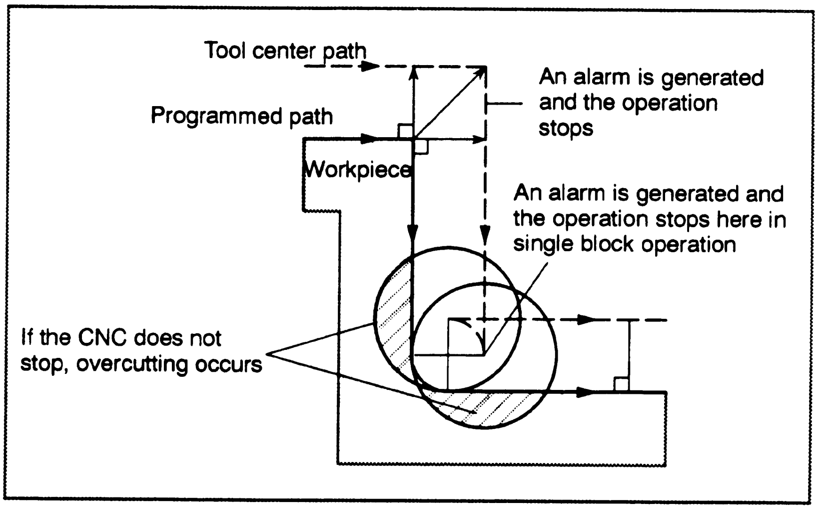
Like (1), interference is assumed with an alarm being displayed and the tool being stopped because the direction is reversed in block B.

14.3.6 Overcutting by Cutter Compensation

Explanations

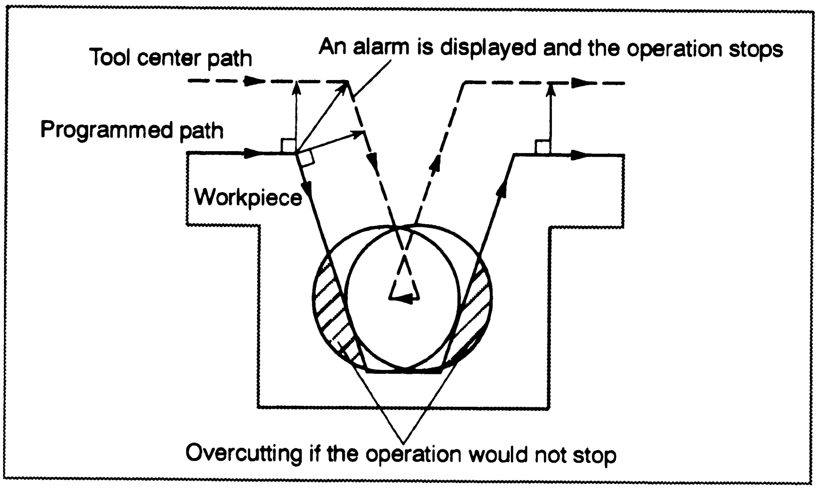
• Machining an inside corner at a radius smaller than the cutter radius

When the radius of a corner is smaller than the cutter radius, because the inner offsetting of the cutter will result in overcuttings, an alarm is displayed and the CNC stops at the start of the block. In single block operation, the overcutting is generated because the tool is stopped after the block execution.



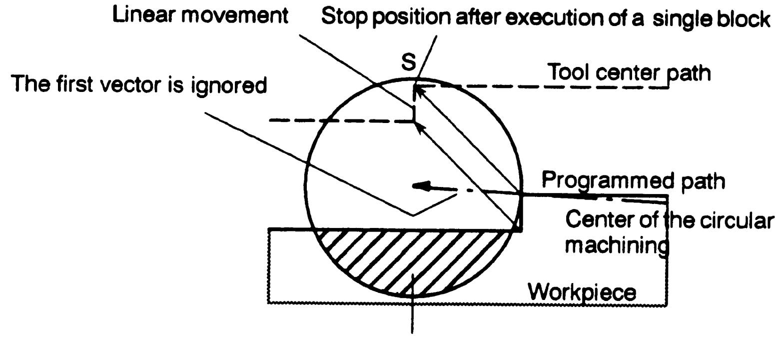
• Machining a groove smaller than the tool radius

Since the cutter compensation forces the path of the center of the tool to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.



• Machining a step smaller than the tool radius

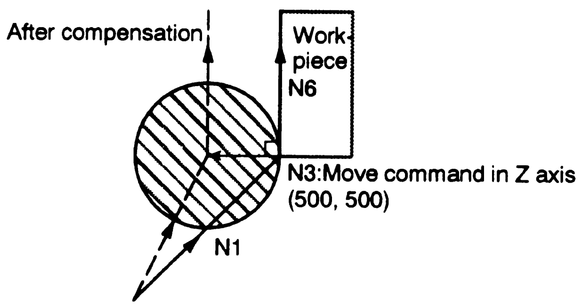
When machining of the step is commanded by circular machining in the case of a program containing a step smaller than the tool radius, the path of the center of tool with the ordinary offset becomes reverse to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The single block operation is stopped at this point. If the machining is not in the single block mode, the cycle operation is continued. If the step is of linear, no alarm will be generated and cut correctly. However uncut part will remain.



• Starting compensation and cutting along the Z–axis

It is usually used such a method that the tool is moved along the Z axis after the cutter compensation is effected at some distance from the workpiece at the start of the machining.
In the case above, if it is desired to divide the motion along the Z axis into rapid traverse and cutting feed, follow the procedure below.

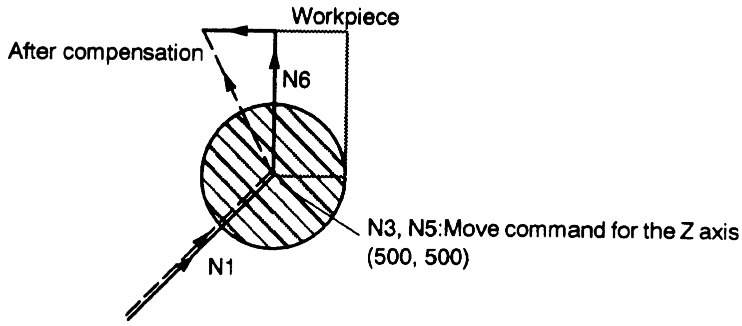
```
N1 G91 G00 G41 X500.0 Y500.0 D1 ;
N3 G01 Z–300.0 F100 ;
N6 Y1000.0 F200 ;
```



In the program example above, when executing block N1, blocks N3 and N6 are also entered into the buffer storage, and by the relationship among them the correct compensation is performed as in the figure above.
Then, if the block N3 (move command in Z axis) is divided as follows:
As there are two move command blocks not included in the selected plane and the block N6 cannot be entered into the buffer storage, the tool center path is calculated by the information of N1 in the figure above. That is, the offset vector is not calculated in start–up and the overcutting may result.

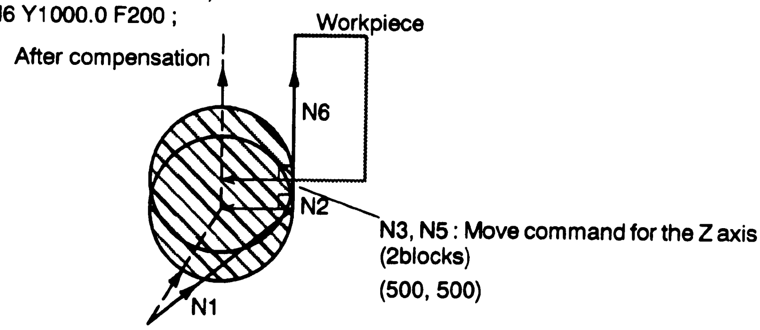
The above example should be modified as follows:

```
N1 G91 G00 G41 X500.0 Y500.0 D1 ;
N3 G01 Z–250.0 ;
N5 G01 Z–50.0 F100 ;
N6 Y1000.0 F200 ;
```



The move command in the same direction as that of the move command after the motion in Z axis should be programmed.

```
N1 G91 G00 G41 X500.0 Y400.0 D1 ;
N2 Y100.0 ;
N3 Z–250.0 ;
N5 G01 Z–50.0 F100 ;
N6 Y1000.0 F200 ;
```



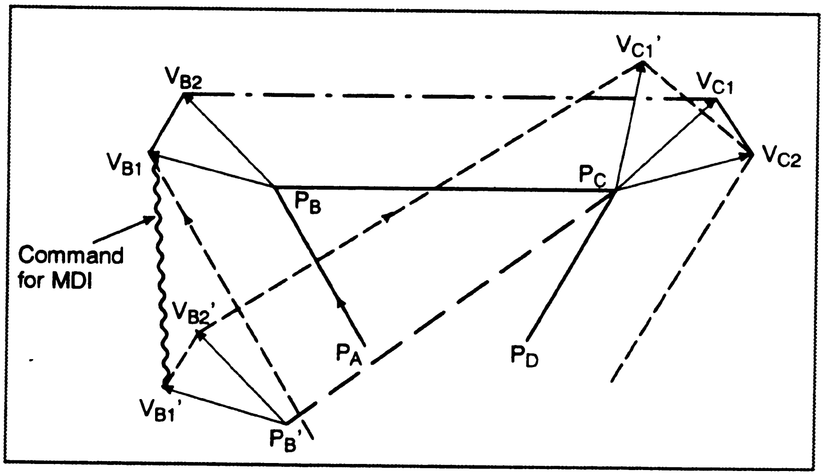
As the block with sequence No. N2 has the move command in the same direction as that of the block with sequence No. N6, the correct compensation is performed.

14.3.7 Input Command from MDI

Cutter compensation C is not performed for commands input from the MDI.

However, when automatic operation by the absolute commands is temporarily stopped by the single block function, MDI operation is performed, then automatic operation starts again, the tool path is as follows :

In this case, the vectors at the start position of the next block are translated and the other vectors are produced by the next two blocks. Therefore, from next block but one, cutter compensation C is accurately performed.



When position P_A, P_B, and P_C are programmed in an absolute command, tool is stopped by the single block function after executing the block from P_A to P_B and the tool is moved by MDI operation. Vectors V_{B1} and V_{B2} are translated to V_{B1}' and V_{B2}' and offset vectors are recalculated for the vectors V_{C1} and V_{C2} between block P_B–P_C and P_C–P_D.

However, since vector V_{B2} is not calculated again, compensation is accurately performed from position P_C.

14.4 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

Tool compensation values can be entered into CNC memory from the CRT/MDI panel (see section III–11.4.1) or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address H or D in a program.
The value is used for tool length compensation, or cutter compensation.

Explanations

- **Valid range of tool compensation values**

Table 14.4 (a) shows the valid input range of tool compensation values.

Table14.4 (a) The valid input range of tool compensation value

Increment system	Tool compensation value	
	Metric input	Inch input
IS–B	±999.999 mm	±99.9999inch

- **Number of tool compensation values and the addresses to be specified**

The memory can hold 32 tool compensation values.
Address D or H is used in the program. The address used depends on which of the following functions is used: tool length compensation(see section 14.1), or cutter compensation C (see section 14.2).
The range of the number that comes after the address (D or H) depens on the number of tool compensation values : 0 to 32, 0 to 64, or 0 to 99.

- **Input of tool compensation value by programing**

Format

```
G10 L11 P_ R_ ;
```

P : Number of tool compensation

R : Tool compensation value in the absolute command(G90) mode
Value to be added to the specified tool compensation value in the incremental command(G91) mode (the sum is also a tool compensation value.)

> **Note**
> To provide compatibility with the format of older CNC programs, the system allows L1 to be specified instead of L11.

14.5 SCALING (G50,G51)

A programmed figure can be magnified or reduced (scaling).
The dimensions specified with X_, Y_, and Z_ can each be scaled up or down with the same or different rates of magnification.
The magnification rate can be specified in the program.
Unless specified in the program, the magnification rate specified in the parameter is applied.

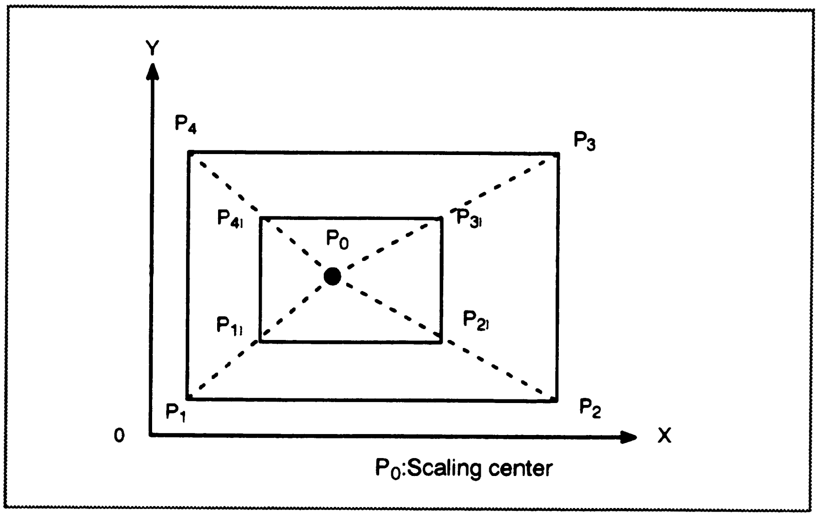


Fig.14.5 (a) Scaling(P_1 P_2 P_3 P_4→$P_1'$$P_2'$$P_3'$$P_4'$)

Format

SCALING UP OR DOWN ALONG ALL AXES AT THE SAME RATE OF MAGNIFICATION	
Format	Meaning of command
G51X_Y_Z_P_ ; Scaling start ⋮ Scaling is effective. (Scaling mode) G50 ; Scaling cancel	X_Y_Z_ : Absolute command for center coordinate value of scaling P_ : Scaling magnification

Scaling up or down along each axes at a different rate of magnification (mirror image)	
Format	Meaning of command
G51_X_Y_Z_I_J_K_;Scaling start ⋮ Scaling is effective (Scaling mode) G50 Scaling cancel	X_Y_Z_ Absolute command for center coordinate value of scaling I_J_K_ Scaling magnification for X axis Y axis and Z axis respectively

Note
Specify G51 in a separate block. After the figure is enlarged or reduced, specify G50 to cancel the scaling mode.

Explanations

- **Scaling up or down along all axes at the same rate of magnification**

Least input increment of scaling magnification is: 0.001 or 0.00001 It is depended on parameter SCR (No. 5400#7) which value is selected. If scaling P is not specified on the block of scaling (G51X_Y_Z_P_ ;), the scaling magnification set to parameter (No. 5411) is applicable. If X,Y,Z are omitted, the tool position where the G51 command was specified serves as the scaling center.

- **Scaling of each axis, programmable mirror image (negative magnification)**

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. First of all, set a parameter XSC (No. 5400#6) which validates each axis scaling (mirror image).
Then, set parameter SCLx (No. 5401#0) to enable scaling along each axis.
Least input increment of scaling magnification of each axis (I, J, K) is 0.001 or 0.00001(set parameter SCR (No. 5400#7)).
Magnification is set to parameter 5421 within the range ±0.00001 to ±9.99999 or ±0.001 to ±999.999
If a negative value is set, mirror image is effected.
If magnification I, J or K is not commanded, a magnification value set to parameter (No. 5421) is effective. However, a value other than 0 must be set to the parameter.

> **Note**
> Decimal point programming can not be used to specify the rate of magnification (I, J, K).

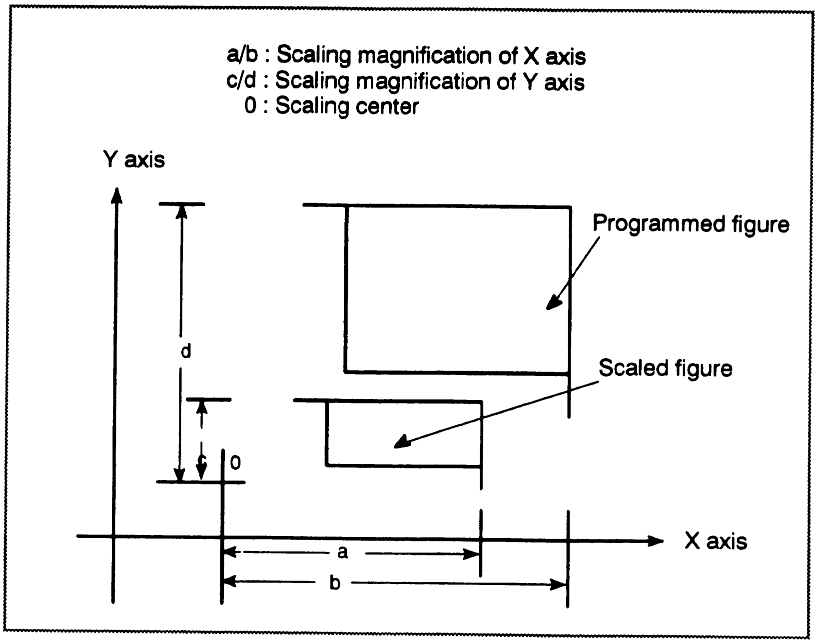


Fig14.5 (b) Scaling of each axis

- **Scaling of circular interpolation**

Even if different magnifications are applie to each axis in circular interpolation, the tool will not trace an ellipse.
When different magnifications are applied to axes and a circular interpolation is specified with radius R, it becomes as following figure 14.5 (c) (in the example shown below, a magnification of 2 is applied to the X–component and a magnification of 1 is applied to the Y–component.).

```
G90 G00 X0.0 Y100.0 ;
G51 X0.0 Y0.0 Z0.0 I2000 J1000;
G02 X100.0 Y0.0 R100.0 F500 ;
```

Above commands are equivalent to the following command:

```
G90 G00 X0.0 Y100.0 Z0.0 ;
G02 X200.0 Y0.0 R200.0 F500 ;
```

Magnification of radius R depends on I, or J whichever is larger.

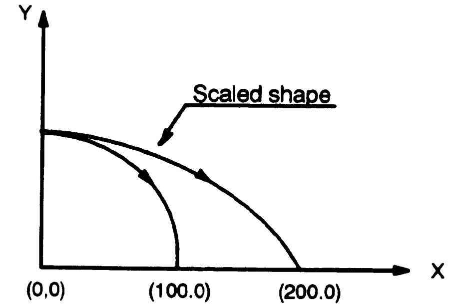


Fig14.5 (c) Scaling for circular interpolation1

When different magnifications are applied to axes and a circular interpolation is specified with I, J and K, it becomes as following figure 14.5 (d) (In the example shown below, a magnification of 2 is applied to the X–component and a magnification of 1 is applied to the Y–component.).

```
G90 G00 X0.0 Y0.0 ;
G51 X0.0 Y0.0 I2000 J1000;
G02 X100.0 Y0.0 I0.0 J–100.0 F500 ;
```

Above commands are equivalent to the following commands.

```
G90 G00 X0.0 Y100.0;
G02 X200.0 Y0.0 I0.0 J–100.0 F500 ;
```

In this case, the end point does not beet the radius, a linear section is included.

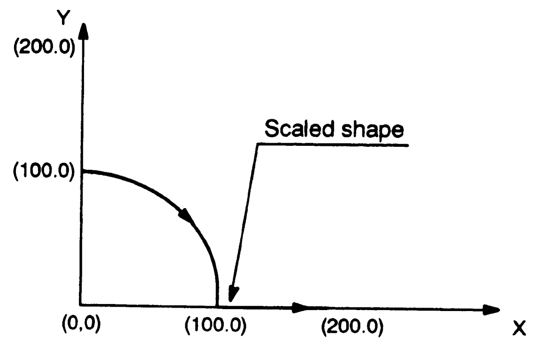


Fig14.5 (d) Scaling for circular interpolation 2

• Invalid scaling

This scaling is not applicable to cutter compensation values and tool length offset values (Fig. 14.5 (e)).

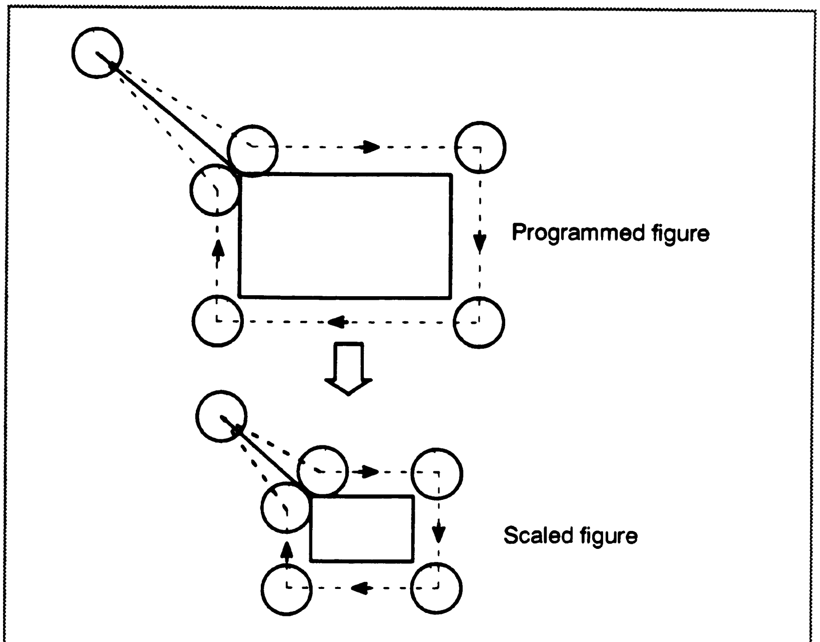


Fig14.5 (e) Scaling during cutter compensation

Scaling is not applicable to the Z–axis movement in case of the following canned cycle.
·Cut–in value Q and retraction value d of peck drilling cycle (G83, G73).
·Fine boring cycle (G76)
·Shift value Q of X and Y axes in back boring cycle (G87).
In manual operation, the travel distance cannot be increased or decreased using the scaling function.

Notes

1. The position display represents the coordinate value after scaling.
2. If a parameter setting value is employed as a scaling magnification without specifying P, the setting value at G51 command time is employed as the scaling magnification, and a change of this value, if any, is not effective.
3. Before specifying the G code for reference position return (G27, G28, G29, G30) or coordinate system setting (G92), cancel the scaling mode.
4. If scaling results are rounded by counting fractions of 5 and over as a unit and disregarding the rest, the move amount may become zero. In this case, the block is regarded as a no movement block, and therefore, it may affect the tool movement by cutter compensation C. See the description of blocks that do not move the tool at subsection 15.6.3.
5. When a mirror image was applied to one axis of the specified plane, the following results:
   (1)Circular command .... Direction of rotation is reversed.
   (2)Cutter compensation C .... Offset direction is reversed.
   (3)Coordinate system rotation .... Rotation angle is reversed.

Examples

Example of a mirror image program
Subprogram
O9000 ;
G00 G90 X60.0 Y60.0;
G01 X100.0 F100;
G01 Y100.0;
G01 X60.0 Y60.0;
M99;
Main program
N10 G00 G90;
N20M98P9000;
N30 G51 X50.0 Y50.0 I–1000 J1000;
N40 M98 P9000;
N50 G51 X50.0 Y50.0 I–1000 J–1000;
N60 M98 P9000;
N70 G51 X50.0 Y50.0 I1000 J–1000
N80 M98 P9000;
N90 G50;

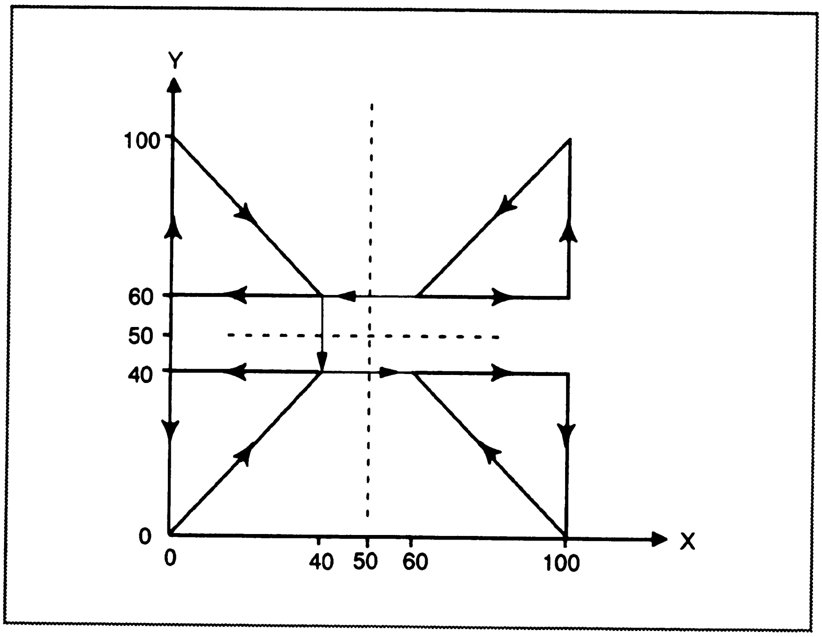


Fig14.5 (f) Example of a mirror image program

14.6 COORDINATE SYSTEM ROTATION (G68, G69)

A programmed shape can be rotated. By using this function it becomes possible, for example, to modify a program using a rotation command when a workpiece has been placed with some angle rotated from the programmed position on the machine.Further, when there is a pattern comprising some identical shapes in the positions rotated from a shape, the time required for programming and the length of the program can be reduced by preparing a subprogram of the shape and calling it after rotation.

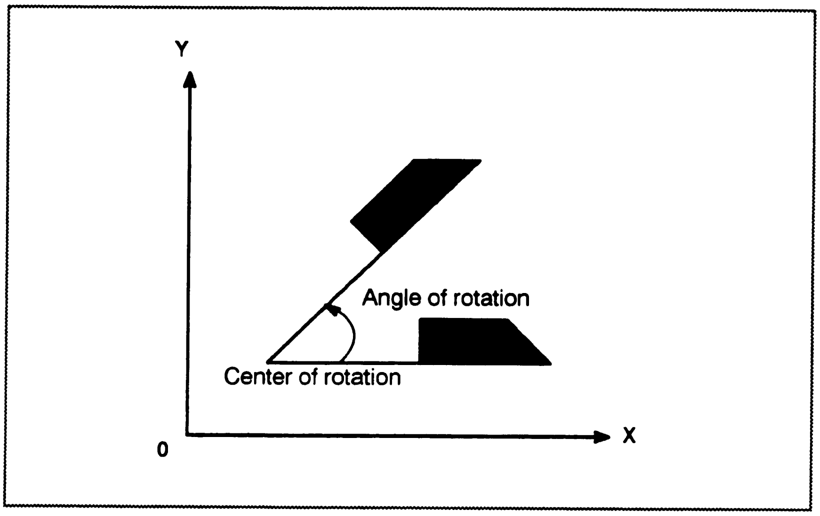


Fig14.6 (a) Coordinate system rotation

Format

Format

```
⎧G17⎫
⎨G18⎬G68 α_β_R_ ; Start rotation of a coordinate system.
⎩G19⎭
  :              ⎫ Coordinate system rotation mode
  :              ⎭ (The coordinate system is rotated.)
G69 ;              Coordinate system rotation cancel command
```

Meaning of command

G17 (G18 or G19) : Select the plane in which contains the figure to be rotated.

α_β_ Absolute command for two of the x_,y_,and Z_ axes that correspond to the current plane selected by a command (G17, G18, or G19). The command specifies the coordinates of the center of rotation for the values specified subsequent to G68.

R_ Angular displacement with a positive value indicates counter clockwise rotation. Parameter RIN (No. 5400#0) selects whether the specified angular displacement is alwarys considered an absolute value or is considered an absolute or incremental value depending on the specified G code (G90 or G91).

Least input increment : 0.001 deg
Valid data range : –360,000 A360,000

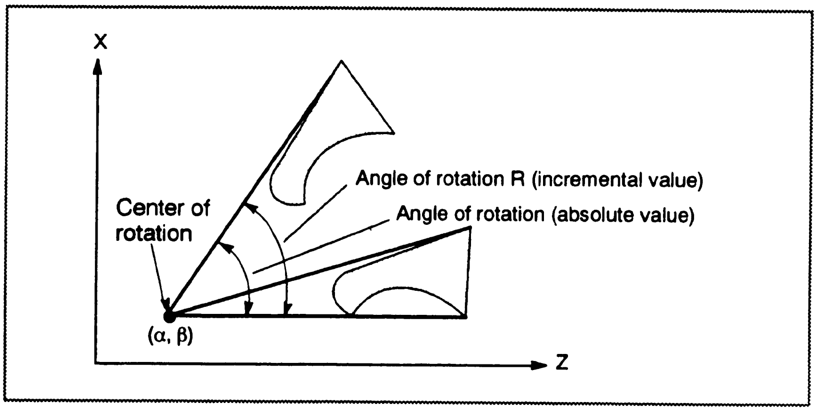


Fig14.6 (b) Coordinate system rotation

> **Note**
> When a decimal fraction is used to specify angular displacement (R_), the 1's digit corresponds to degree units.

Explanations

• G code for selecting a plane: G17,G18 or G19

The G code for selecting a plane (G17,G18,or G19) can be specified before the block containing the G code for coordinate system rotation (G68). G17, G18 or G19 must not be designated in the mode of coordinate system rotation.
As for the incremental position commands designated between the G68 block and a block with an absolute command; it is regarded that the position where G68 was designated is the center of rotation (Fig. 14.6 (c)).
When α_ and β_ are omitted, the position where G68 is commanded is set as the center of rotation.
When angle of rotation is omitted, the value set to parameter (No. 5410) is regarded as the rotation angle. The coordinate system rotation is cancelled by G69;
G69 may be designated in the same block as the other commands. Tool offset, such as cutter compensation, or tool length offset is performed after the coordinate system is rotated for the command program.

> **Note**
> If move commands follow the G69 command, the first must be an absolute command.

```
N1 G92 X–500.0 Y–500.0 G69 G17 ;
N2 G68 X700.0 Y300.0 R60.0 ;
N3 G90 G01 X0 Y0 F200 ;
        (G91X500.0Y500.0)
N4 G91 X1000.0 ;
N5 G02 Y1000.0 R1000.0 ;
N6 G03 X–1000.0 I–500.0 J–500.0 ;
N7 G01 Y–1000.0 ;
N8 G69 G90 X–500.0 Y–500.0 M02 ;
```

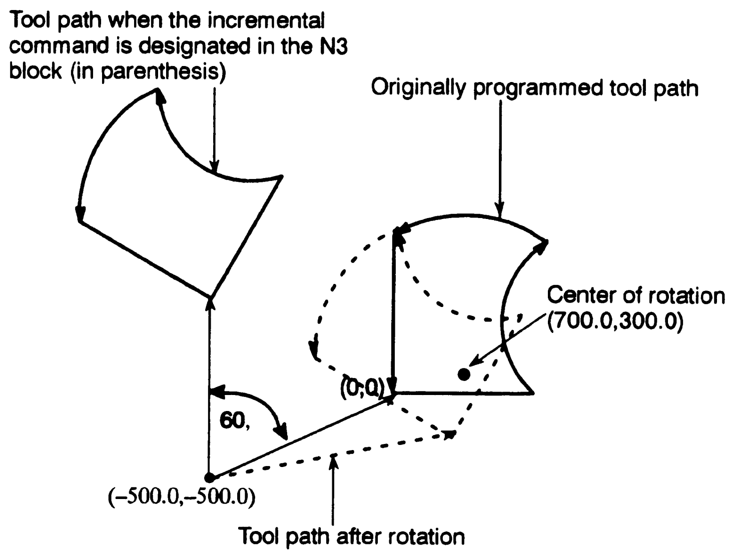


Fig14.6 (c) Absolute/Incremental command during coordinate system rotation

Examples

• Cutter compensation C and coordinate system rotation

It is possible to specify G68 and G69 in cutter compensation C mode.
The rotation plane must coincide with the plane of cutter compensation C.

```
N1 G92 X0 Y0 G69 G01 ;
N2 G42 G90 X1000 Y1000 F1000 D01 ;
N3 G68 R–30000 ;
N4 G91 X2000 ;
N5 G03 Y1000 R1000 J500 ;
N6 G01 X–2000 ;
N7 Y–1000 ;
N8 G69 G40 G90 X0 Y0 M30 ;
```

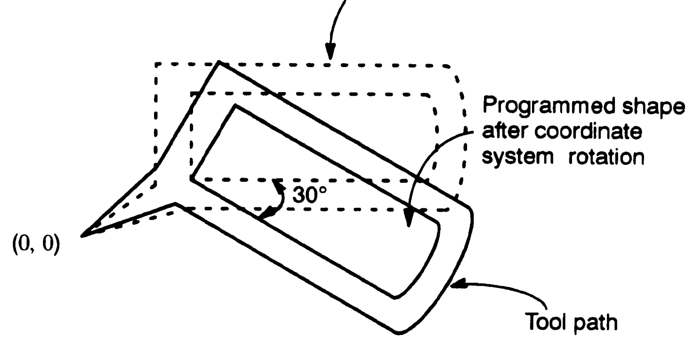


Fig14.6 (d) Cutter compensation C and coordinate system rotation

• Scaling and coordinate system rotation

If a coordinate system rotation command is executed in the scaling mode (G51 mode), the coordinate value (α,β,) of the rotation center will also be scaled, but not the rotation angle (R). When a move command is issued, the scaling is applied first and then the coordinates are rotated.
A coordinate system rotation command (G68) should not be issued in cutter compensation C mode (G41, G42) on scaling mode (G51). The coordinate system rotation command should always be specified prior to setting the cutter compensation C mode.

1. When the system is not in cutter compensation mode C, specify the commands in the following order :

G51 ;	scaling mode start
G68 ;	coordinate system rotation mode start
:	
G69 ;	coordinate system rotation mode cancel
G50 ;	scaling mode cancel

2. When the system is in cutter compensation model C, specify the commands in the following order (Fig.14.6(e)) :
   (cutter compensation C cancel)
   G51 ; scaling mode start
   G68 ; coordinate system rotation start
   :
   G41 ; cutter compensation C mode start
   :

```
G92 X0 Y0 ;
G51 X300.0  Y150.0  P500 ;
G68 X200.0  Y100.0  R45.0 ;
G01 X400.0  Y100.0 ;
Y100.0 ;
X–200.0 ;
Y–100.0 ;
X200.0 ;
```

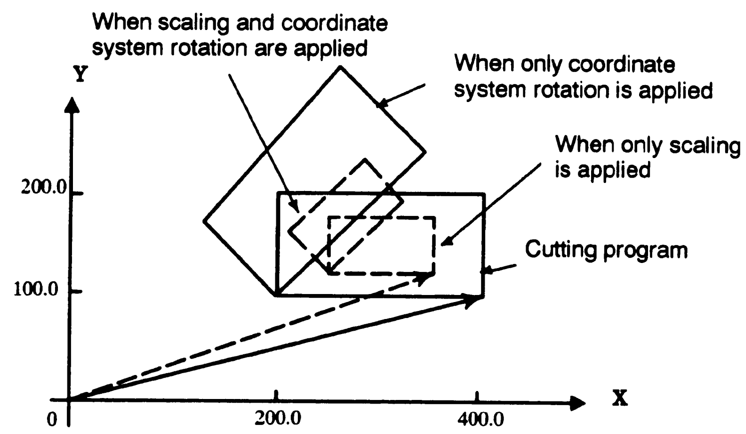


Fig14.6 (e) Scaling and coordinate system rotation in cutter compensation C mode

- **Repetitive commands for coordinate system rotation**

It is possible to store one program as a subprogram and recall subprogram by changing the angle.

Sample program for when the RIN bit (bit 0 of parameter 5400) is set to 1.
The specified angular displancement is treated as an absolute or incremental value depending on the specified G code (G90 or G91).

```
   G92 X0 Y0 G69 G17;
   G01 F200 H01 ;
   M98 P2100 ;
   M98 P072200 ;
   G00 G90 X0 Y0 M30 ;

O 2200 G68 X0 Y0 G91 R45.0 ;
   G90 M98 P2100 ;
   M99 ;

O 2100 G90 G01 G42 X0 Y– (MINUS) 10.0 ;
   X4.142 ;
   X7.071  Y– (MINUS) 7.071 ;
   G40 ;
   M99 ;
```

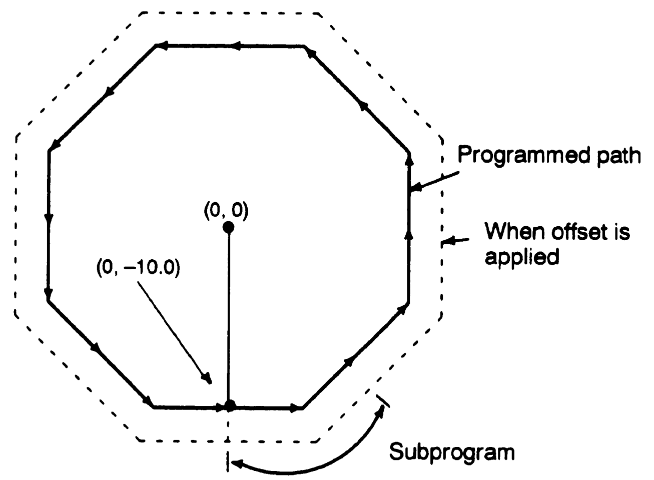
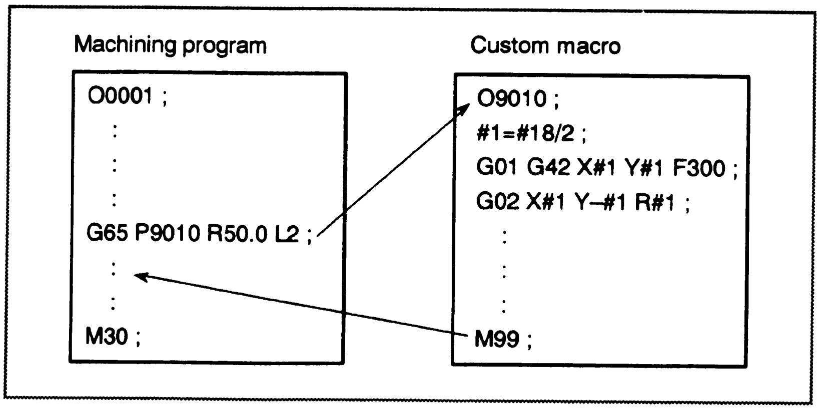


Fig14.6 (f) Coordinate system rotation command

15 CUSTOM MACRO

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user–defined canned cycles.A machining program can call a custom macro with a simple command, just like a subprogram.



15.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 and X100.0.
With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI panel.

```
#1=#2+100 ;
G01 X#1 F300 ;
```

Explanation

• Variable representation

When specifying a variable, specify a number sign (#) followed by a variable number. Personal computers allow a name to be assigned to a variable, but this capability is not available for custom macros.

Example: #1

An expression can be used to specify a variable number. In such a case, the expression must be enclosed in brackets.

Example: #[#1+#2–12]

• Types of variables

Variables are classified into four types by variable number.

Table 15.1 Types of variables

Variable number	Type of variable	Function
#0	Always null	This variable is always null. No value can be assigned to this variable.
#1 – #33	Local variables	Local variables can only be used within a macro to hold data such as the results of operations. When the power is turned off, local variables are initialized to null. When a macro is called, arguments are assigned to local variables.
#100 – #149 #500 – #531	Common variables	Common variables can be shared among different macro programs. When the power is turned off, variables #100 to #149 are initialized to null. Variables #500 to #531 hold data even when the power is turned off.
#1000 –	System variables	System variables are used to read and write a variety of CNC data items such as the current position and tool compensation values.

• Range of variable values

Local and common variables can have value 0 or a value in the following ranges :

-10_{47} to -10_{-29}

10_{-29} to 10_{47}

If the result of calculation turns out to be invalid, an alarm No. 111 is issued.

- **Omission of the decimal point**

When a variable value is defined in a program, the decimal point can be omitted.

Example:
When #1=123; is defined, the actual value of variable #1 is 123.000.

- **Referencing variables**

To reference the value of a variable in a program, specify a word address followed by the variable number. When an expression is used to specify a variable, enclose the expression in brackets.

Example: G01X[#1+#2]F#3;

A referenced variable value is automatically rounded according to the least input increment of the address.

Example:
When G00X#1; is executed on a 1/1000–mm CNC with 12.3456 assigned to variable #1, the actual command is interpreted as G00X12.346;.

To reverse the sign of a referenced variable value, prefix a minus sign (–) to #.

Example: G00X–#1;

When an undefined variable is referenced, the variable is ignored up to an address word.

Example:
When the value of variable #1 is 0, and the value of variable #2 is null, execution of G00X#1Y#2; results in G00X0;.

- **Undefined variable**

When the value of a variable is not defined, such a variable is referred to as a "null" variable. Variable #0 is always a null variable. It cannot be written to, but it can be read.

(a) Quotation
When an undefined variable is quotated, the address itself is also ignored.

When #1 = < vacant >	When #1 = 0
G90 X100 Y#1 ↓ G90 X100	G90 X100 Y#1 ↓ G90 X100 Y0

(b) Operation
< vacant > is the same as 0 except when replaced by < vacant>

When #1 = < vacant >	When #1 = 0
#2 = #1 ↓ #2 = < vacant >	#2 = #1 ↓ #2 = 0
#2 = #1*5 ↓ #2 = 0	#2 = #1*5 ↓ #2 = 0
#2 = #1+#1 ↓ #2 = 0	#2 = #1 + #1 ↓ #2 = 0

(c) **Conditional expressions**
< vacant > differs from 0 only for EQ and NE.

When #1 = < vacant >	When #1 = 0
#1 EQ #0 ↓ Established	#1 EQ #0 ↓ Not established
#1 NE 0 ↓ Established	#1 NE 0 ↓ Not established
#1 GE #0 ↓ Established	#1 GE #0 ↓ Established
#1 GT 0 ↓ Not established	#1 GT 0 ↓ Not established

- **Displaying variable values**

Procedure for displaying variable values

Procedure

1 Press the [OFFSET SETTING] key to display the tool compensation screen.

2 Press the continuous menu key [▷].

3 Press the soft key **[MACRO]** to display the macro variable screen.

4 Enter a variable number, then press soft key **[NO.SRH]**.
The cursor moves to the position of the entered number.

```
VARIABLE                                        O1234 N12345
 NO.          DATA          NO.                  DATA
 100          123.456       108
 101            0.000       109
 102                        110
 103                        111
 104                        112
 105                        113
 106                        114
 107                        115

ACTUAL POSITION (RELATIVE)
   X          0.000          Y            0.000
   Z          0.000          B            0.000

MEM **** *** ***           18:42:15

[ MACRO ] [        ] [  OPR  ] [        ] [ (OPRT) ]
```

·When the value of a variable is blank, the variable is null.
·The mark ******** indicates an overflow (when the absolute value of a variable is greater than 99999999) or an underflow (when the absolute value of a variable is less than 0.0000001).

Limitations

Program numbers and sequence numbers cannot be referenced using variables.

Example:
Variables cannot be used in the following ways:
O#1;
N#3Y200.0;

15.2 SYSTEM VARIABLES

System variables can be used to read and write internal CNC data such as tool compensation values and current position data. Note, however, that some system variables can only be read. System variables are essential for automation and general–purpose program development.

Explanations

• Interface signals

Signals can be exchanged between the programmable machine controller (PMC) and custom macros.

Table 15.2(a) System variables for interface signals

Variable number	Function
#1000–#1015 #1032	A 16–bit signal can be sent from the PMC to a custom macro. Variables #1000 to #1015 are used to read a signal bit by bit. Variable #1032 is used to read all 16 bits of a signal at one time.
#1100–#1115 #1132	A 16–bit signal can be sent from a custom macro to the PMC. Variables #1100 to #1115 are used to write a signal bit by bit. Variable #1132 is used to write all 16 bits of a signal at one time.
#1133	Variable #1133 is used to write all 32 bits of a signal at one time from a custom macro to the PMC. Note, that values from –99999999 to +99999999 can be used for #1133.

For detailed information, refer to the connection manual (B–62173E).

• Tool compensation values

Tool compensation values can be read and written using system variables. Usable variable numbers depend on the number of compensation pairs, whether a distinction is made between geometric compensation and wear compensation, and whether a distinction is made between tool length compensation and cutter compensation. When the number of compensation pairs is not greater than 200, variables #2001 to #2400 can also be used.

Table 15.2(b) System variables for tool compensation memory

Compensation number	System variable
1 : 99	#10001 (#2001) : #10099 (#2099)

• Macro alarms

Table 15.2(c) System variable for macro alarms

Variable number	Function
#3000	When a value from 0 to 200 is assigned to variable #3000, the CNC stops with an alarm. After an expression, an alarm message not longer than 26 characters can be described. The CRT screen displays alarm numbers by adding 3000 to the value in variable #3000 along with an alarm message.

Example:
#3000=1(TOOL NOT FOUND);
→ The alarm screen displays "3001 TOOL NOT FOUND."

• Time information

Time information can be read and written.

Table 15.2(d) System variables for time information

Variable number	Function
#3001	This variable functions as a timer that counts in 1–millisecond increments at all times. When the power is turned on, the value of this variable is reset to 0. When 65535 milliseconds is reached, the value of this timer returns to 0.
#3002	This variable functions as a timer that counts in 1–hour increments when the cycle start lamp is on. This timer preserves its value even when the power is turned off. When 1145324.612 hours is reached, the value of this timer returns to 0.
#3011	This variable can be used to read the current date (year/month/day). Year/month/day information is converted to an apparent decimal number. For example, September 28, 1994 is represented as 19940928.
#3012	This variable can be used to read the current time (hours/minutes/seconds). Hours/minutes/seconds information is converted to an apparent decimal number. For example, 34 minutes and 56 seconds after 3 p.m. is represented as 153456.

• Automatic operation control

The control state of automatic operation can be changed.

Table 15.2(e) System variable (#3003) for automatic operation control

#3003	Single block	Completion of an auxiliary function
0	Enabled	To be awaited
1	Disabled	To be awaited
2	Enabled	Not to be awaited
3	Disabled	Not to be awaited

- When the power is turned on, the value of this variable is 0.
- When single block stop is disabled, single block stop operation is not performed even if the single block switch is set to ON.
- When a wait for the completion of auxiliary functions (M, S, and T functions) is not specified, program execution proceeds to the next block before completion of auxiliary functions. Also, distribution completion signal DEN is not output.

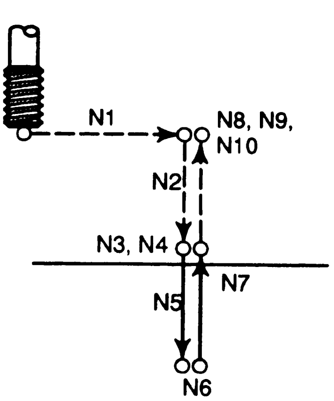
Table 15.2(f) System variable (#3004) for automatic operation control

#3004	Feed hold	Feedrate Override	Exact stop
0	Enabled	Enabled	Enabled
1	Disabled	Enabled	Enabled
2	Enabled	Disabled	Enabled
3	Disabled	Disabled	Enabled
4	Enabled	Enabled	Disabled
5	Disabled	Enabled	Disabled
6	Enabled	Disabled	Disabled
7	Disabled	Disabled	Disabled

- When the power is turned on, the value of this variable is 0.
- When feed hold is disabled:

  (1) When the feed hold button is held down, the machine stops in the single block stop mode. However, single block stop operation is not performed when the single block mode is disabled with variable #3003.

  (2) When the feed hold button is pressed then released, the feed hold lamp comes on, but the machine does not stop; program execution continues and the machine stops at the first block where feed hold is enabled.
- When feedrate override is disabled, an override of 100% is always applied regardless of the setting of the feedrate override switch on the machine operator's panel.
- When exact stop check is disabled, no exact stop check (position check) is made even in blocks including those which do not perform cutting.




```
O0001 ;
N1  G00 G91 X#24 Y#25 ;
N2  Z#18 ;
    G04 ;
N3  #3003=3 ;
N4  #3004=7 ;
N5  G01 Z#26 F#9 ;
N6  M04 ;
N7  G01
Z–[ROUND[#18]+ROUND[#26]] ;
    G04 ;
N8  #3004=0 ;
N9  #3003=0 ;
N10 M03 ;
    M99 ;
```

Fig. 15.2(a) Example of using variable #3004 in a tapping cycle

• Settings

Settings can be read and written. Binary values are converted to decimals.

#3005

	#15	#14	#13	#12	#11	#10	#9	#8
Setting								

	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC

#5 (SEQ) : Whether to automatically insert sequence numbers
#2 (INI) : Millimeter input or inch input
#1 (ISO) : Whether to use EIA or ISO as the output code
#0 (TVC) : Whether to make a TV check

• Mirror image

The mirror–image status for each axis set using an external switch or setting operation can be read through the output signal (mirror–image check signal). The mirror–image status present at that time can be checked. (See III–4.8.)
The value obtained in binary is converted into decimal notation.

#3007

	#7	#6	#5	#4	#3	#2	#1	#0
Setting						3th axis	2th axis	1th axis

For each bit, 0 (mirror–image function is disabled) or 1 (mirror–image function is enabled) is indicated.

Example : If #3007 is 3, the mirror–image function is enabled for the first and second axes.

- When the mirror–image function is set for a certain axis by both the mirror–image signal and setting, the signal value and setting value are ORed and then output.
- When mirror–image signals for axes other than the controlled axes are turned on, they are still read into system variable #3007.
- ·System variable #3007 is a write–protected system variable. If an attempt is made to write data in the variable, P/S 116 alarm "WRITE PROTECTED VARIABLE" is issued.

• Number of machined parts

The number (target number) of parts required and the number (completion number) of machined parts can be read and written.

Table 15.2(g) System variables for the number of parts required and the number of machined parts

Variable number	Function
#3901	Number of machined parts (completion number)
#3902	Number of required parts (target number)

Note

Do not substitute a negative value.

• Modal information

Modal information specified in blocks up to the immediately preceding block can be read.

Table 15.2(h) System variables for modal information

Variable number	Function	
#4001	G00, G01, G02, G03	(Group 01)
#4002	G17, G18, G19	(Group 02)
#4003	G90, G91	(Group 03)
#4004		(Group 04)
#4005	G94	(Group 05)
#4006	G20, G21	(Group 06)
#4007	G40, G41, G42	(Group 07)
#4008	G43, G44, G49	(Group 08)
#4009	G73, G74, G76, G80–G89	(Group 09)
#4010	G98, G99	(Group 10)
#4011	G50, G51	(Group 11)
#4012	G65, G66, G67	(Group 12)
#4014	G54–G59	(Group 14)
#4015	G61–G64	(Group 15)
#4016	G68, G69	(Group 16)
:	:	:
#4022		(Group 22)
#4107	D code	
#4109	F code	
#4111	H code	
#4113	M code	
#4114	Sequence number	
#4115	Program number	
#4119	S code	
#4120	T code	

Example:

When #1=#4001; is executed, the resulting value in #1 is 0, 1, 2, or 3.

- **Current position**

Position information cannot be written but can be read.

Table 15.2(l) System variables for position information

<table>
<tr><th>Variable number</th><th>Position information</th><th>Coordinate system</th><th>Tool compensation value</th><th>Read operation during movement</th></tr>
<tr><td>#5001–#5003</td><td>Block end point</td><td>Workpiece coordinate system</td><td>Not included</td><td>Enabled</td></tr>
<tr><td>#5021–#5023</td><td>Current position</td><td>Machine coordinate system</td><td rowspan="3">Included</td><td rowspan="2">Disabled</td></tr>
<tr><td>#5041–#5043</td><td>Current position</td><td rowspan="2">Workpiece coordinate system</td></tr>
<tr><td>#5061–#5063</td><td>Skip signal position</td><td>Enabled</td></tr>
<tr><td>#5081–#5083</td><td>Tool length conpensation value</td><td></td><td></td><td rowspan="2">Disabled</td></tr>
<tr><td>#5101–#5103</td><td>Deviated servo position</td><td></td><td></td></tr>
</table>

- The first digit (from 1 to 3) represents an axis number.
- The tool length compensation value currently used for execution rather than the immediately preceding tool offset value is held in variables #5081 to 5088.
- The tool position where the skip signal is turned on in a G31 (skip function) block is held in variables #5061 to #5068. When the skip signal is not turned on in a G31 block, the end point of the specified block is held in these variables.
- When read during movement is "disabled," this means that expected values cannot be read due to the buffering (preread) function.

- **Workpiece coordinate system compensation values (workpiece zero point offset values)**

Workpiece zero point offset values can be read and written.

Table 15.2(j) System variables for workpiece zero point offset values

Variable number	Function
#5201 : #5203	First–axis external workpiece zero point offset value : Third–axis external workpiece zero point offset value
#5221 : #5223	First–axis G54 workpiece zero point offset value : Third–axis G54 workpiece zero point offset value
#5241 : #5243	First–axis G55 workpiece zero point offset value : Third–axis G55 workpiece zero point offset value
#5261 : #5263	First–axis G56 workpiece zero point offset value : Third–axis G56 workpiece zero point offset value
#5281 : #5283	First–axis G57 workpiece zero point offset value : Third–axis G57 workpiece zero point offset value
#5301 : #5303	First–axis G58 workpiece zero point offset value : Third–axis G58 workpiece zero point offset value
#5321 : #5323	First–axis G59 workpiece zero point offset value : Third–axis G59 workpiece zero point offset value

15.3 ARITHMETIC AND LOGIC OPERATION

The operations listed in Table 15.3(a) can be performed on variables. The expression to the right of the operator can contain constants and/or variables combined by a function or operator. Variables #j and #K in an expression can be replaced with a constant. Variables on the left can also be replaced with an expression.

Table 15.3(a) Arithmetic and logic operation

Function	Format	Remarks
Definition	#i=#j	
Sum Difference Product Quotient	#i=#j+#k; #i=#j–#k; #i=#j*#k; #i=#j/#k;	
Sine Cosine Tangent Arctangent	#i=SIN[#j]; #i=COS[#j]; #i=TAN[#j]; #i=ATAN[#j]/[#k];	An angle is specified in degrees. 90 degrees and 30 minutes is represented as 90.5 degrees.
Square root Absolute value Rounding off Rounding down Rounding up	#i=SQRT[#j]; #i=ABS[#j]; #i=ROUND[#j]; #i=FIX[#j]; #i=FUP[#j];	
OR XOR AND	#i=#j OR #k; #i=#j XOR #k; #i=#j AND #k;	A logical operation is performed on binary numbers bit by bit.
Conversion from BCD to BIN Conversion from BIN to BCD	#i=BIN[#j]; #i=BCD[#j];	Used for signal exchange to and from the PMC

Explanations

• Angle units

The units of angles used with the SIN, COS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

• ATAN function

After the ATAN function, specify the lengths of two sides separated by a slash. A result is found where 0 ≦ result < 360.

Example :
When #1=ATAN[1]/[–1], the value of #1 is 135.0

• ROUND function

- When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.

  Example:
  When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.

- When the ROUND function is used in NC statement addresses, the ROUND function rounds off the specified value according to the least input increment of the address.

  Example:

  Creation of a drilling program that cuts according to the values of variables #1 and #2, then returns to the original position
  Suppose that the increment system is 1/1000 mm, variable #1 holds 1.2345, and variable #2 holds 2.3456. Then,

  G00 G91 X–#1; **Moves 1.235 mm.**
  G01 X–#2 F300; **Moves 2.346 mm.**
  G00 X[#1+#2];
  Since 1.2345 + 2.3456 = 3.5801, the travel distance is 3.580, which does not return the tool to the original position.
  This difference comes from whether addition is performed before or after rounding off. G00X–[ROUND[#1]+ROUND[#2]] must be specified to return the tool to the original position.

• Rounding up and down to an integer

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handling negative numbers.

Example:

Suppose that #1=1.2 and #2=–1.2.
When #3=FUP[#1] is executed, 2.0 is assigned to #3.
When #3=FIX[#1] is executed, 1.0 is assigned to #3.
When #3=FUP[#2] is executed, –2.0 is assigned to #3.
When #3=FIX[#2] is executed, –1.0 is assigned to #3.

• Abbreviations of arithmetic and logic operation commands

When a function is specified in a program, the first two characters of the function name can be used to specify the function.

Example:

ROUND → RO
FIX → FI

• Priority of operations

Functions
$\boxed{2}$ Operations such as multiplication and division (*, /, AND)
$\boxed{3}$ Operations such as addition and subtraction (+, –, OR, XOR)

Example) $\#1=\underbrace{\#2+\underbrace{\#3*\underbrace{SIN[\#4]}_{\boxed{1}}}_{\boxed{2}}}_{\boxed{3}};$

$\boxed{1}$, $\boxed{2}$ and $\boxed{3}$ indicate the order of operations.

- **Bracket nesting**

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, alarm No. 118 occurs.

Example) $\#1=\underbrace{SIN\ \underbrace{[\ \underbrace{[\ \underbrace{\underbrace{[\#2+\#3]}_{}\ *\#4}_{\boxed{2}}\ +\#5]}_{\boxed{3}}\ *\#6]}_{\boxed{4}}}_{\boxed{5}}$;

to $\boxed{5}$ indicate the order of operations.

Limitations

- **Brackets**

Brackets ([,]) are used to enclose an expression. Note that parentheses are used for comments.

- **Operation error**

Errors may occur when operations are performed.

Table 15.3(b) Errors involved in operations

Operation	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error(*1) $\lvert \frac{\varepsilon}{a} \rvert$
a = b / c	4.66×10^{-10}	1.88×10^{-9}	
$a = \sqrt{b}$	1.24×10^{-9}	3.73×10^{-9}	
a = b + c a = b – c	2.33×10^{-10}	5.32×10^{-10}	(*2) Min $\lvert \frac{\varepsilon}{b} \rvert$, $\lvert \frac{\varepsilon}{c} \rvert$
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error(*3) $\lvert \varepsilon \rvert$ degrees
a = ATAN [b] / [c] (*4)	1.8×10^{-6}	3.6×10^{-6}	

Notes

1. The relative error depends on the result of the operation.
2. Smaller of the two types of errors is used.
3. The absolute error is constant, regardless of the result of the operation.
4. Function TAN performs SIN/COS.

- The precision of variable values is about 8 decimal digits. When very large numbers are handled in an addition or subtraction, the expected results may not be obtained.

  Example:

  When an attempt is made to assign the following values to variables #1 and #2:
  #1=9876543210123.456
  #2=9876543277777.777
  the values of the variables become:
  #1=9876543200000.000
  #2=9876543300000.000
  In this case, when #3=#2–#1; is calculated, #3=100000.000 results. (The actual result of this calculation is slightly different because it is performed in binary.)

- Also be aware of errors that can result from conditional expressions using EQ, NE, GE, GT, LE, and LT.

  Example:

  IF[#1 EQ #2] is effected by errors in both #1 and #2, possibly resulting in an incorrect decision.
  Therefore, instead find the difference between the two variables with IF[ABS[#1–#2]LT0.001].
  Then, assume that the values of the two variables are equal when the difference does not exceed an allowable limit (0.001 in this case).

- Also, be careful when rounding down a value.

  Example:

  When #2=#1*1000; is calculated where #1=0.002;, the resulting value of variable #2 is not exactly 2 but 1.99999997.
  Here, when #3=FIX[#2]; is specified, the resulting value of variable #1 is not 2.0 but 1.0. In this case, round down the value after correcting the error so that the result is greater than the expected number, or round it off as follows:
  #3=FIX[#2+0.001]
  #3=ROUND[#2]

• Divisor

When a divisor of zero is specified in a division or TAN[90], alarm No. 112 occurs.

15.4 MACRO STATEMENTS AND NC STATEMENTS

The following blocks are referred to as macro statements:

- **Blocks containing an arithmetic or logic operation (=)**
- **Blocks containing a control statement (such as GOTO, DO, END)**
- **Blocks containing a macro call command (such as macro calls by G65, G66, G67, or other G codes, or by M codes)**

Any block other than a macro statement is referred to as an NC statement.

Explanations

• Differences from NC statements

- Even when single block mode is on, the machine does not stop. Note, however, that the machine stops in the single block mode when parameter SBM (No. 6000#5) is 1.
- Macro blocks are not regarded as blocks that involve no movement in the cutter compensation mode (see Section 15.7).

• NC statements that have the same property as macro statements

- NC statements that include a subprogram call command (such as subprogram calls by M98 or other M codes, or by T codes) and which do not include a command address other than O, N, P or L have the same property as macro statements.
- NC statements that include M99 and which do not include a command address other than O, N, L or P have the same property as macro statements..

15.5 BRANCH AND REPETITION

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:

Branch and repetition
- GOTO statement (unconditional branch)
- IF statement (conditional branch: if ..., then...)
- WHILE statement (repetition while ...)

15.5.1 Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999 is specified, alarm No. 128 occurs. A sequence number can also be specified using an expression.

GOTO n ; n: Sequence number (1 to 99999)

Example:
GOTO1;
GOTO#10;

15.5.2 Conditional Branch (IF Statement)

Specify a conditional expression after IF. If the specified conditional expression is satisfied, a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.

```
If the condition      IF [#1 GT 10] GOTO 2 ;
is not satisfied
                      Processing            If the condition is satisfied
                 N2 G00 G91 X10.0 ;
                      :
```

Explanations

• Conditional expression

A conditional expression must include an operator inserted between two variables or between a variable and constant, and must be enclosed in brackets ([,]). An expression can be used instead of a variable.

• Operators

Operators each consist of two letters and are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value. Note that the inequality sign cannot be used.

Table 15.5.2 Operators

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≧)
LT	Less than(<)
LE	Less than or equal to(≦)

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O9500;
#1=0; ................. Initial value of the variable to hold the sum
#2=1; ................. Initial value of the variable as an addend
N1 IF[#2 GT 10] GOTO 2; Branch to N2 when the addend is greater than 10
#1=#1+#2; ............. Calculation to find the sum
#2=#2+1; .............. Next addend
 GOTO 1; .............. Branch to N1
N2 M30;................ End of program
```

15.5.3 Repetition (While Statement)

Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If the specified condition is not satisfied, program execution proceeds to the block after END.

```
 ┌──────────────── WHILE [conditional expression] DO m ; (m=1,2,3)
 │                                │ ┌────────────┐
 │ If the condition  If the condition │ │ Processing │
 │ is not satisfied  is satisfied  │ │            │
 │                                ↓ └────────────┘
 │                              END m ;
 └─────────────────────────────→   :
```

Explanations

While the specified condition is satisfied, the program from DO to END after WHILE is executed. If the specified condition is not satisfied, program execution proceeds to the block after END. The same format as for the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The numbers 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, alarm No. 126 occurs.

• Nesting

The identification numbers (1 to 3) in a DO–END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), alarm No. 124 occurs.

1. The identification numbers (1 to 3) can be used as many times as required.

```
WHILE [ ... ] DO 1 ;
  Processing
END 1 ;
  :
WHILE [ ... ] DO 1 ;
  Processing
END 1 ;
```

2. DO ranges cannot overlap.

```
WHILE [ ... ] DO 1 ;
  Processing
WHILE [ ... ] DO 2 ;
  :
END 1 ;
  Processing
END 2 ;
```

3. DO loops can be nested to a maximum depth of three levels.

```
WHILE [ ... ] DO 1 ;
  :
  WHILE [ ... ] DO 2 ;
    :
    WHILE [ ... ] DO 3 ;
      Processing
    END 3 ;
    :
  END 2 ;
  :
END 1 ;
```

4. Control can be transferred to the outside of a loop.

```
WHILE [ ... ] DO 1 ;
IF [ ... ] GOTO n ;
END 1 ;
Nn
```

5. Branches cannot be made to a location within a loop.

```
IF [ ... ] GOTO n ;
  :
WHILE [ ... ] DO 1 ;
Nn ... ;
END 1 ;
```

Limitations

• Infinite loops

When DO m is specified without specifying the WHILE statement, an infinite loop ranging from DO to END is produced.

• Processing time

When a branch to the sequence number specified in a GOTO statement occurs, the sequence number is searched for. For this reason, processing in the reverse direction takes a longer time than processing in the forward direction. Using the WHILE statement for repetition reduces processing time.

• Undefined variable

In a conditional expression that uses EQ or NE, a <null value> and 0 (zero) have different effects. In other types of conditional expressions, a <null value> is regarded as zero.

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

15.6 MACRO CALL

A macro program can be called using the following methods:

```
Macro call ──┬── Simple call (G65)
             ├── modal call (G66, G67)
             ├── Macro call with G code
             ├── Macro call with M code
             ├── Subprogram call with M code
             └── Subprogram call with T code
```

Limitations

• Differences between macro calls and subprogram calls

Macro call (G65) differs from subprogram call (M98) as described below.

- With G65, an argument (data passed to a macro) can be specified. M98 does not have this capability.
- When an M98 block contains another NC command (for example, G01 X100.0 M98Pp), the subprogram is called after the command is executed. On the other hand, G65 unconditionally calls a macro.
- When an M98 block contains another NC command (for example, G01 X100.0 M98Pp), the machine stops in the single block mode. On the other hand, G65 does not stops the machine.
- With G65, the level of local variables changes. With M98, the level of local variables does not change.

15.6.1 Simple Call (G65)

When G65 is specified, the custom macro specified at address P is called. Data (argument) can be passed to the custom macro program.

```
G65 P p L ℓ <argument-specification> ;
  P          : Number of the program to call
  ℓ          : Repetition count (1 by default)
  Argument   : Data passed to the macro
```

```
O0001 ;                               O9010 ;
  :                                    #3=#1+#2 ;
G65 P9010 L2 A1.0 B2.0 ;  ──────►     IF [#3 GT 360] GOTO 9 ;
  :  ◄──────────────────────────       G00 G91 X#3 ;
M30 ;                                  N9 M99 ;
```

Explanations

• Call

- After G65, specify at address P the program number of the custom macro to call.
- When a number of repetitions is required, specify a number from 1 to 9999 after address L. When L is omitted, 1 is assumed.
- By using argument specification, values are assigned to corresponding local variables.

- **Argument specification**

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N, and P once each. Argument specification II uses A, B, and C once each and also uses I, J, and K up to ten times. The type of argument specification is determined automatically according to the letters used.

Argument specification I

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11

Address	Variable number
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19

Address	Variable number
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

- Addresses G, L, N, O, and P cannot be used in arguments.
- Addresses that need not be specified can be omitted. Local variables corresponding to an omitted address are set to null.

Argument specification II

Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three–dimensional coordinates as arguments.

Address	Variable number
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11

Address	Variable number
K_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22

Address	Variable number
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

- Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

Limitations

- **Format**

G65 must be specified before any argument.

- **Mixture of argument specifications I and II**

The CNC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence.

- **Position of the decimal point**

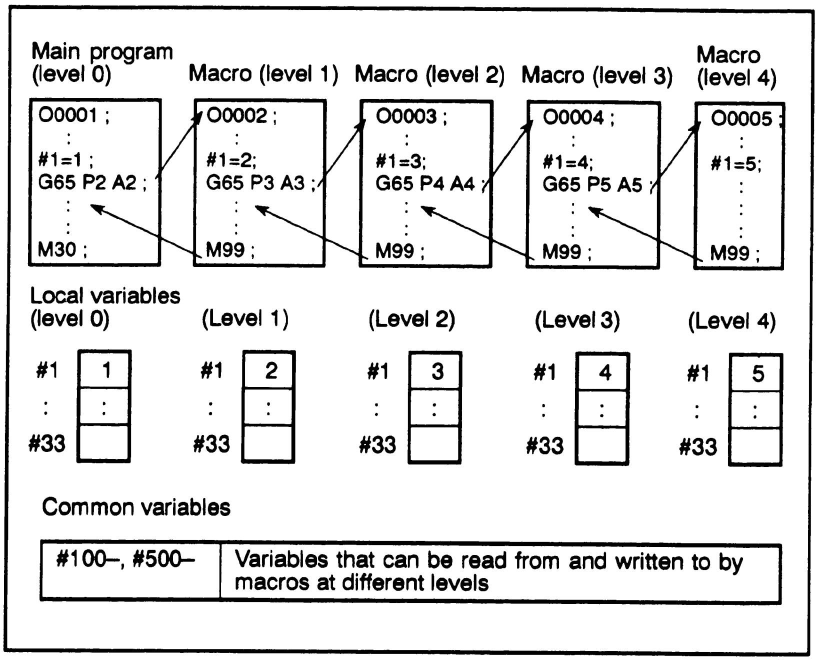
The units used for argument data passed without a decimal point correspond to the least input increment of each address. The value of an argument passed without a decimal point may vary according to the system configuration of the machine. It is good practice to use decimal points in macro call arguments to maintain program compatibility.

• Call nesting

Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).

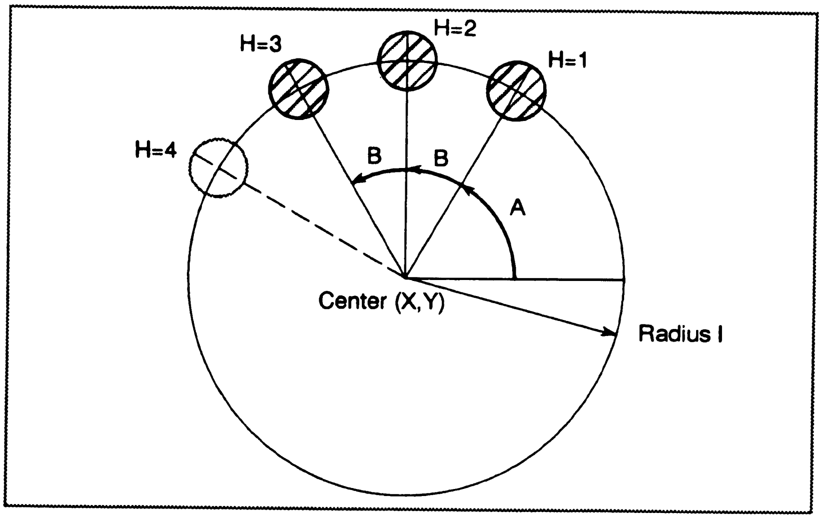
• Local variable levels

- Local variables from level 0 to 4 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G65 or G66), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC.
- When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.



Sample program (bolt hole circle)

A macro is created which drills H holes at intervals of B degrees after a start angle of A degrees along the periphery of a circle with radius I. The center of the circle is (X,Y). Commands can be specified in either the absolute or incremental mode. To drill in the clockwise direction, specify a negative value for B.



- **Calling format**

```
G65 P9100 Xx Yy Zz Rr Ff Ii Aa Bb Hh;
```

X : X coordinate of the center of the circle (absolute or incremental specification) . . . . (#24)
Y : Y coordinate of the center of the circle (absolute or incremental specification) . . . . (#25)
Z : Hole depth . . . . (#26)
R : Coordinates of an approach point . . . . (#18)
F : Cutting feedrate . . . . (#9)
I : Radius of the circle . . . . (#4)
A : Drilling start angle . . . . (#1)
B : Incremental angle (clockwise when a negative value is specified) . . . . (#2)
H : Number of holes . . . . (#11)

- **Program calling a macro program**

```
O0002;
G90 G92 X0 Y0 Z100.0;
G65 P9100 X100.0 Y50.0 R30.0 Z–50.0 F500 I100.0 A0 B45.0 H5;
M30;
```

- **Macro program (called program)**

```
O9100;
 #3=#4003; ........................... Stores G code of group 3.
 G81 Z#26 R#18 F#9 K0; (Note) .................... Drilling cycle.
                                           Note: L0 can also be used.
 IF[#3 EQ 90]GOTO 1; .......... Branches to N1 in the G90 mode.
 #24=#5001+#24; ......... Calculates the X coordinate of the center.
 #25=#5002+#25; ......... Calculates the Y coordinate of the center.
N1 WHILE[#11 GT 0]DO 1;
 ................... Until the number of remaining holes reaches 0
 #5=#24+#4*COS[#1]; ... Calculates a drilling position on the X–axis.
 #6=#25+#4*SIN[#1]; .... Calculates a drilling position on the Y–axis.
 G90 X#5 Y#6; .. Performs drilling after moving to the target position.
 #1=#1+#2; ................................ Updates the angle.
 #11=#11–1; .................... Decrements the number of holes.
 END 1;
 G#3 G80; ............... Returns the G code to the original state.
 M99;
```

Meaning of variables:
#3: Stores the G code of group 3. #5: X coordinate of the next hole to drill #6: Y coordinate of the next hole to drill

15.6.2 Modal Call (G66)

Once G66 is issued to specify a modal call a macro is called after a block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.

```
G66 P p L ℓ <argument–specification>
;
  P          : Number of the program to call
  ℓ          : Repetition count (1 by default)
  Argument   : Data passed to the macro
```

```
O0001 ;
  :
G66 P9100 L2 A1.0 B2.0 ;
G00 G90 X100.0 ;
Y200.0 ;
X150.0 Y300.0 ;
G67 ;
  :
M30 ;
```

```
O9100 ;
  :
G00 Z–#1 ;
G01 Z–#2 F300 ;
  :
  :
  :
  :
M99 ;
```

Explanations

• Call

- After G66, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 9999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.

• Cancellation

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

• Call nesting

Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).

• Modal call nesting

Modal calls can be nested by specifying another G66 code during a modal call.

Limitations

- In a G66 block, no macros can be called.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a miscellaneous function that does not involve movement along an axis.
- Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

Sample program

The same operation as the drilling canned cycle G81 is created using a custom macro and the machining program makes a modal macro call. For program simplicity, all drilling data is specified using absolute values.

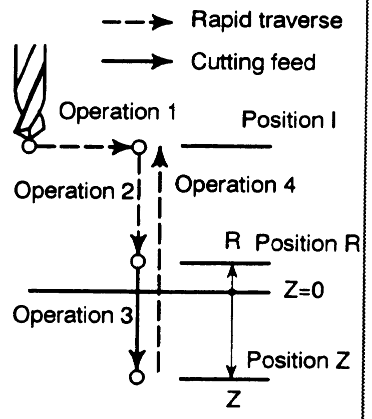
The canned cycle consists of the following basic operations:

Operation 1:
Positioning along the X–axis and Y–axis

Operation 2:
Rapid traverse to point R

Operation 3:
Cutting feed to point Z

Operation 4:
Rapid traverse to point R or I



- **Calling format**

```
G65 P9110 Xx Yy Zz Rr Ff Ll;
```

X : X coordinate of the hole (absolute specification only) ..... (#24)
Y : Y coordinate of the hole (absolute specification only) ..... (#25)
Z : Coordinates of position Z (absolute specification only) .... (#26)
R : Coordinates of position R (absolute specification only) .... (#18)
F : Cutting feedrate .... (#9)
L : Repetition count

- **Program that calls a macro program**

```
O0001;
G28 G91 X0 Y0 Z0;
G92 X0 Y0 Z50.0;
G00 G90 X100.0 Y50.0;
G66 P9110 Z–20.0 R5.0 F500;
G90 X20.0 Y20.0;
X50.0;
Y50.0;
X70.0 Y80.0;
G67;
M30;
```

- **Macro program (program called)**

```
O9110;
 #1=#4001; ................................ Stores G00/G01.
 #3=#4003; ................................ Stores G90/G91.
 #4=#4109; ......................... Stores the cutting feedrate.
 #5=#5003; ........... Stores the Z coordinate at the start of drilling.
 G00 G90 Z#18; ...................... Positioning at position R
 G01 Z#26 F#9; ...................... Cutting feed to position Z
 IF[#4010 EQ 98]GOTO 1; ................... Return to position I
 G00 Z#18; .......................... Positioning at position R
 GOTO 2;
N1 G00 Z#5; .......................... Positioning at position I
N2 G#1 G#3 F#4; .................. Restores modal information.
 M99;
```

15.6.3 Macro Call Using G Code

By setting a G code number used to call a macro program in a parameter, the macro program can be called in the same way as for a simple call (G65).

```
O0001 ;
  :
G81 X10.0 Y20.0 Z–10.0 ;
  :
M30 ;
```

```
O9010 ;
  :
  :
  :
N9 M99 ;
```

Parameter No. 6050 = 81

Explanations

By setting a G code number (from 1 to 9999) used to call a custom macro program (O9010 to O9019) in the corresponding parameter (No. 6050 to No. 6059), the macro program can be called in the same way as with G65. For example, when a parameter is set so that macro program O9010 can be called with G81, a user–specific cycle created using a custom macro can be called without modifying the machining program.

• Correspondence between parameter numbers and program numbers

Program number	Parameter number
O9010	6050
O9011	6051
O9012	6052
O9013	6053
O9014	6054
O9015	6055
O9016	6056
O9017	6057
O9018	6058
O9019	6059

• Repetition

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

• Argument specification

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitations

• Nesting of calls using G codes

In a program called with a G code, no macros can be called using a G code. A G code in such a program is treated as an ordinary G code. In a program called as a subprogram with an M or T code, no macros can be called using a G code. A G code in such a program is also treated as an ordinary G code.

15.6.4 Macro Call Using an M Code

By setting an M code number used to call a macro program in a parameter, the macro program can be called in the same way as with a simple call (G65).

```
O0001 ;                       O9020 ;
  :                             :
M50 A1.0 B2.0 ;   ───────▶      :
  :               ◀───────      :
M30 ;                         M99 ;

Parameter No. 6080 = 50
```

Explanations

By setting an M code number (from 1 to 99999999) used to call a custom macro program (O9020 to O9029) in the corresponding parameter (No. 6080 to No. 6089), the macro program can be called in the same way as with G65.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9020	6080
O9021	6081
O9022	6082
O9023	6083
O9024	6084
O9025	6085
O9026	6086
O9027	6087
O9028	6088
O9029	6089

- **Repetition**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

- **Argument specification**

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Limitations

- An M code used to call a macro program must be specified at the start of a block.
- In a macro called with a G code or in a program called as a subprogram with an M or T code, no macros can be called using an M code. An M code in such a macro or program is treated as an ordinary M code.

15.6.5 Subprogram Call Using an M Code

By setting an M code number used to call a subprogram (macro program) in a parameter, the macro program can be called in the same way as with a subprogram call (M98).

```
O0001 ;          O9001 ;
  :                :
M03 ;              :
  :                :
M30 ;            M99 ;

Parameter No. 6071 = 03
```

Explanations

By setting an M code number (from 1 to 99999999) used to call a subprogram in a parameter (No. 6071 to No. 6079), the corresponding custom macro program (O9001 to O9009) can be called in the same way as with M98.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9001	6071
O9002	6072
O9003	6073
O9004	6074
O9005	6075
O9006	6076
O9007	6077
O9008	6078
O9009	6079

- **Repetition**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

- **Argument specification**

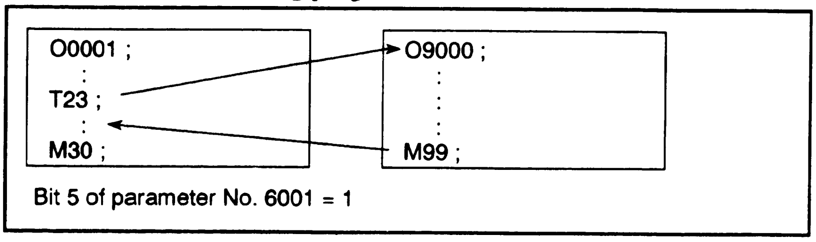
Argument specification is not allowed.

Limitations

In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using an M code. An M code in such a macro or program is treated as an ordinary M code.

15.6.6 Subprogram Calls Using a T Code

By enabling subprograms (macro program) to be called with a T code in a parameter, a macro program can be called each time the T code is specified in the machining program.



Explanations

- **Call**

By setting parameter (TCS) No. 6001#5 to 1, the macro program O9000 can be called when a T code is specified in the machining program. A T code specified in a machining program is assigned to common variable #149.

Limitations

In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using a T code. A T code in such a macro or program is treated as an ordinary T code.

15.6.7 Sample Program

By using the subprogram call function that uses M codes, the cumulative usage time of each tool is measured.

Conditions

- The cumulative usage time of each of tools T01 to T05 is measured. No measurement is made for tools with numbers greater than T05.
- The following variables are used to store the tool numbers and measured times:

#501	Cumulative usage time of tool number 1
#502	Cumulative usage time of tool number 2
#503	Cumulative usage time of tool number 3
#504	Cumulative usage time of tool number 4
#505	Cumulative usage time of tool number 5

- Usage time starts being counted when the M03 command is specified and stops when M05 is specified. System variable #3002 is used to measure the time during which the cycle start lamp is on. The time during which the machine is stopped by feed hold and single block stop operation is not counted, but the time used to change tools and pallets is included.

Operation check

- **Parameter setting**

Set 3 in parameter No. 6071, and set 05 in parameter No. 6072.

- **Variable value setting**

Set 0 in variables #501 to #505.

- **Program that calls a macro program**

```
O0001;
T01 M06;
M03;
 :
M05; ......................... Changes #501.
T02 M06;
M03;
 :
M05; ......................... Changes #502.
T03 M06;
M03;
 :
M05; ......................... Changes #503.
T04 M06;
M03;
 :
M05; ......................... Changes #504.
T05 M06;
M03;
 :
M05; ......................... Changes #505.
M30;
```

Macro program (program called)

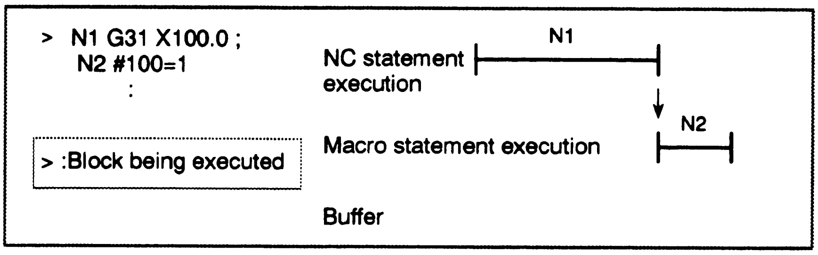
```
O9001(M03); .......................... Macro to start counting
 M01;
 IF[#4120 EQ 0]GOTO 9; ..................... No tool specified
 IF[#4120 GT 5]GOTO 9; .............. Out–of–range tool number
 #3002=0; ................................. Clears the timer.
N9 M03; ............. Rotates the spindle in the forward direction.
 M99;

O9002(M05); .......................... Macro to end counting
 M01;
 IF[#4120 EQ 0]GOTO 9; ..................... No tool specified
 IF[#4120 GT 5]GOTO 9; .............. Out–of–range tool number
 #[500+#4120]=#3002+#[500+#4120]; ..... Calculates cumulative time.

N9 M05; ................................. Stops the spindle.
 M99;
```

15.7 PROCESSING MACRO STATEMENTS

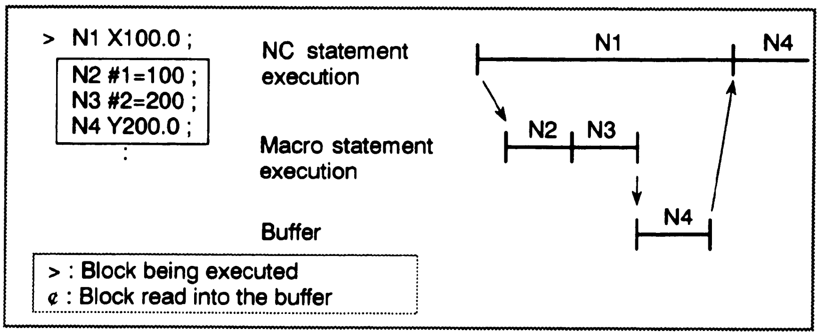
For smooth machining, the CNC prereads the NC statement to be performed next. This operation is referred to as buffering. In cutter compensation mode (G41, G42), the NC prereads NC statements two or three blocks ahead to find intersections. Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Blocks containing M00, M01, M02, or M30, blocks containing M codes for which buffering is suppressed by setting parameters No. 3411 to 3420, and blocks containing G31 are not preread.

Explanations

- **When the next block is not buffered (M codes that are not buffered, G31, etc.)**

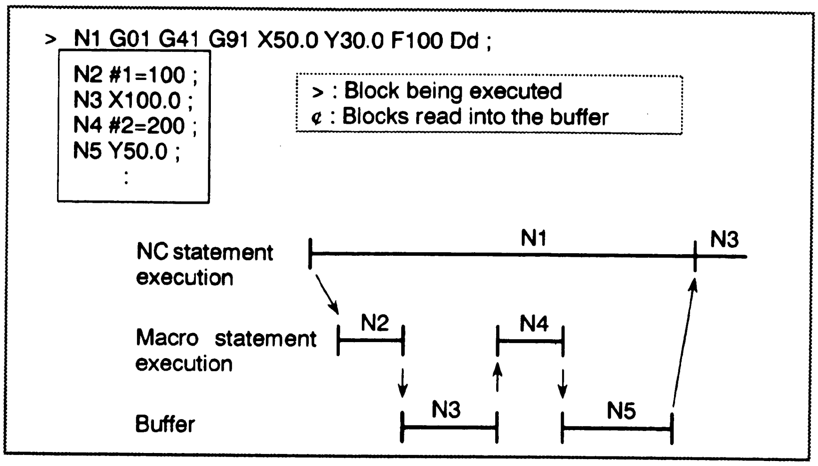


- **Buffering the next block in other than cutter compensation mode (G41, G42) (normally prereading one block)**



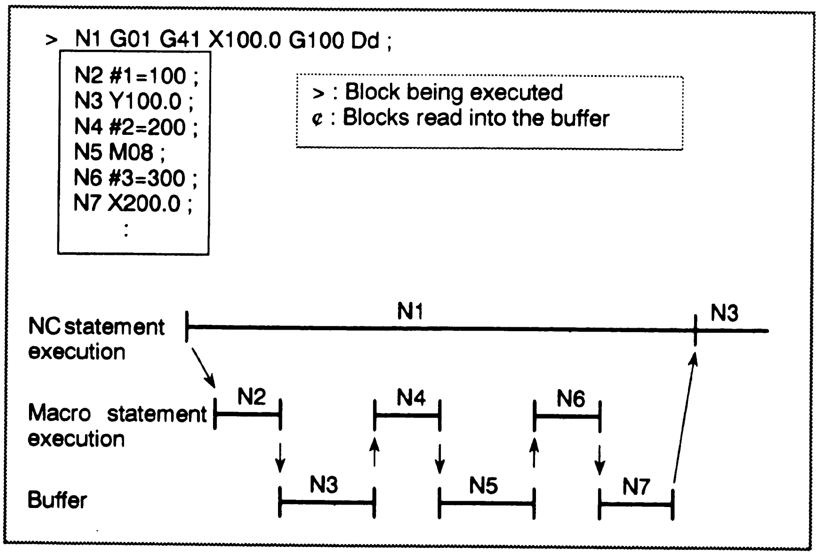
When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- **Buffering the next block in cutter compensation mode (G41, G42)**



When N1 is being executed, the NC statements in the next two blocks (up to N5) are read into the buffer. The macro statements (N2, N4) between N1 and N5 are processed during execution of N1.

- **When the next block involves no movement in cutter compensation C (G41, G42) mode**



When the NC1 block is being executed, the NC statements in the next two blocks (up to N5) are read into the buffer. Since N5 is a block that involves no movement, an intersection cannot be calculated. In this case, the NC statements in the next three blocks (up to N7) are read. The macro statements (N2, N4, and N6) between N1 and N7 are processed during execution of N1.

15.8 REGISTERING CUSTOM MACRO PROGRAMS

Custom macro programs are similar to subprograms. They can be registered and edited in the same way as subprograms. The storage capacity is determined by the total length of tape used to store both custom macros and subprograms.

15.9 LIMITATIONS

- **MDI operation**

The macro call command can be specified in MDI mode. During automatic operation, however, it is impossible to switch to the MDI mode for a macro program call.

- **Sequence number search**

A custom macro program cannot be searched for a sequence number.

- **Single block**

Even while a macro program is being executed, blocks can be stopped in the single block mode (except blocks containing macro call commands, arithmetic operation commands, and control commands).
A block containing a macro call command (G65, G66, or G67) does not stop even when the single block mode is on. Blocks containing arithmetic operation commands and control commands can be stopped in single block mode by setting SBM (bit 5 of parameter 6000) to 1.
Single block stop operation is used for testing custom macro programs.
Note that when a single block stop occurs at a macro statement in cutter compensation C mode, the statement is assumed to be a block that does not involve movement, and proper compensation cannot be performed in some cases. (Strictly speaking, the block is regarded as specifying a movement with a travel distance 0.)

- **Optional block skip**

A / appearing in the middle of an <expression> (enclosed in brackets [ ] on the right–hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specifier for an optional block skip code.

- **Operation in EDIT mode**

By setting NE8 (bit 0 of parameter 3202) and NE9 (bit 4 of parameter 3202) to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 8000 to 8999 and 9000 to 9999. This prevents registered custom macros or subprograms from being damaged or destroyed by erroneous operation. When the entire memory is cleared (by pressing the [7] and [9] keys at the same time to turn on the power), the contents of memory such as custom macro programs are deleted.

- **Reset**

Local variables and common variables #100 to #149 are cleared to null values by a reset operation. They can be prevented from being cleared by setting, CLV and CCV (bits 7 and 6 of parameter 6001). System variables #1000 to #1133 are not cleared.
A reset operation clears any called states of custom macro programs and subprograms, and any DO states, and returns control to the main program.

- **Display of the PROGRAM RESTART screen**

As with M98, the M and T codes used for subprogram calls are not displayed.

- **Feed hold**

When a feed hold is enabled during execution of a macro statement, the machine stops after execution of the macro statement. The machine also stops when a reset or alarm occurs.

- **Constant values that can be used in <expression>**

+0.0000001 to +99999999
–99999999 to –0.0000001
The number of significant digits is 8 (decimal). If this range is exceeded, alarm No. 003 occurs.

15.10 EXTERNAL OUTPUT COMMANDS

In addition to the standard custom macro commands, the following macro commands are available. They are referred to as external output commands.

- **BPRNT**
- **DPRNT**
- **POPEN**
- **PCLOS**

These commands are provided to output variable values and characters through the reader/punch interface.

Explanations

Specify these commands in the following order:

Open command: POPEN

Before specifying a sequence of data output commands, specify this command to establish a connection to an external input/output device.

Data output command: BPRNT or DPRNT

Specify necessary data output.

Close command: PCLOS

When all data output commands have completed, specify PCLOS to release a connection to an external input/output device.

• Open command POPEN

POPEN

POPEN establishes a connection to an external input/output device. It must be specified before a sequence of data output commands. The CNC outputs a DC2 control code.

• Data output command BPRNT

BPRNT [a #b [c] …]

- c — Number of significant decimal places
- #b — Variable
- a — Character

The BPRNT command outputs characters and variable values in binary.

(i) Specified characters are converted to corresponding codes according to the setting data (ISO) that is output at that time.
Specifiable characters are as follows:
- **Letters (A to Z)**
- **Numbers**
- **Special characters (*, /, +, –, etc.)**

An asterisk (*) is output by a space code.

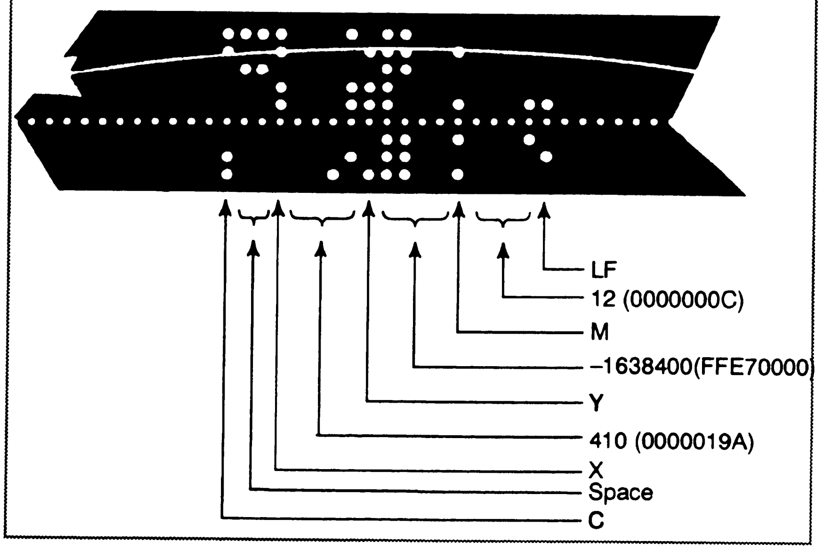
(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2–word (32–bit) data, including the decimal digits. It is output as binary data starting from the highest byte.

(iii) When specified data has been output, an EOB code is output according to the ISO code settings.

(iv) Null variables are regarded as 0.

Example)

BPRNT [C X#100 [3] Y#101 [3] M#10 [0]]**
Variable value
#100=0.40956
#101=–1638.4
#10=12.34



- **Data output command DPRNT**

DPRNT [a #b [c d] ...]

- c d: Number of significant decimal places
- c: Number of significant digits in the integer part
- #b: Variable
- a: Character

The DPRNT command outputs characters and each digit in the value of a variable according to the code set in the settings (ISO).

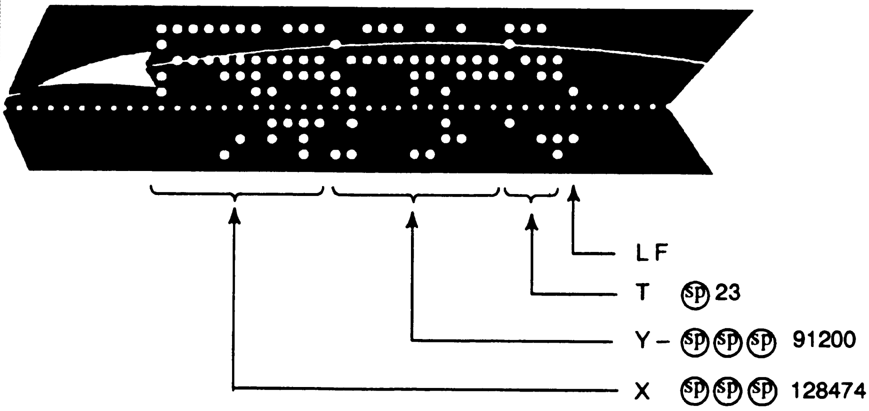
(i) For an explanation of the DPRNT command, see Items (i), (iii), and (iv) for the BPRNT command.

(ii) When outputting a variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in brackets.
One code is output for each of the specified number of digits, starting with the highest digit. For each digit, a code is output according to the settings (ISO). The decimal point is also output using a code set in the settings (ISO).
Each variable must be a numeric value consisting of up to eight digits. When high–order digits are zeros, these zeros are not output if PRT (bit1 of parameter 6001) is 1. If PRT is 0, a space code is output each time a zero is encountered.
When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output.
When PRT (bit 1 of parameter 6001) is 0, a space code is output to indicate a positive number instead of +; if PRT is 1, no code is output.

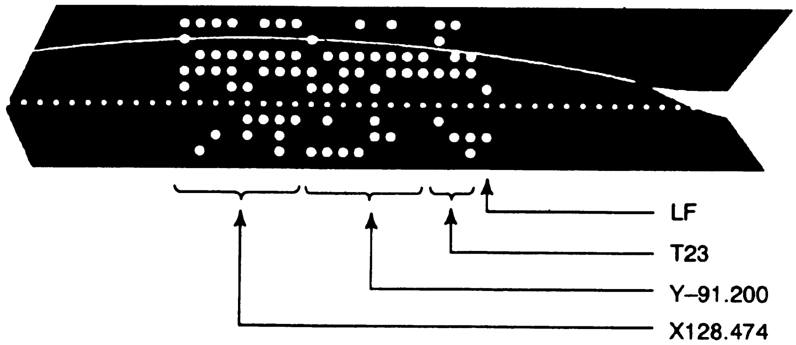
Example)

DPRNT [X#2 [53] Y#5 [53] T#30 [20]]
Variable value
#2=128.47398
#5=–91.2
#30=123.456

(1) Parameter PRT(No.6001#1)=0



(2) Parameter PRT(No.6001#1)=1



- **Close command PCLOS**

PCLOS ;

The PCLOS command releases a connection to an external input/output device. Specify this command when all data output commands have terminated. DC4 control code is output from the CNC.

- **Required setting**

Specify the channel use for data (I/O channel). According to the specification of this data, set data items (such as the baud rate) for the reader/punch interface.

I/O channel 0 : Parameters No. 101, 102, and 103
I/O channel 1 : Parameters No. 111, 112, and 113
I/O channel 2 : Parameters No. 121, 122, and 123

Specify parameter 102, 112 or 122 so that the reader/punch interface is used as the output device for punching. (Never specify output to the Fanuc Cassette or floppy disks.)

When specifying a DPRNT command to output data, specify whether leading zeros are output as spaces (by setting PRT (bit 1 of parameter 6001) to 1 or 0).

To indicate the end of a line of data in ISO code, specify whether to use only an LF (CRO, of bit 4 of parameter 6001 is 0) or an LF and CR (CRO is 1).

Notes

1 It is not necessary to always specify the open command (POPEN), data output command (BPRNT, DPRNT), and close command (PCLOS) together. Once an open command is specified at the beginning of a program, it does not need to be specified again except after a close command was specified.

2 Be sure to specify open commands and close commands in pairs. Specify the close command at the end of the program. However, do not specify a close command if no open command has been specified.

3 When a reset operation is performed while commands are being output by a data output command, output is stopped and subsequent data is erased. Therefore, when a reset operation is performed by a code such as M30 at the end of a program that performs data output, specify a close command at the end of the program so that processing such as M30 is not performed until all data is output.

4 Abbreviated macro words enclosed in brackets [ ] remains unchanged. However, note that when the characters in brackets are divided and input several times, the second and subsequent abbreviations are converted and input.

5 O can be specified in brackets [ ]. Note that when the characters in brackets [ ] are divided and input several times, O is omitted in the second and subsequent inputs.

16 PROGRAMMABLE PARAMETER ENTRY (G10)

General

The values of parameters can be entered in a lprogram. This function is used for setting pitch error compensation data when attachments are changed or the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.

Format

Format

```
G10L50;   Parameter entry mode setting
N_R_;     For parameters other than the axis type
N_P_R_;   For axis type parameters
  :
  :
G11;      Parameter entry mode cancel
```

Meaning of command

N_: Parameter No. (4digids) or compensation position No. for pitch errors compensation +10,000 (5digid)
R_: Parameter setting value (Leading zeros can be omitted.)
P_: Axis No. 1 to 3 (Used for entering axis type parameters)

Explanations

- **Parameter setting value (R_)**

Do not use a decimal point in a value set in a parameter (R_).
a decimal point cannot be used in a custom macro variable for R_ either.

- **Axis No.(P_)**

Specify an axis number (P_) from 1 to 3 (up to three axes) for an axis type parameter. The control axes are numbered in the order in which they are displayed on theCNC display.
For example, specity P2 for the control axis which is displayed second.

Notes
Do not fail to perform reference point return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
Other NC statements cannot be specified while in parameter input mode.
The canned–cycle mode must be cancelled before entering of parameters. When not cancelled, the drilling motion will be activated.

Examples

1. Set bit 2 (SPB) of bit type parameter No. 3404

G10L50 ;	Parameter entry mode
N3404 R 00000100 ;	SBP setting
G11 ;	cancel parameter entry mode

2. Change the values for the Y–axis (2nd axis) and Z–axis (3rd axis) in axis type parameter No. 1322 (the coordinates of stored stroke limit 2 in the positive direction for each axis).

G10L50 ;	Parameter entry mode
N1322P2R4500 ;	Modify Y axis
N1322P3R12000 ;	Modify Z axis
G11 ;	cancel parameter entry mode

III OPERATION

1 GENERAL

1.1 MANUAL OPERATION

Explanations

• Manual reference position return (See Section III–3.1)

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.

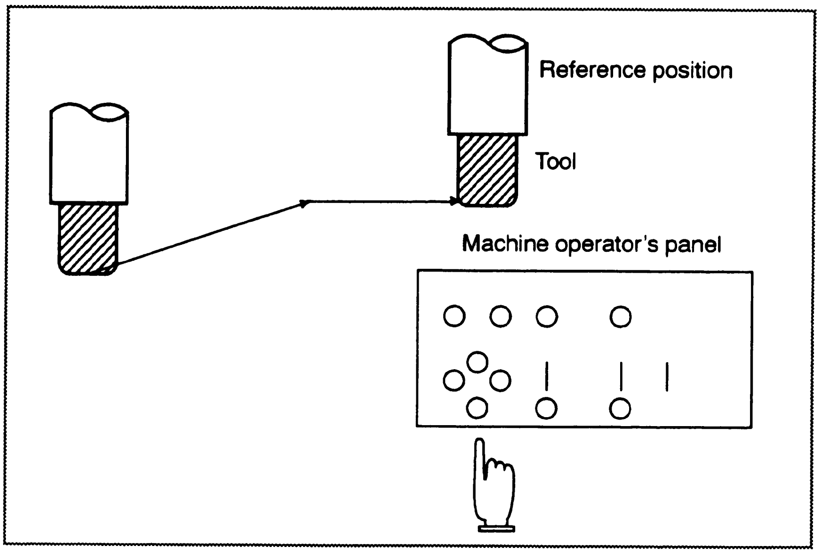


Fig.1.1 (a) Manual reference position return

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.

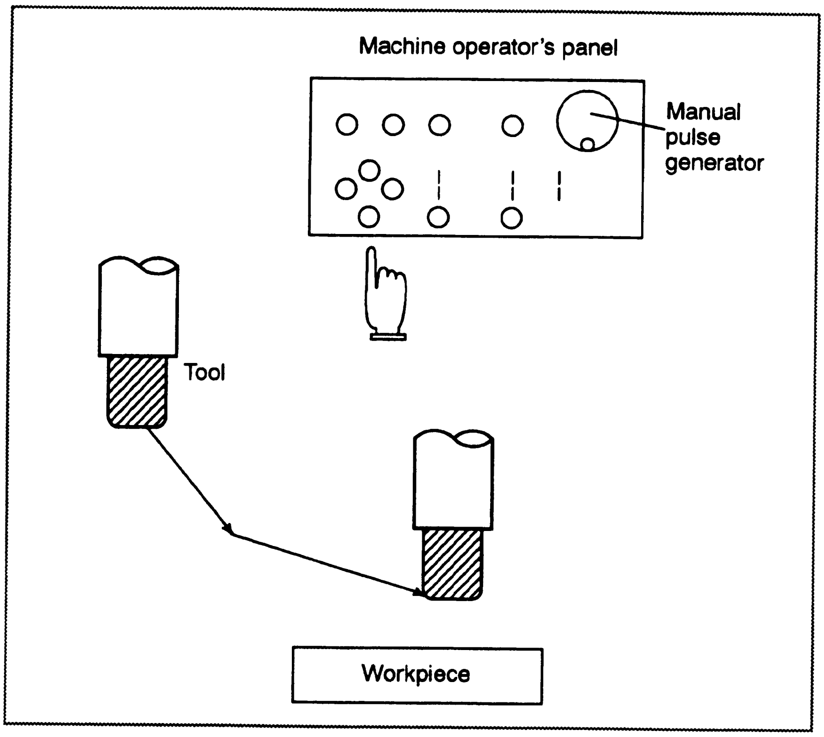


Fig.1.1 (b) The tool movement by manual operation

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.

1.2 TOOL MOVEMENT BY PROGRAMING – AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory, DNC, and MDI operations. (See Section III–4).

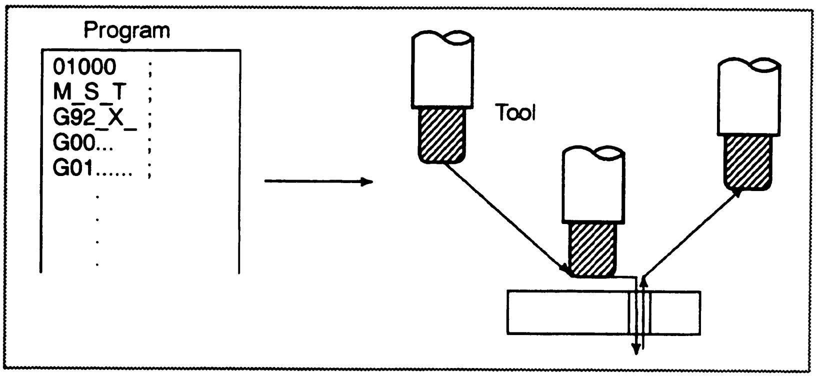


Fig.1.2 (a) Tool Movement by Programming

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.

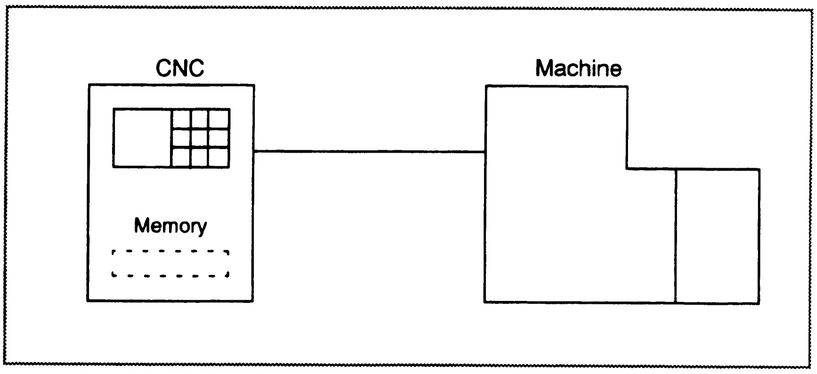


Fig.1.2 (b) Memory Operation

• DNC operation

The machine can operate by reading programs directly from external input/output devices without having to register those programs in CNC memory. This is called DNC 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.

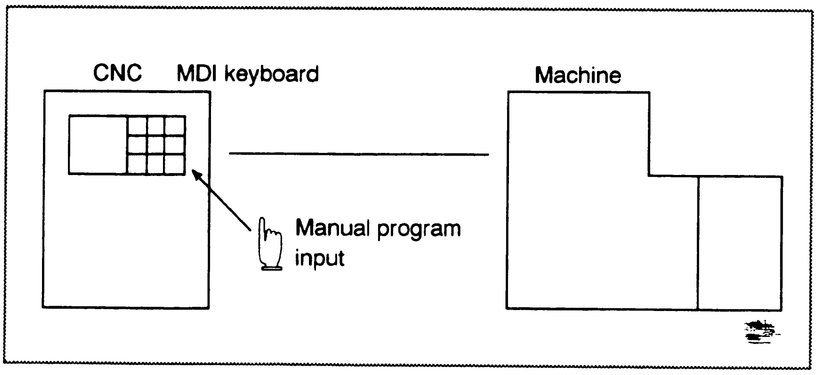


Fig.1.2 (c) MDI operation

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–9.4).

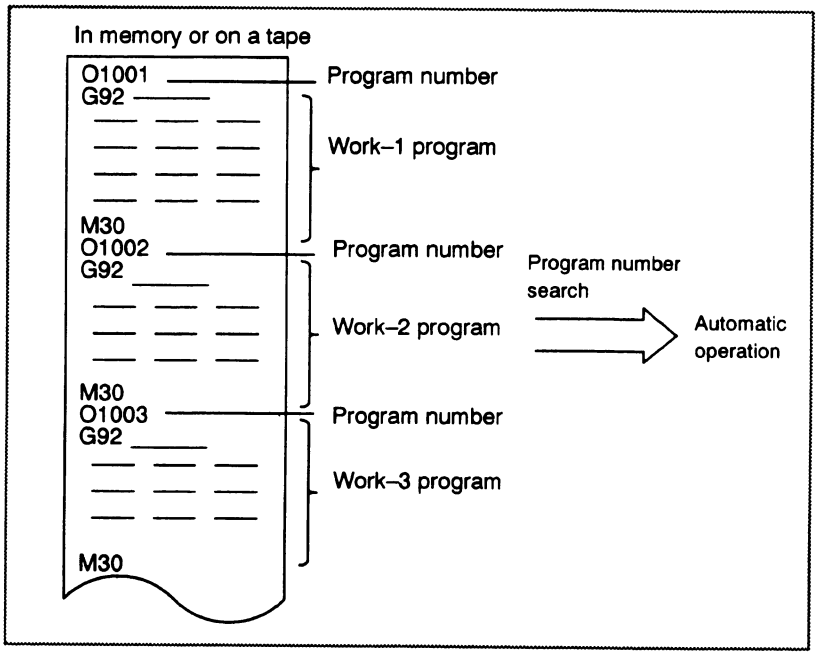


Fig.1.3 (a) Program Selection for Automatic Operation

• Start and stop (See Section III–4)

Pressing the cycle start pushbutton causes automatic operation to start. By pressing the feed hold or reset pushbutton, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops.

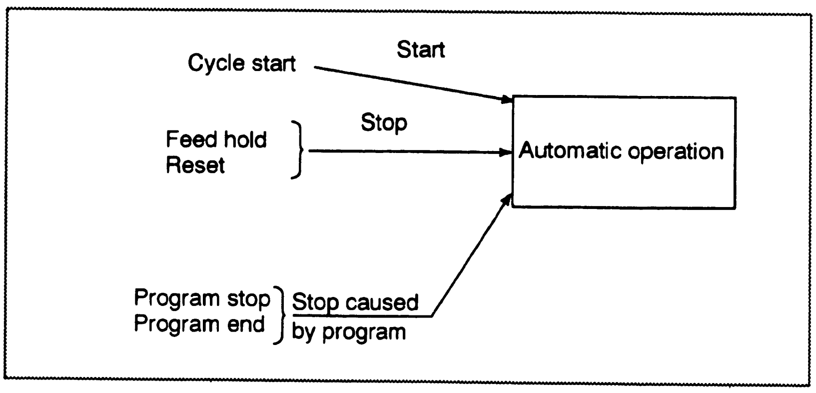


Fig.1.3 (b) Start and Stop for Automatic Operation

- **Handle interruption (See Section III–4.6)**

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle.

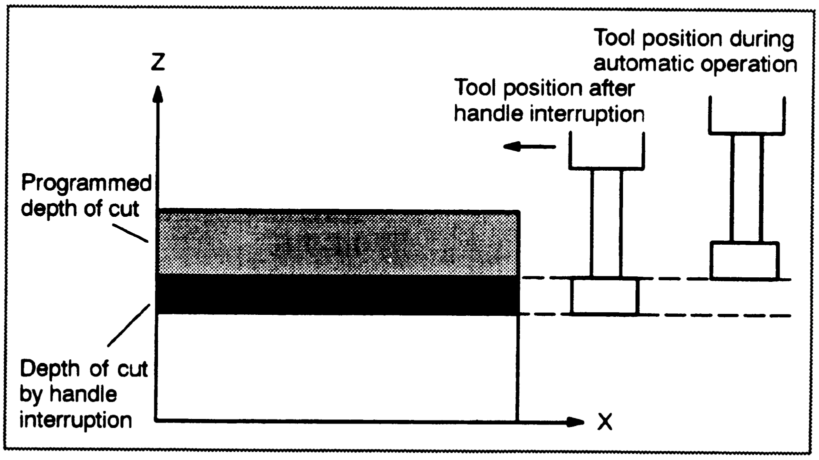


Fig.1.3 (c) Manual Handle Interruption

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 (See Section III–5.4)**

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel.

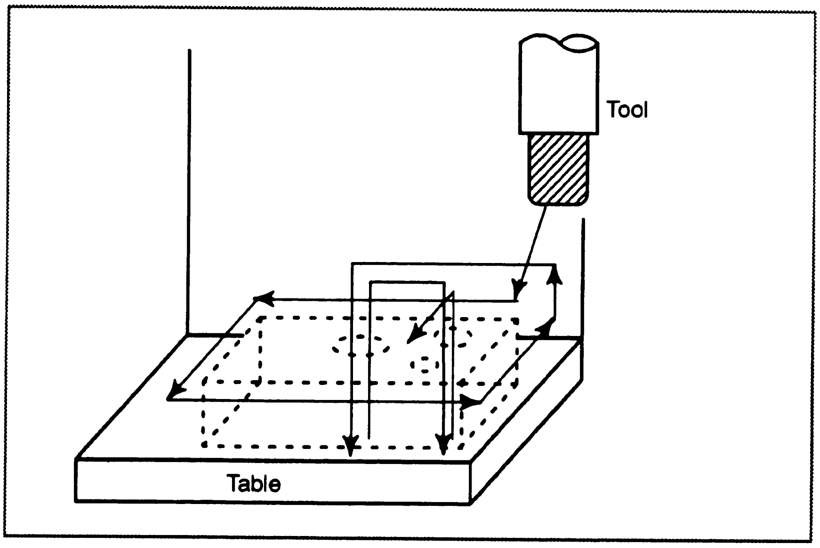


Fig.1.4 (a) Dry run

- **Feedrate override (See Section III–5.2)**

Check the program by changing the rate specified in the program.

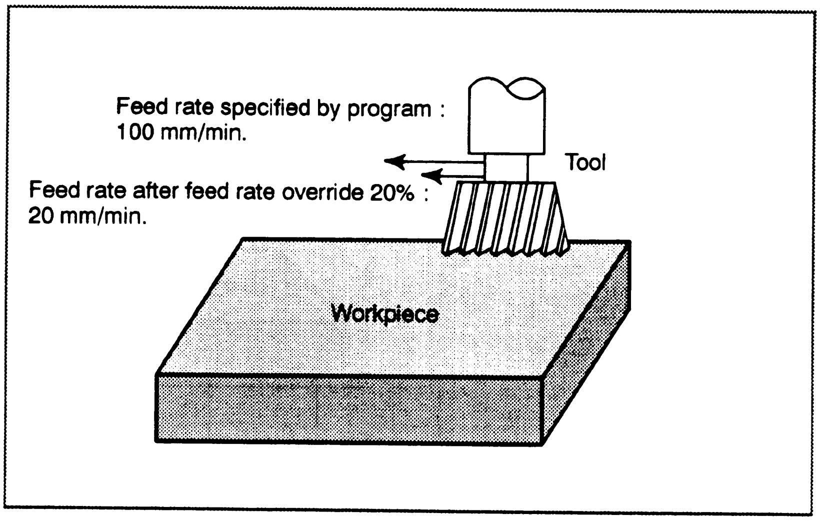


Fig1.4 (b) Feedrate Override

- **Single block (See Section III–5.5)**

When the cycle start pushbutton is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner.

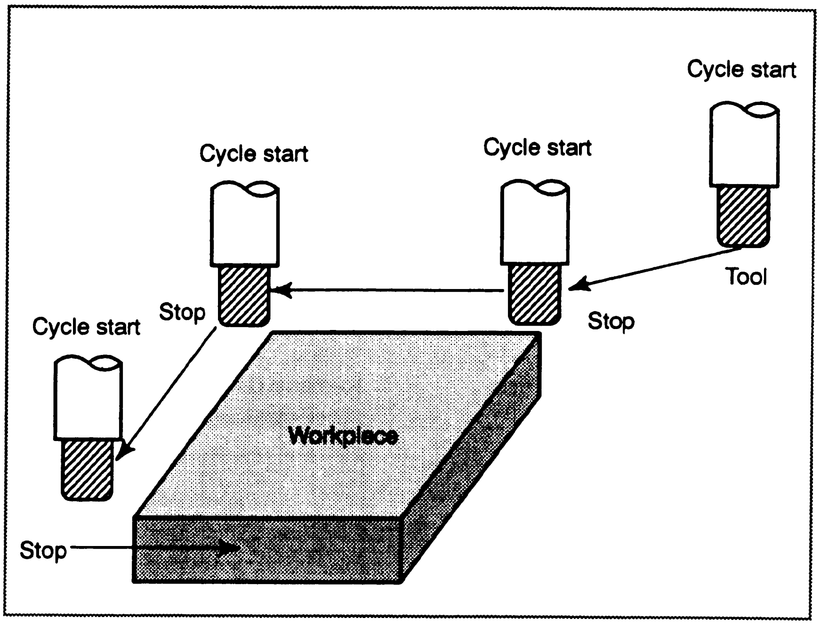


Fig.1.4 (c) Single Block

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

Explanations

- **Machine lock (See Sections III–5.1)**

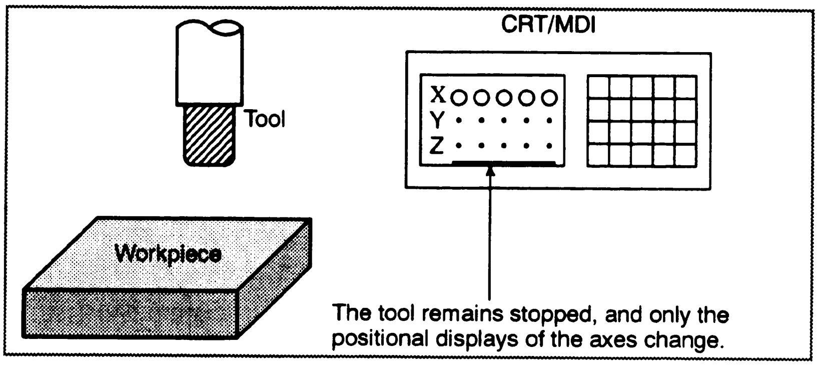


Fig1.4 (d) Machine Lock

- **Auxiliary function lock (See Section III–5.1)**

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

1.5 EDITING A PART PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the CRT/MDI panel (See Section III–9).
This operation can be executed using the part program storage/edit function.

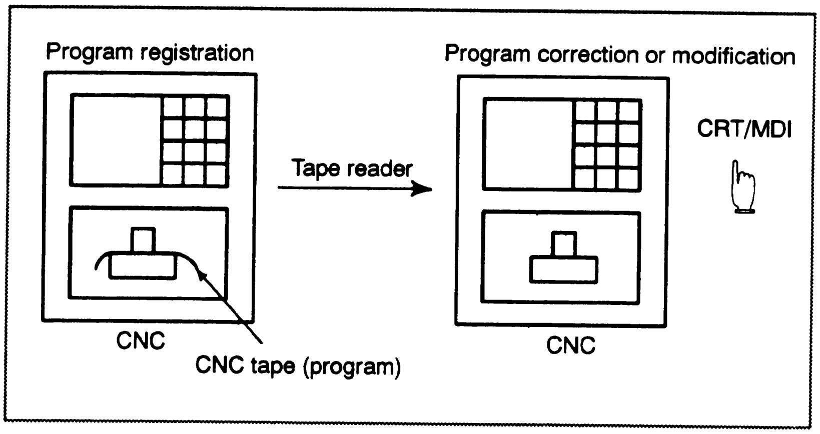


Fig.1.5 (a) Part Program Editing

1.6 DISPLAYING AND SETTING DATA

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

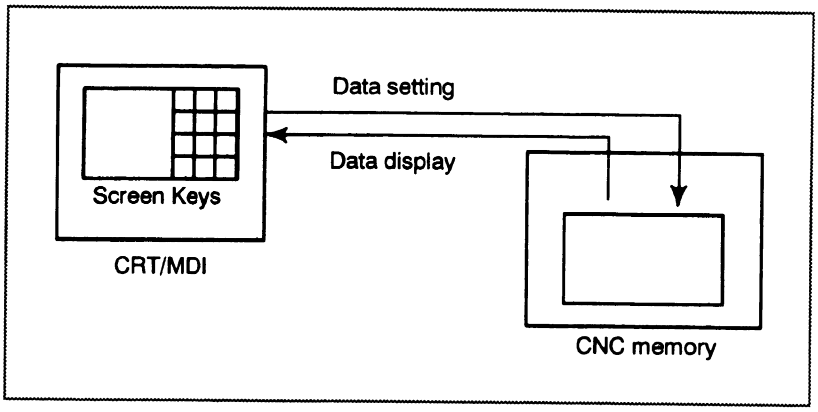


Fig.1.6 (a) Displaying and Setting Data

Explanations

- **Offset value**

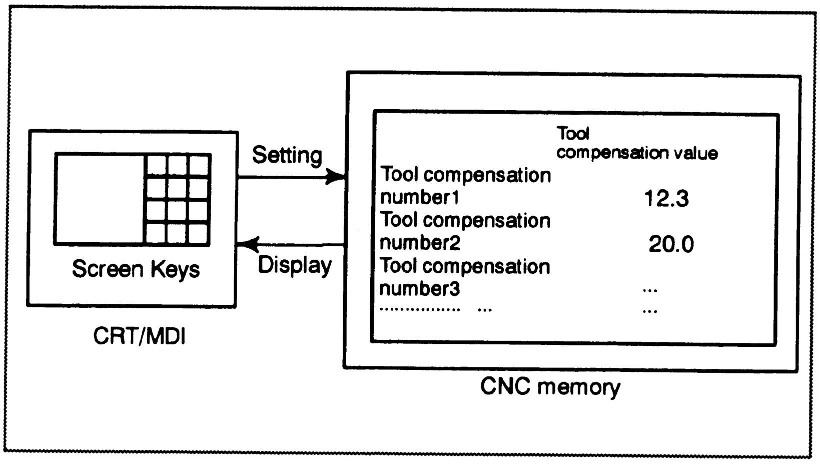


Fig.1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement route 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).

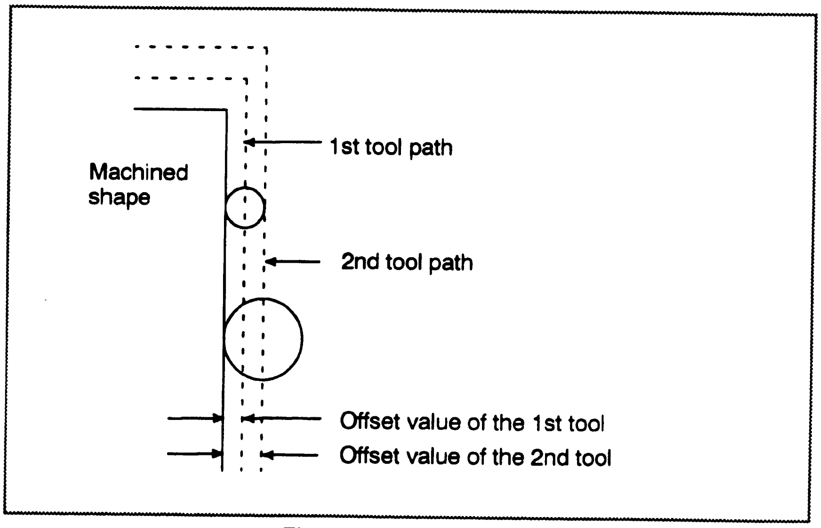


Fig.1.6 (c) Offset Value

- **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
·I/O devices selecting
·Mirror image cutting on/off
The above data is called setting data (See Section III–11.4.2).

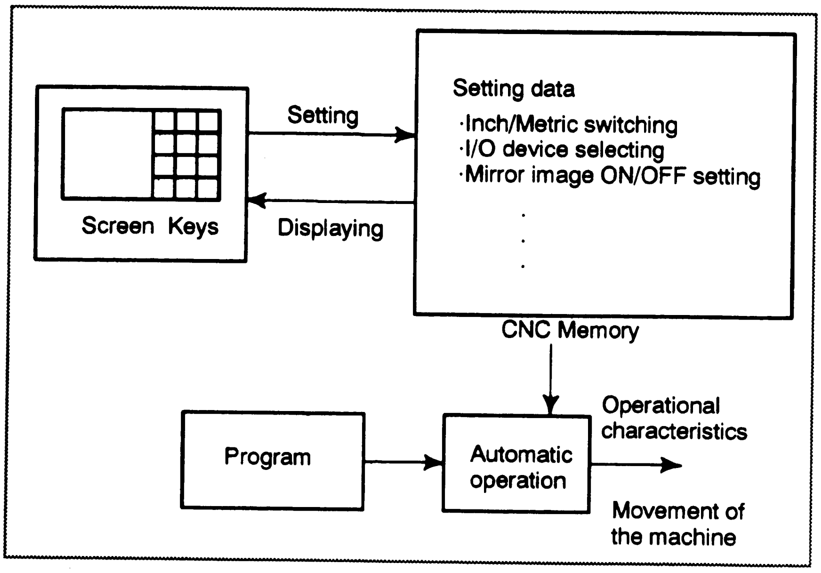


Fig.1.6 (d) Displaying and Setting Operator's setting data

- **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.

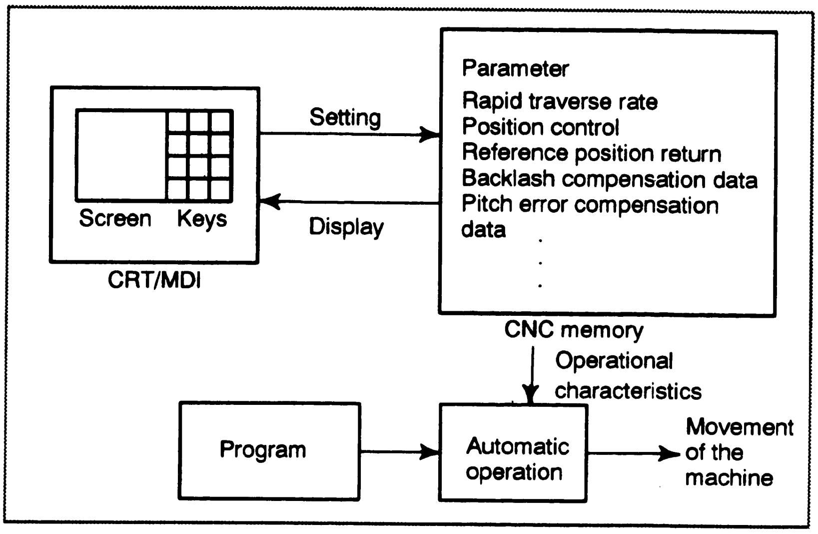


Fig.1.6 (e) Displaying and setting parameters

- **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).

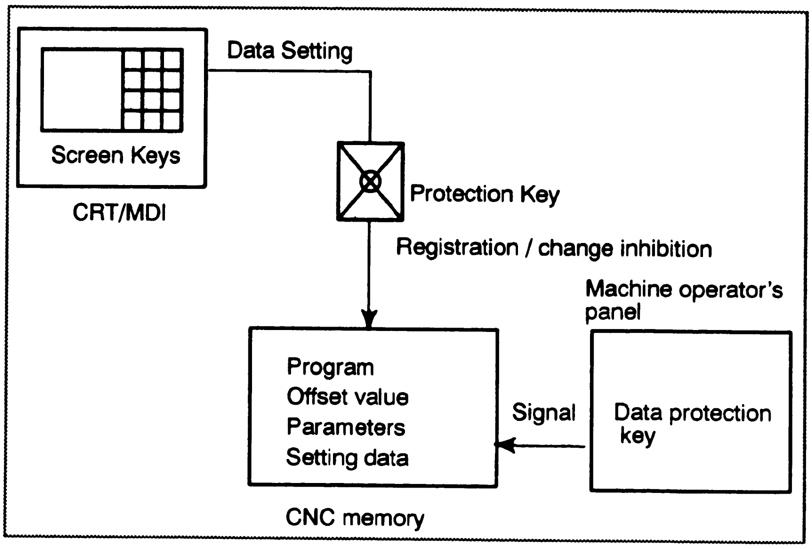
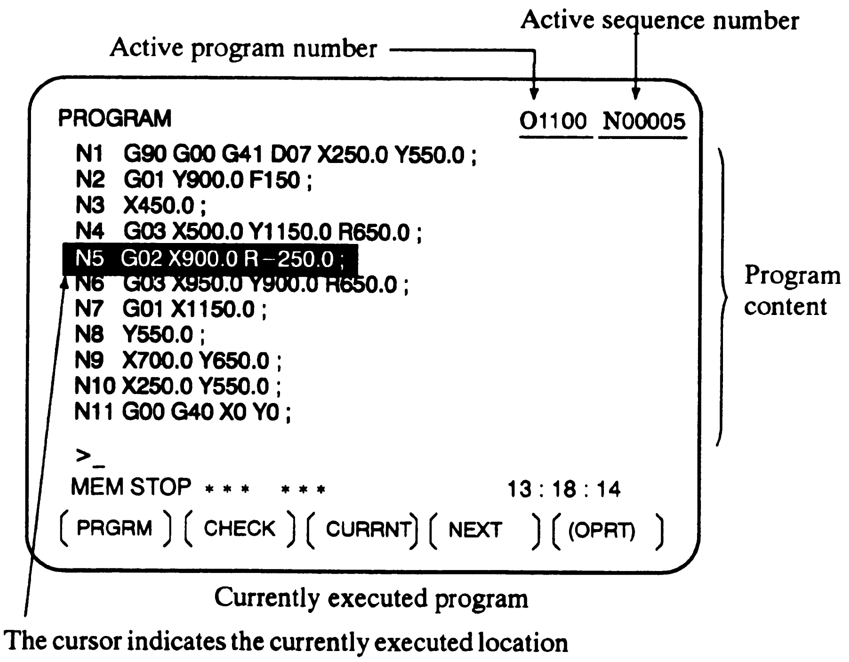


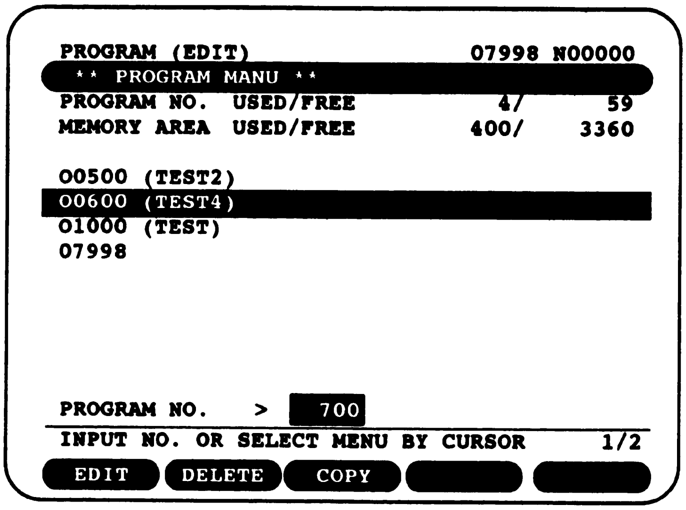
Fig.1.6 (f) Data Protection Key

1.7 DISPLAY

1.7.1 Program Display

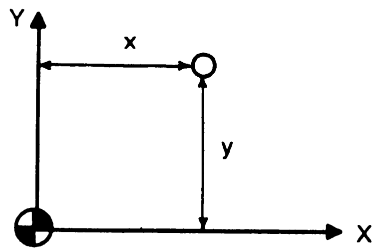
The contents of the currently active program are displayed. In addition, the programs scheduled next and the program list are displayed.
(See Section III–11.2, III–11.3)





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 to 11.1.3)



Workpiece coordinate system

```
ACTUAL POSITION (ABSOLUTE)                    O0003  N00003

   X          150.000
   Y          300.000
   Z          100.000

                              PART COUNT              30
RUN TIME             0H41M    CYCLE TIME      0H  0M22S
MEM  ****  ***  ***                  19 : 47 : 45
( ABS )( REL )( ALL )(       )( (OPRT) )
```

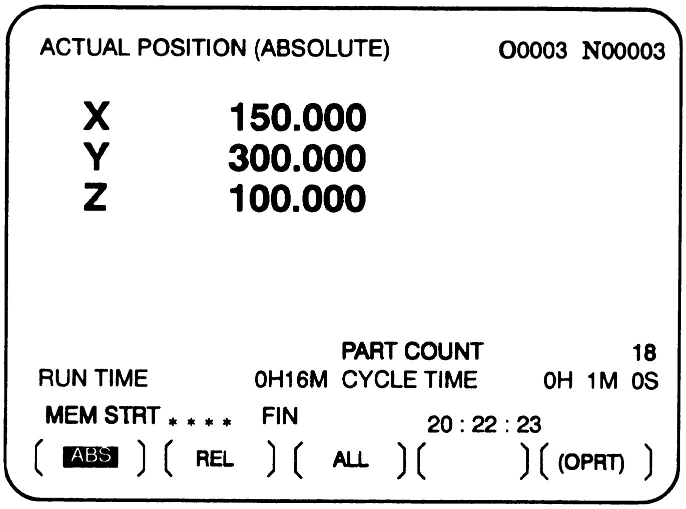
1.7.3 Alarm Display

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

```
ALARM MESSAGE                                 O1000  N00003

010     IMPROPER G–CODE




>_
MEM STOP  ****   *** ALM              19 : 55 : 22
( ALARM )( MSG )( HISTRY )(       )(       )
```

1.7.4 Parts Count Display, Run Time Display

When this option is selected, two types of run time and number of parts are displayed on the screen. (See Section III–11.1.5)



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)

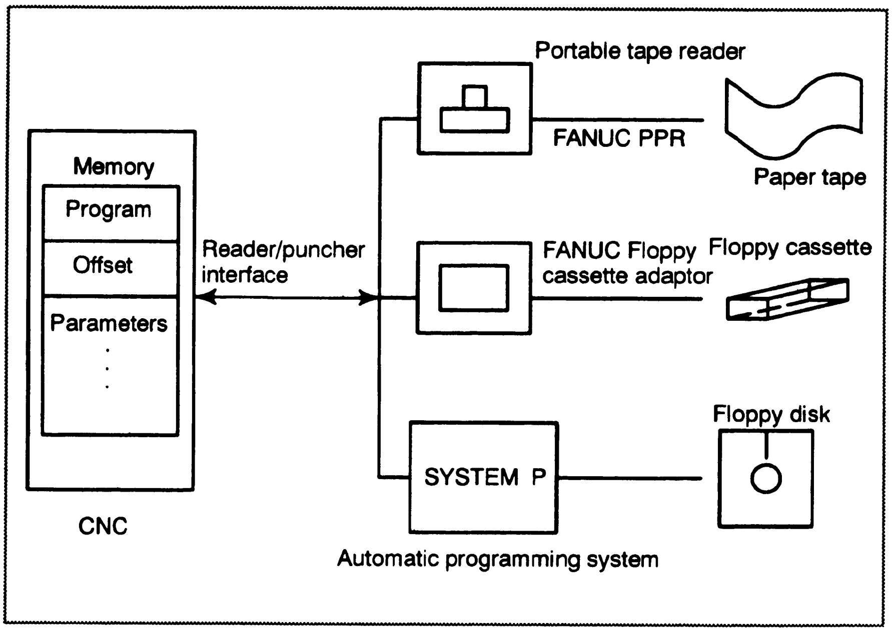


Fig.1.8 (a) Data Input/Output

2 OPERATIONAL DEVICES

The peripheral devices available include the CRT/MDI panel attached to the CNC, machine operator's panel and external input/output devices such as tape reader, PPR, floppy cassette, and FA card.

2.1 CRT/MDI PANELS

Figs. 2.1 (a) show the CRT/MDI panels.

9" small monochrome CRT/MDI panel (horizontal type) . . . Fig.2.1(a)

External view

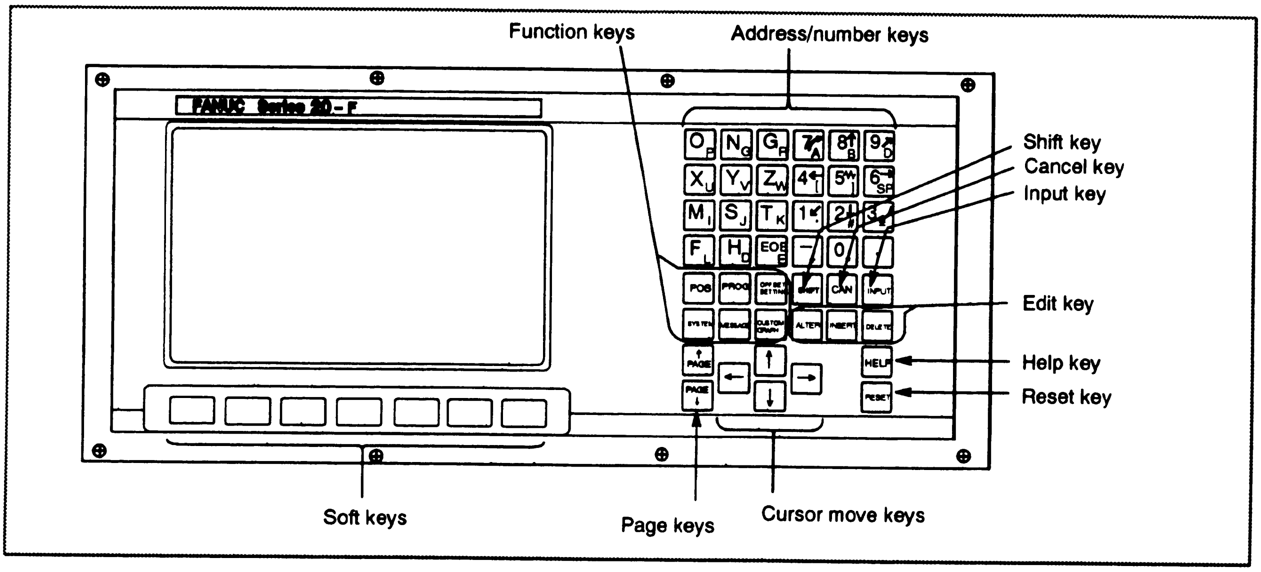


Fig2.1(a) 9" small monochrome CRT/MDI (horizontal type)

> **Note**
> For the series 20, an MDI created by the machine tool builder can be used. The key layout of such an MDI may not be as shown in Fig. 2.1 (a). Refer to the relevant manual supplied by the machine tool builder.

Table2.1 Explanation of the MDI keyboard

Number	Name	Explanation
1	RESET key RESET	Press this key to reset the CNC, to cancel an alarm, etc.
2	HELP key HELP	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 CRT screen.
4	Address and numeric keys N (4← …	Press these keys to input alphabetic, numeric, and other characters.
5	SHIFT key SHIFT	Some keys have two characters on their keytop. Pressing the <SHIFT> key switches the characters. Special character Ê is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered.
6	INPUT key INPUT	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the CRT screen. To copy the data in the key input buffer to the offset register, etc., press the <INPUT> 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 CAN	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 CAN key is pressed, Z is canceled and >N001X100_ is displayed.
8	Program edit keys ALTER INSERT DELETE	Press these keys when editing the program. ALTER : Alteration INSERT : Insertion DELETE : Deletion
9	Function keys POS PROG …	Press theses keys to switch display screens for each function. See sec. 2.2 for detailas of the function keys.

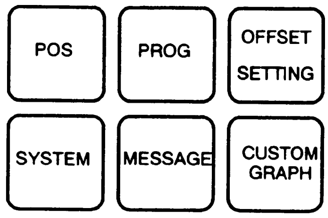
Table2.1 Explanation of the MDI keyboard

Number	Name	Explanation
10	Cursor move keys ← ↑ ↓ →	There are four different cursor move keys. → : 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 ↑ PAGE PAGE ↓	Two kinds of page change keys are described below. PAGE ↓ : This key is used to changeover the page on the CRT screen in the forward direction. ↑ PAGE : This key is used to changeover the page on the CRT screen in the reverse direction.

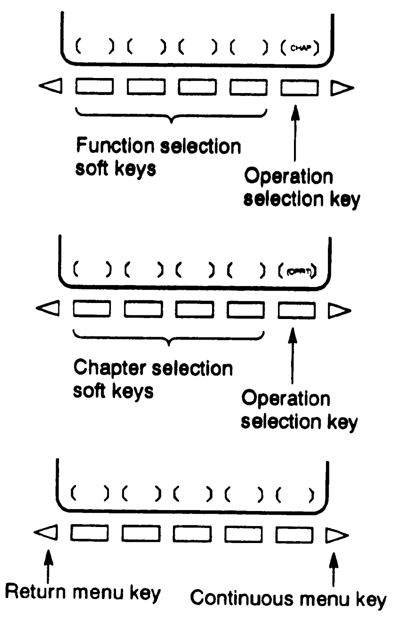
2.2 FUNCTION KEYS AND SOFT KEYS

Function keys are used to select the type of screen (function) to be displayed. After pressing a function key, you can then press a chapter selection soft key to select a subordinate screen (chapter) for that function. You can use the function selection soft keys, instead of the function keys, to select screens (when bit 0 (FSK) of parameter No. 3101 is set to 0).

2.2.1 General Screen Operations



Function keys



1 Press a function key on the CRT/MDI (or LCD/MDI) panel. The chapter selection soft keys that belong to the selected function appear. After pressing a function selection soft key, you can then press the chapter selection key to display the chapter selection soft keys for that function (when bit 0 (FSK) of parameter No. 3101 is set to 0).

2 Press one of the chapter selection soft keys. The screen for the selected chapter appears. If the soft key for a target chapter is not displayed, press the continuous menu key (next–menu key).
In some cases, additional chapters can be selected within a chapter.

3 When the target chapter screen is displayed, press the operation selection key to display data to be manipulated.

4 To redisplay the chapter selection soft keys, press the return menu key.
When bit 0 (FSK) of parameter No. 3101 is set to 0, pressing the return menu displays the function selection soft keys.
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 individual operations.

> **Note**
> This manual describes how to select screens by using the function keys. The selection procedure is the same regardless of whether you use the function keys or the function selection soft keys.

2.2.2 Function Keys

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

POS

Press this key to display the **position screen**.

PROG

Press this key to display the **program screen**.

OFFSET SETTING

Press this key to display the **offset/setting screen**.

SYSTEM

Press this key to display the **system screen**.

MESSAGE

Press this key to display the **message screen.**

CUSTOM GRAPH

Press this key to display the **custom screen.**

Select the custom screen to display or operate the machining guidance function. Refer to the FANUC Series 20–FA MACHINING GUIDANCE FUNCTION OPERATOR'S MANUAL (B–62174E–1) for details of the machining guidance function.

2.2.3 Soft Keys

To display a more detailed screen, press a function key followed by a soft key. Soft keys are also used for actual operations.
The following illustrates how soft key displays are changed by pressing each function key.

The symbols in the following figures mean as shown below :

	: Indicates screens
	: Indicates a screen that can be displayed by pressing a function key(*1)
[ ]	: Indicates a soft key(*2)
( )	: Indicates input from the MDI panel.
	: Indicates the continuous menu key (rightmost soft key).

*1 Press function keys to switch between screens that are used frequently.

*2 Some soft keys are not displayed depending on the option configuration.

POSITION SCREEN

Soft key transition triggered by the function key POS

POS

- **Absolute coordinate display**
  - [ABS] — [(OPRT)]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]
- **Relative coordinate display**
  - [REL] — [(OPRT)]
    - (Axis or numeral) — [PRESET]
    - [ORIGIN]
      - [ALLEXE]
      - (Axis name) — [EXEC]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]
- **Current position display**
  - [ALL] — [(OPRT)]
    - (Axis or numeral) — [PRESET]
    - [ORIGIN]
      - [ALLEXE]
      - (Axis name) — [EXEC]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]
- **Handle interruption**
  - [HNDL] — [(OPRT)]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]

▷

- **Monitor screen**
  - [MONI] — [(OPRT)]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MEM mode

1/2

PROG

Program display screen

[PRGRM] — [(OPRT)] — [BG–EDT] ⇒ **See"When the soft key [BG–EDT] is pressed"**
(1)
- (O number) — [O SRH]
- (N number) — [N SRH]
- [REWIND]
- ▷
  - [P TYPE]
  - [Q TYPE]
- ▷
  - [F SRH] — [CAN] / [EXEC]

Program check display screen

[CHECK] — [ABS] / [REL] — [(OPRT)] — [BG–EDT] ⇒ **See"When the soft key [BG–EDT] is pressed"**
- (O number) — [O SRH]
- (N number) — [N SRH]
- [REWIND]
- ▷
  - [P TYPE]
  - [Q TYPE]
- ▷
  - [F SRH] — [CAN] / [EXEC]

Current block display screen

[CURRNT] — [(OPRT)] — [BG–EDT] ⇒ **See"When the soft key [BG–EDT] is pressed"**

Next block display screen

[NEXT] — [(OPRT)] — [BG–EDT] ⇒ **See"When the soft key [BG–EDT] is pressed"**

▷

Program restart display screen

[RSTR] — [(OPRT)] — [BG–EDT] ⇒ **See"When the soft key [BG–EDT] is pressed"**

(2)(Continued on the next page)

2/2

(2)

[FL.SDL] — [PRGRM] ⇒ Return to (1) (Program display)

File directory display screen

[DIR] — [(OPRT)] — [SELECT] — (File No.) — [F SET]
[EXEC]

Schedule operation display screen

[SCHDUL] — [(OPRT)] — [CLEAR] — [CAN]
[EXEC]
(Schedule data) — [INPUT]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the EDIT mode

PROG

Program menu screen

- (program number) — [EDIT] — (program name) — [NAME]
- (program number) — [DELETE]
- (program number) — [COPY]
- ▷
- (program number) — [PUNCH]
- (program number) — [READ]
- (program number) — [NC–OUT]
- [COMENT]

Guidance programming screen

- [NC LNG]
- [CYCLE]
- [LIST] ⇒ return to Program menu screen
- [2/2] ◄► [1/2]
- ▷
- (program name) — [NAME]
- [GUIDE]

Cycle programming screen

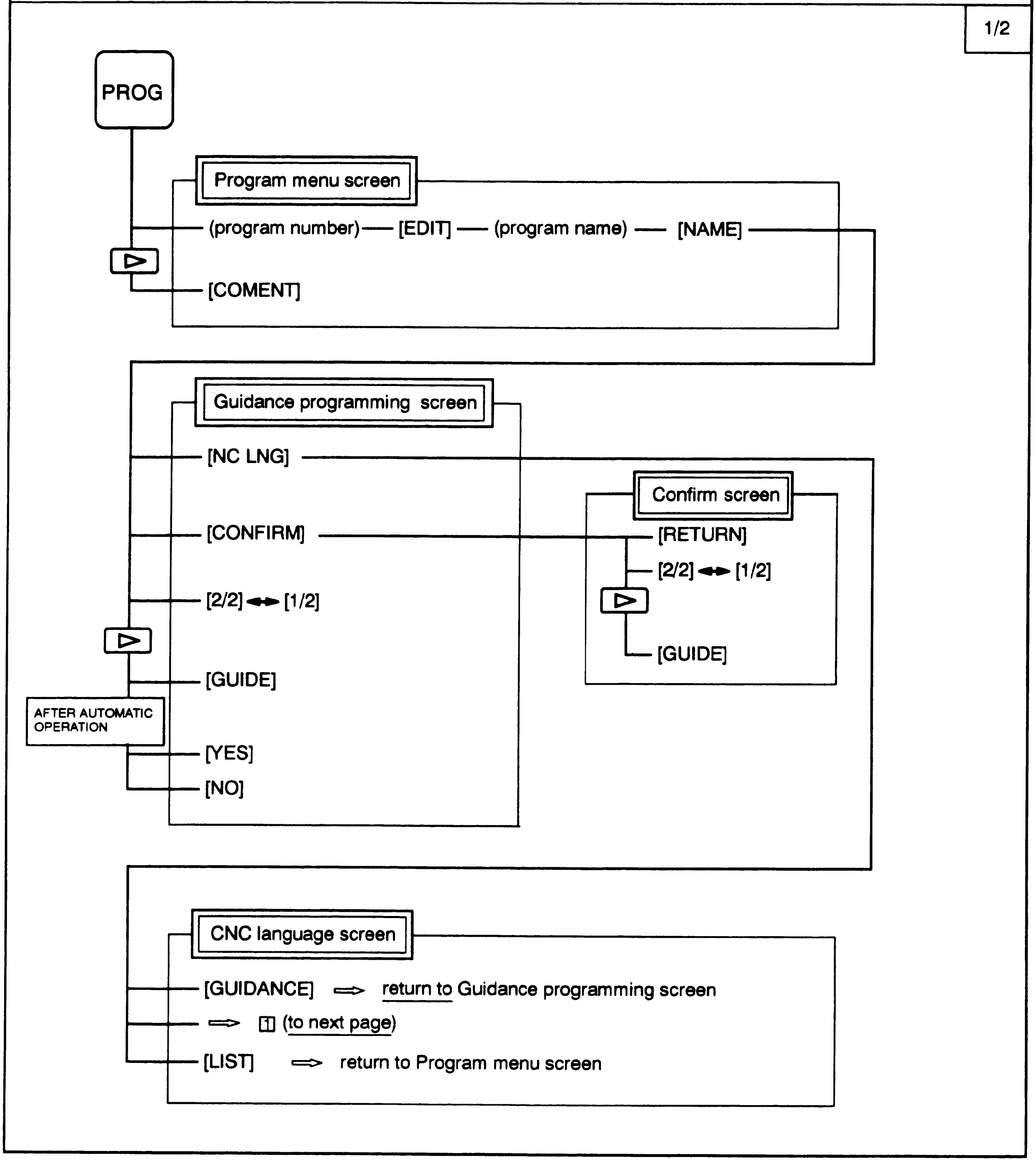
- [GUIDANCE] ⇒ return to Guidance programming screen
- [NC LNG]
- [LIST] ⇒ return to Program menu screen

CNC programming screen

- [GUIDANCE] ⇒ return to Guidance programming screen
- [CYCLE] ⇒ return to CYCLE programming screen
- [LIST] ⇒ return to Program menu screen
- [FLOPPY]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MDI mode



PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MDI mode

2/2

PROG

- **Program display**
  - [PRGRM] ⇒ return to CNC language screen
- **Program input screen**
  - [MDI] — [(OPRT)]
    - [BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed".
    - (Address) — [SRH↓]
    - (Address) — [SRH↑]
    - [REWIND]
- **Current block display screen**
  - [CURRENT]
- **Next block display screen**
  - [NEXT]

▷

- **Program restart display screen**
  - [RSTR]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the HNDL, JOG, or REF mode

PROG

- Program display
  - [PRGRM] —[(OPRT)] —[BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
- Current block display screen
  - [CURRNT] —[(OPRT)] —[BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
- Next block display screen
  - [NEXT] —[(OPRT)] —[BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed"

▷

- Program restart display screen
  - [RSTR] — [(OPRT)] —[BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the TJOG or THDL mode

PROG

- Program display
  - [PRGRM] —[(OPRT)] —[BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed"
- Program input screen
  - [MDI] —[(OPRT)]
    - —[BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed"
    - — (O number) —[O SRH] ⇒ Return to the program
    - — (Address) —[SRH↓]
    - — (Address) —[SRH↑]
    - —[REWIND]
- Program directory display
  - [LIB] —[(OPRT)]
    - —[BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed"
    - — (O number) —[O SRH] ⇒ Return to the program

PROGRAM SCREEN

Soft key transition triggered by the function key PROG
(When the soft key [BG–EDT] is pressed in all modes)

1/2

PROG

- Program menu screen
  - (program number) — [EDIT] — (program name) — [NAME]
  - (program number) — [DELETE]
  - (program number) — [COPY]
  - [BG–END]
  - ▷
  - (program number) — [PUNCH]
  - (program number) — [READ]
  - (program number) — [NC–OUT]
  - [COMENT]

- Guidance programming screen
  - [NC LNG]
  - [CYCLE]
  - [LIST] ⇒ return to Program menu screen
  - [2/2] ◂▸ [1/2]
  - ▷
  - (program name) — [NAME]
  - [GUIDE]

- Cycle programming screen
  - [GUIDANCE] ⇒ return to Guidance programming screen
  - [NC LNG]
  - [LIST] ⇒ return to Program menu screen

- CNC programming screen
  - [GUIDANCE] ⇒ return to Guidance programming screen
  - [CYCLE] ⇒ return to Cycle programming screen
  - [LIST] ⇒ return to Program menu screen
  - [FLOPPY] ⇒ (1)

2/2

(1)

Floppy directory display

```
[FLOPPY] ─┬─[PRGRM] ⇒ Return to the CNC programming screen
          └─[DIR] ─[(OPRT)] ─┬─[F SRH]  ─┬─ (Numeral) ──[F SET]
                             │           ├─ [CAN]
                             │           └─ [EXEC]
                             ├─[READ]   ─┬─ (Numeral) ──[F SET]
                             │           ├─ (Numeral) ──[O SET]
                             │           ├─ [STOP]
                             │           ├─ [CAN]
                             │           └─ [EXEC]
                             ├─[PUNCH]  ─┬─ (Numeral) ──[F SET]
                             │           ├─ (Numeral) ──[O SET]
                             │           ├─ [STOP]
                             │           ├─ [CAN]
                             │           └─ [EXEC]
                             └─[DELETE] ─┬─ (Numeral) ──[F SET]
                                         ├─ [CAN]
                                         └─ [EXEC]
```

OFFSET/SETTING SCREEN

Soft key transition triggered by the function key OFFSET SETTING

OFFSET SETTING

- **Tool offset screen**
  - [OFFSET] — [(OPRT)]
    - (Number) — [NO SRH]
    - (Numeral) — [+INPUT]
    - (Numeral) — [INPUT]
    - ▷
      - [CLEAR] — [ALL]
      - [READ] — [CAN] / [EXEC]
      - [PUNCH] — [CAN] / [EXEC]
- **Setting screen**
  - [SETING] — [(OPRT)]
    - (Number) — [NO SRH]
    - [ON:1]
    - [OFF:0]
    - (Numeral) — [+INPUT]
    - (Numeral) — [INPUT]
- **Work coordinate system setting screen**
  - [WORK] — [(OPRT)]
    - (Number) — [NO SRH]
    - (Axis name) — [MEASUR]
    - (Numeral) — [+INPUT]
    - (Numeral) — [INPUT]
- ▷
- **Macro variables display screen**
  - [MACRO] — [(OPRT)]
    - (Number) — [NO SRH]
    - (Numeral) — [INPUT]
    - ▷
      - [PUNCH]
- **Software operator's panel screen**
  - [OPR]

SYSTEM SCREEN

Soft key transition triggered by the function key SYSTEM

1/3

SYSTEM

Parameter screen

- [PARAM] — [(OPRT)]
  - (Number) — [NO SRH]
  - [ON:1]
  - [OFF:0]
  - (Numeral) — [+INPUT]
  - (Numeral) — [INPUT]
  - ▷
  - [READ] — [CAN] / [EXEC]
  - [PUNCH] — [CAN] / [EXEC]

Diagnosis screen

- [DGNOS] — [(OPRT)] — (Number) — [NO SRH]

PMC screen

- [PMC] — [PMCLAD]
  - [TOP]
  - [BOTTOM]
  - [SRCH]
  - [W–SRCH]
  - [N–SRCH]
  - ▷
  - [F–SRCH]
  - [ADRESS]/[SYMBOL]

(1) (2)

(Continued on the next page)

2/3

(1) (2)

- [PMCDGN]
  - [TITLE]
  - [STATUS]
    - [SEARCH]
  - [ALARM]
  - [TRACE]
    - [T.DISP]/[TRCPRM]
    - [EXEC]
- [PMCPRM]
  - [TIMER]
  - [COUNTR]
  - [KEEPRL]
  - [DATA]
    - [G.DATA]
      - [C.DATA]
      - [G-SRCH]
      - [SEARCH]
    - [G.CONT]
    - [NO.SRH]
    - [INIT]
      - [EXEC]
      - [CANCEL]
- ▷
  - [STOP]/[RUN]
  - [I/O]
    - [EXEC]
    - [CANCEL]
    - (No.)
    - ▷
      - [SPEED]
        - [INPUT]
        - [INIT]

System configuration screen

- [SYSTEM]

▷

(4)

(Continued on the next page)

3/3

(4)

Pitch error compensation screen

```
[PITCH] ── [(OPRT)] ─┬─ (No.) ──[NO SRH]
                     ├─[ON:1]
                     ├─[OFF:0]
                     ├─ (Numeral) ──[+INPUT]
                     ├─ (Numeral) ──[INPUT]
                    [▷]
                     ├─ [READ] ─┬─ [CAN]
                     │          └─ [EXEC]
                     └─ [PUNCH]─┬─ [CAN]
                                └─ [EXEC]
```

Servo parameter screen

```
[SV.PRM] ─┬─ [SV.SET] ─┐
          └─ [SV.TUN] ─┴─[(OPRT)] ─┬─ [ON:1]
                                   ├─ [OFF:0]
                                   └─ (Numeral) ── [INPUT]
```

Spindle parameter screen

```
[SP.PRM] ─┬─[SP.SET]  ┐
          ├─[SP.TUN]  ├─[(OPRT)] ─┬─[ON:1]
          └─[SP.MON]  ┘           ├─[OFF:0]
                                  └─[INPUT]
```

MESSAGE SCREEN Soft key transition triggered by the function key MESSAGE

MESSAGE

- Alarm display screen
  - [ALARM]
- Message display screen
  - [MSG]
- Alarm history screen
  - [HISTRY] —[(OPRT)] —[CLEAR]

HELP SCREEN Soft key transition triggered by the function key HELP

HELP

- Alarm detail screen
  - [1 ALAM] —[(OPRT)] —[SELECT]
- Operation method screen
  - [2 OPR] —[(OPRT)] —[SELECT]
- Parameter table screen
  - [3 PARA]

CUSTOM SCREEN **Soft key transition triggered by the function key CUSTOM**

Select the custom screen to display or operate the machining guidance function.
Refer to the FANUC Series 20–FA MACHINING GUIDANCE FUNCTION OPERATOR'S MANUAL (B–62174E–1) for details of the machining guidance function.
You can also use the custom screen to display an original screen created using the machine tool builder.
For details, refer to the relevant manual supplied by the machine tool builder.
To display another screen (function) from the custom screen, press the corresponding function key.
Simultaneously pressing the left–most and right–most soft keys of the custom screen switches the screen to the position display screen.

2.2.4 Key Input and Input Buffer

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 CRT screen.

In order to indicate that it is key input data, a ">" symbol is displayed immediately in front of it. A "_" is displayed at the end of the key input data indicating the input position of the next character.

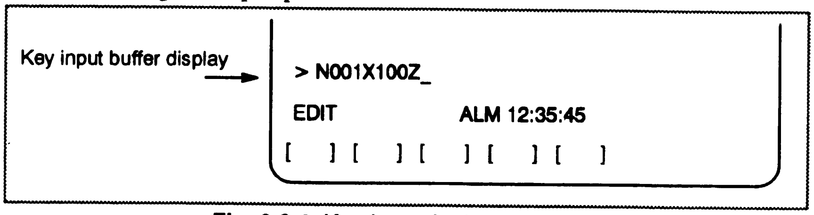


Fig. 2.2.4 Key input buffer display

To input the lower character of the keys that have two characters inscribed on them, first press the SHIFT key and then the key in question.

When the SHIFT key 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 the SHIFT key is pressed in shift status, the shift status is canceled.

It is possible to input up to 32 characters at a time in the key input buffer.

Press the CAN key to cancel a character or symbol input in the key input buffer.

(Example)

When the key input buffer displays
>N001X100Z_
and the cancel CAN key is pressed, Z is canceled and
>N001X100_
is displayed.

2.2.5 Warning Messages

After a character or number has been input from the MDI panel, a data check is executed when INPUT key or a 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.

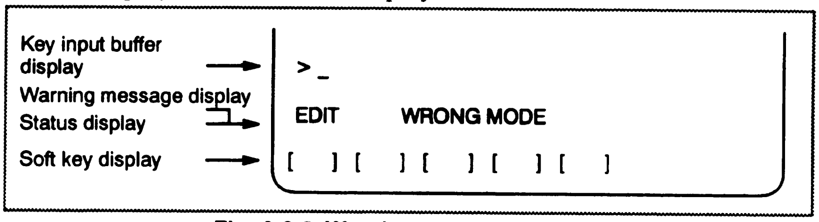


Fig. 2.2.5 Warning message display

Table2.2.5 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 value input exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.
WRONG MODE	Parameter input is not possible in any mode other than MDI mode.
EDIT REJECTED	It is not possible to edit in the current CNC status.

2.2.6 Function selection soft keys

The function selection soft keys are used to select screens, in exactly the same way as function keys. For example, pressing the **[POS]** function selection soft key has the same effect as pressing the POS function key.

Function selection soft keys are displayed when bit 0 (FSK) of parameter No.3101 is set to 0. They are not displayed when the bit is set to 1.

The following function selection soft keys are supported :

```
[ POS ][ PRGRM ][SETTING][SYSTEM][ (CHAP) ]
```

```
[ MSG ][       ][CUSTOM][       ][ (CHAP) ]
```

The table below lists the correspondence between function selection soft keys and function keys.

Function selection soft key	Corresponding function key
[POS]	POS
[PRGRM]	PROG
[SETTING]	OFFSET SETTING
[SYSTEM]	SYSTEM
[MSG]	MESSAGE
[CUSTOM]	CUSTOM GRAPH

Pressing the **[(CHAP)]** soft key displays the chapter selection soft keys.

> **Note**
> This manual describes how to select screen by using function keys. The selection procedure is the same regardless of whether you use the function keys or function selection soft keys.

2.3 EXTERNAL I/O DEVICES

Five 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 2.3(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
Portable Tape Reader	Input device for reading paper tape.	———	Appendix H

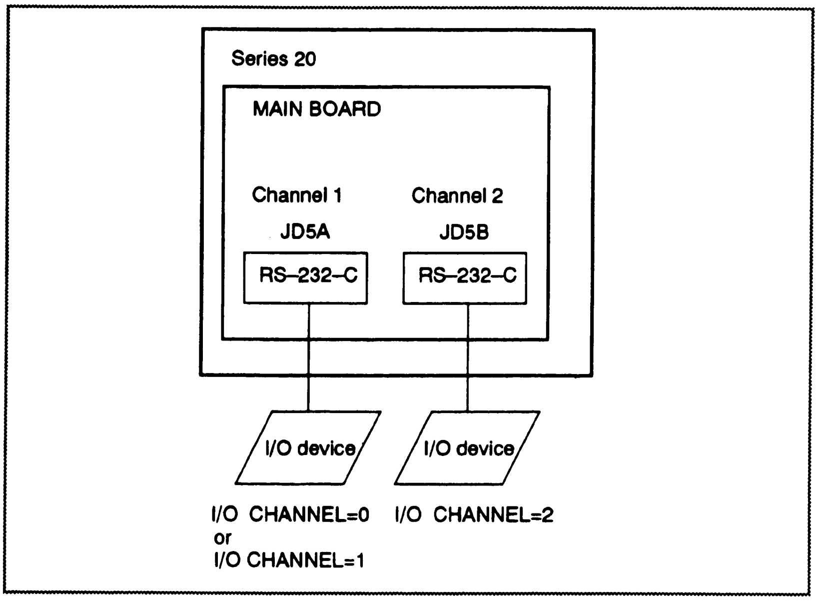
The following data can be input/output to or from external input/output devices:

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

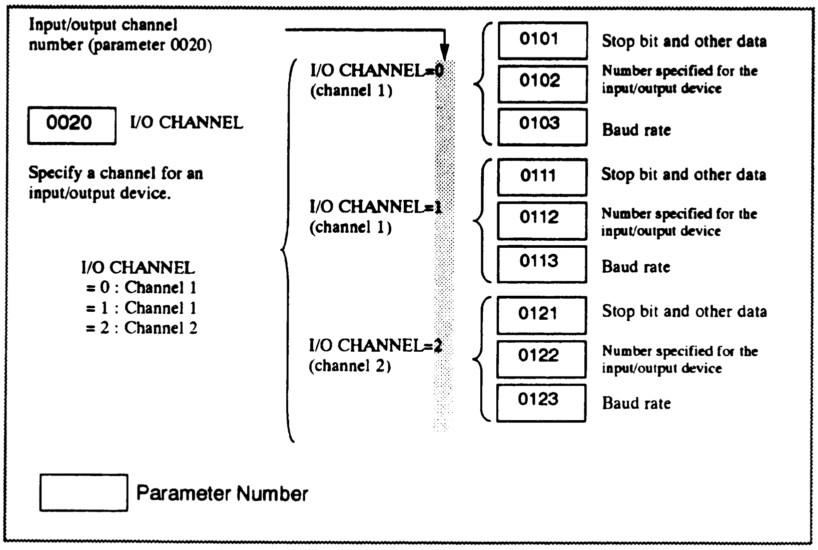
For how data is input and output, see Chapter 8.

Parameter

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



Series 20 has two channels of reader/punch interfaces. The input/output device to be used is specified by setting the channel connected to that device in setting parameter I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance.
For channel 1, two combinations of parameters to specify the input/output device data are provided.
The following shows the interrelation between the reader/punch interface parameters for the channels.

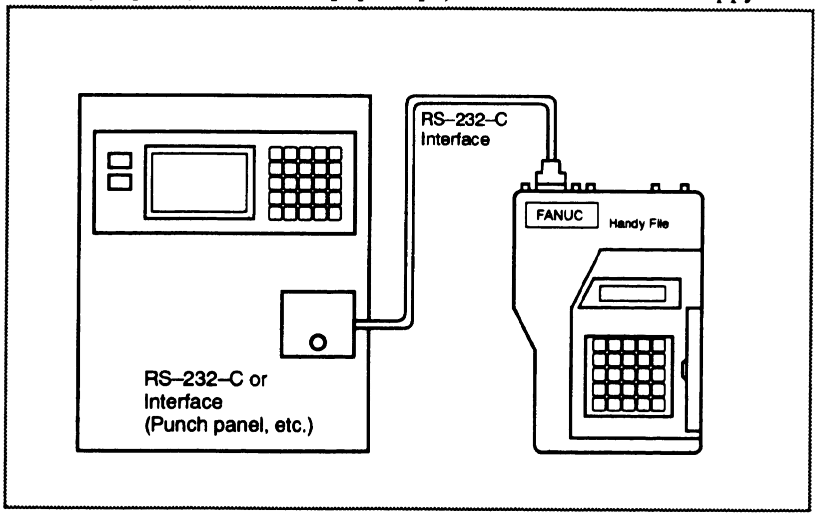


2.3.1 FANUC Handy File

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.

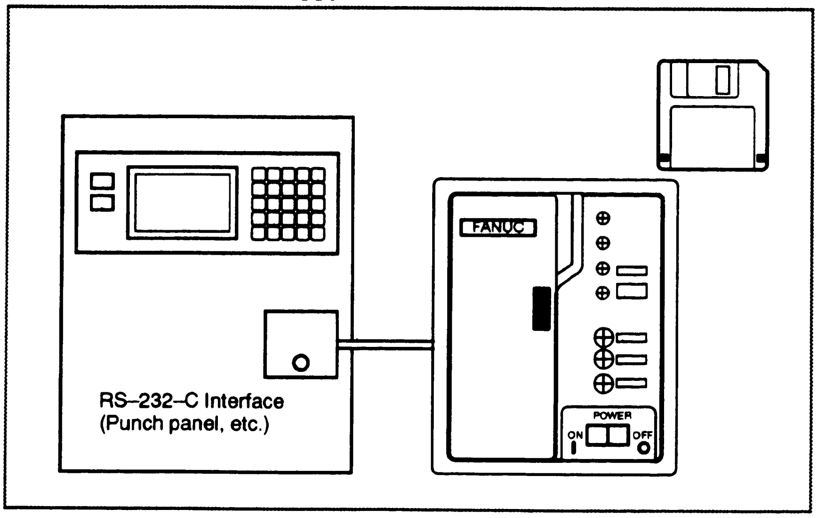
The Handy File uses 3.5–inch floppy disks, which do not have the problems of paper tape (i.e., noisy during input/output, easily broken, and bulky).

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.



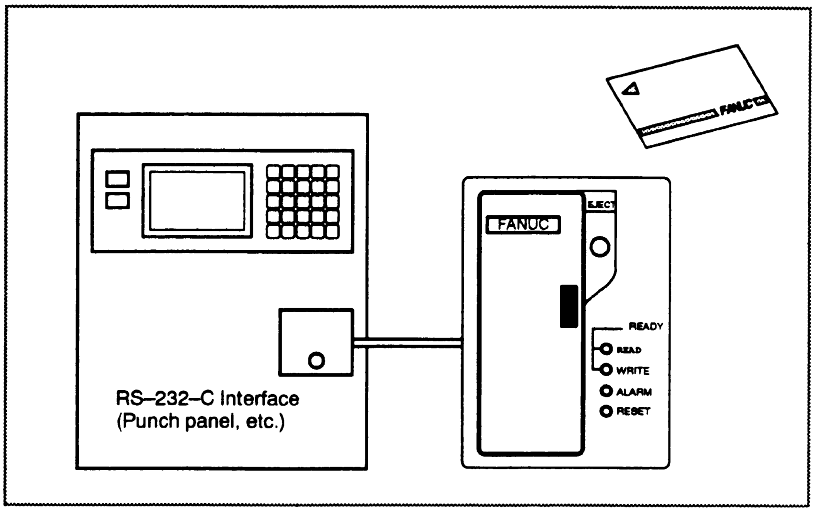
2.3.2 FANUC Floppy Cassette

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.



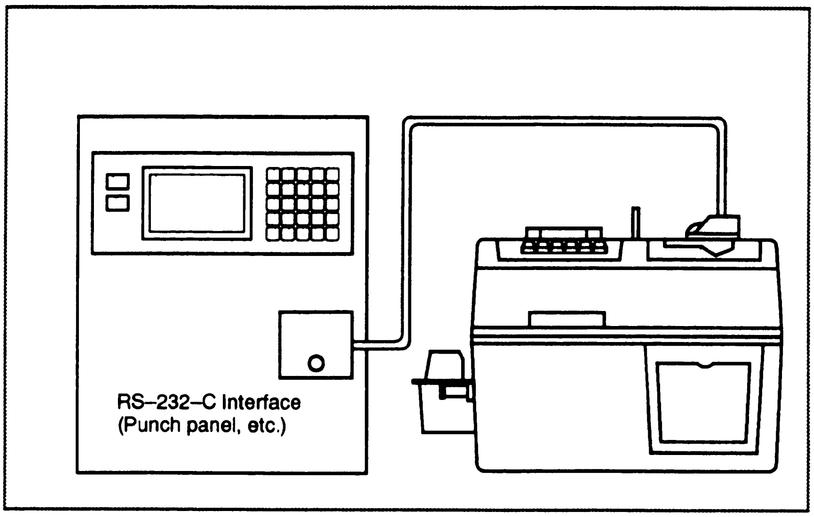
2.3.3 FANUC FA Card

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.



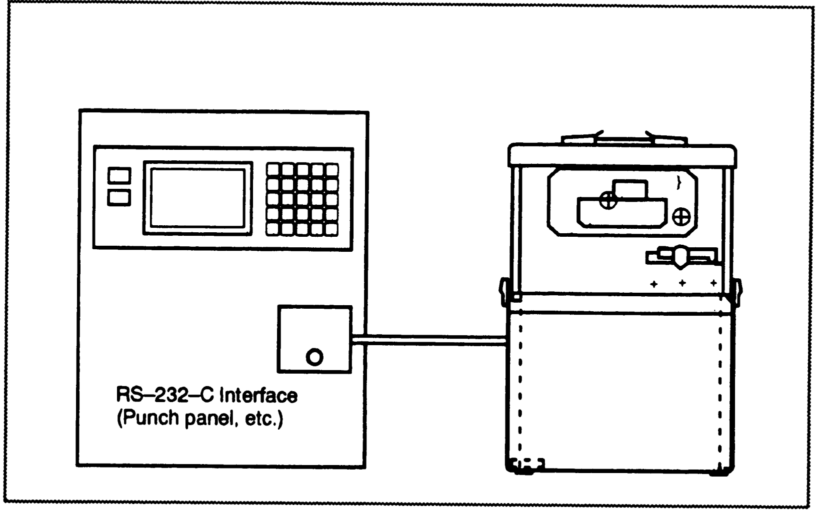
2.3.4 FANUC PPR

The FANUC PPR consists of three units: A printer, paper tape punch, and paper tape reader.
When the PPR is used alone, data can be read from the tape reader and printed or punched out. It is also possible to perform TH and TV checks on data that was read.



2.3.5 Portable Tape Reader

The portable tape reader is used to input data from paper tape.



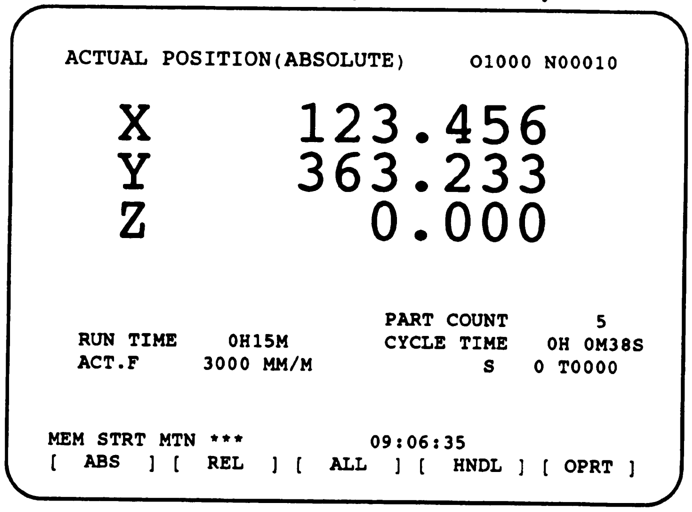
2.4 POWER ON/OFF

2.4.1 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 has occurred at power–on, however, the corresponding alarm screen is displayed. If the screen shown in Section 2.4.2 is displayed, a system failure may have occurred.



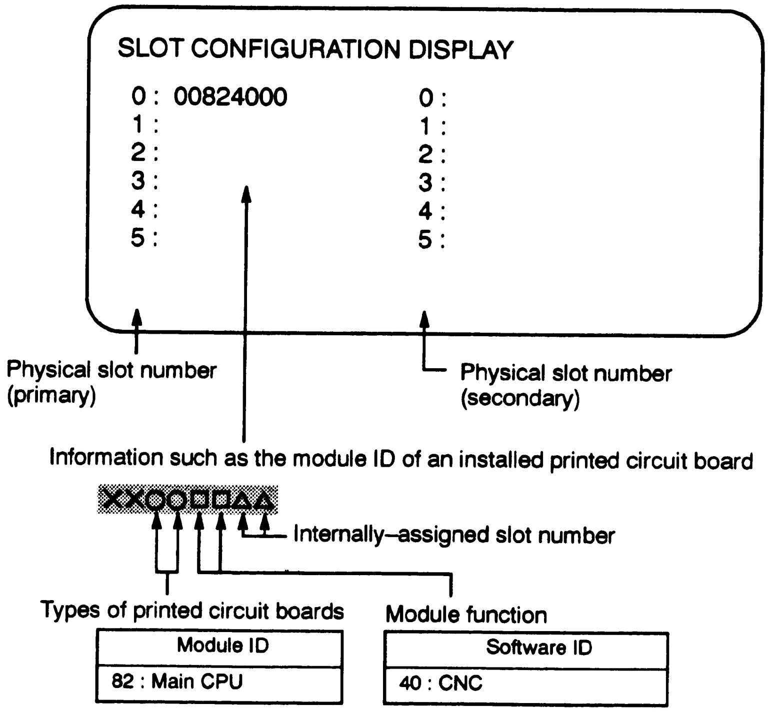
4. Check that the fan motor is rotating.

> **Note**
> When the power is turned on, do not touch CRT/MDI panel keys until the positional or alarm screen is displayed. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

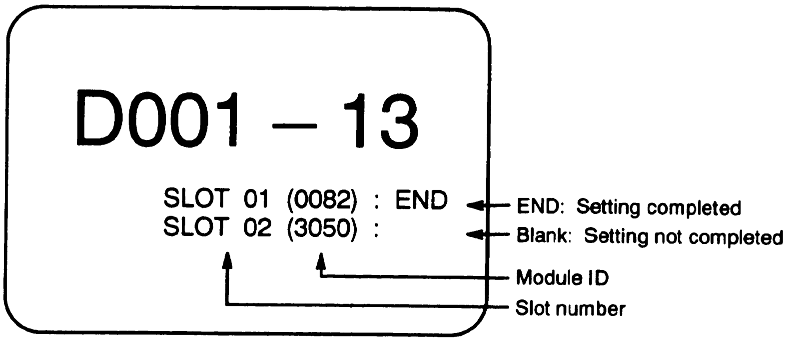
2.4.2 Screen Displayed at Power–on

If a hardware failure or installation error occurs, the system displays one of the following three types of screens then stops.
Information such as the type of printed circuit board installed in each slot is indicated. This information and the LED states are useful for failure recovery.

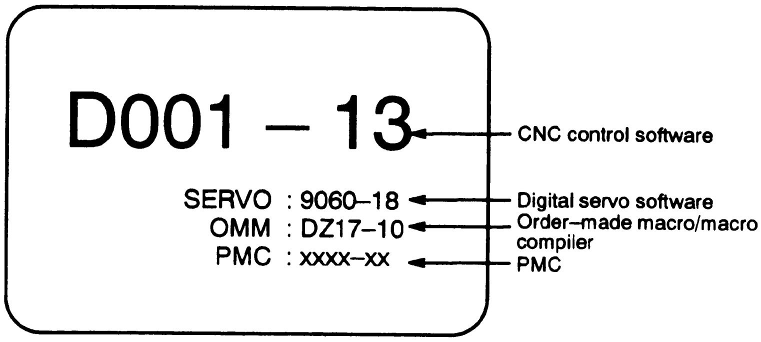
Slot status display



Screen indicating module setting status



Display of software configuration



The software configuration can also be displayed on the system configuration screen.

2.4.3 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.

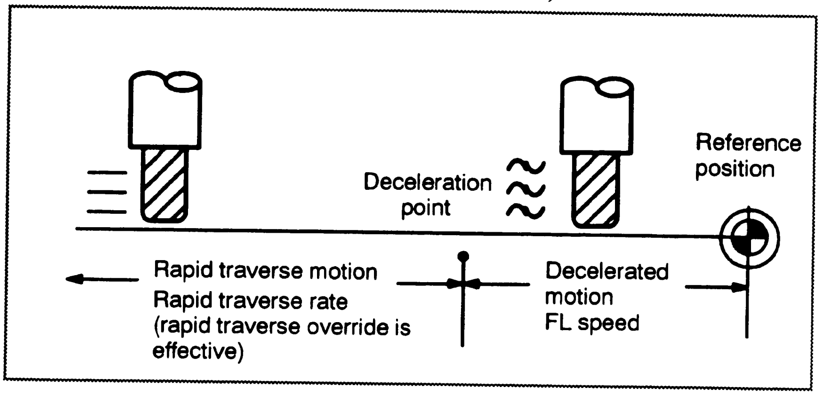
3 MANUAL OPERATION

MANUAL OPERATION are four kinds as follows :

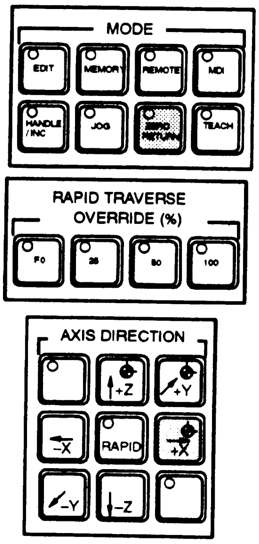
1. **Manual reference position return**
2. **Jog feed**
3. **Incremental feed**
4. **Manual handle feed**
5. **Manual absolute on/off**

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. 1421, 1424 and 1425).
Fourstep 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. The tool generally moves along only a single axis, but can move along three axes simultane ously when specified so in parameter JAX(bit 0 of No.1002).



Procedure for Manual Reference Position Return

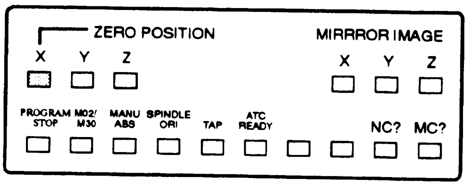


1 Press the reference position return switch, one of the mode selection swithces.

2 To decerease 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 can be moved along three axes simultaneously when specified so in an appropriate parameter setting. 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 machine tool 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.

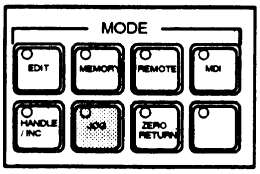
- **Reference position return completion LED**

The REFERENCE POSITION RETURN COMPLETION LED is extinguished by either of the following operations:
– Moving from the reference position.
– Entering an emergency stop state.

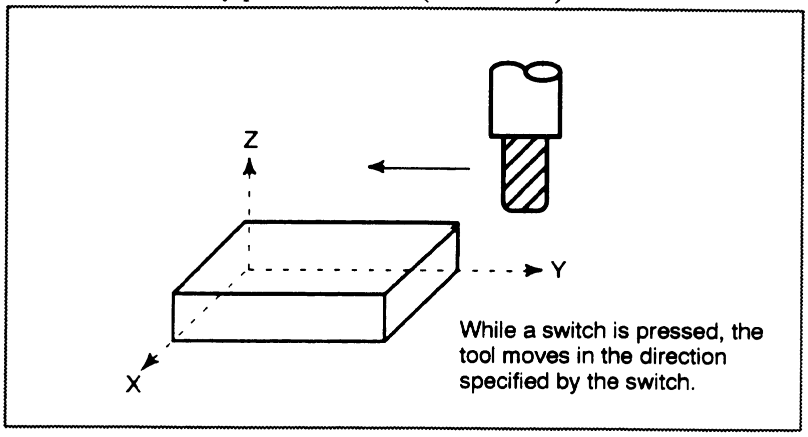
- **The distance to return to reference position**

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.

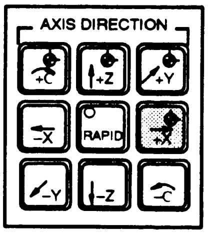
3.2 JOG FEED

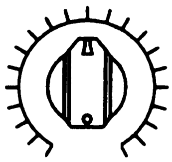


In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.
The jog feedrate is specified in a parameter (No.1423)
The jog feedrate can be adjusted with the jog feedrate override dial.
Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate (parameter No. 1424) regardless of the postiotion of the jog feedrate override dial. This feed is called the manual rapid feed.
Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter JAX (No.1002#0).



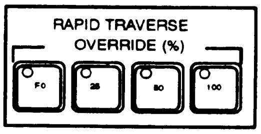
Procedure for Jog Feed





0 2000

JOG FEED RATE OVERRIDE



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**

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

- **Change of modes**

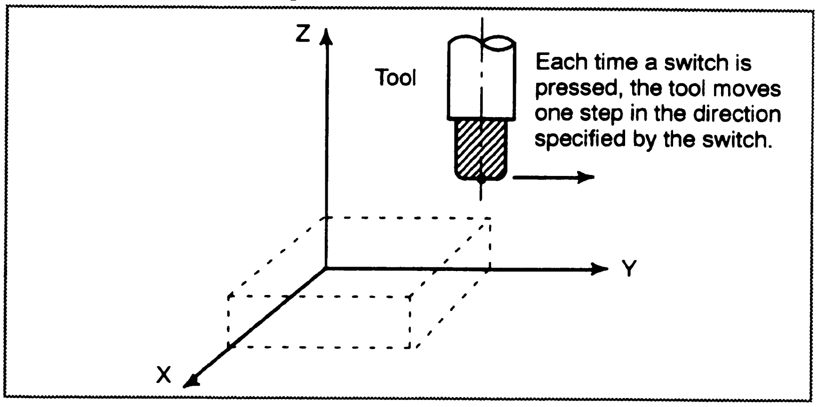
Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

- **Rapid traverse prior to reference position return**

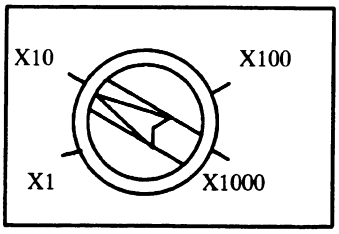
If reference position return is not performed after power–on, pushing RAPID TRAVERSE button does not actuate the rapid traverse but the remains at the JOG feedrate. This function can be disabled by setting parameter RPD (No.1401#01).

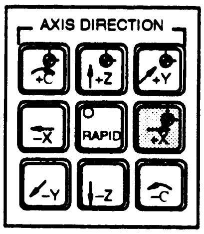
3.3 INCREMENTAL FEED

In the handle/incremental mode, 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. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.



Procedure for Incremental Feed





1 Press the HANDLE/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.

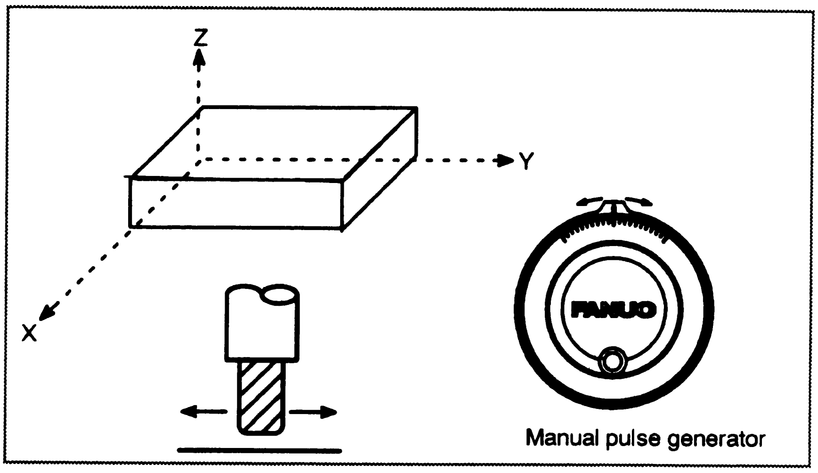
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

- **Handle/incremental feed mode**

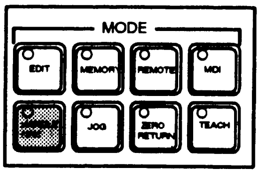
When bit 0 (JHD) of parameter No. 7100 is set to 0, only manual handle feed is enabled in handle/incremental feed mode. When bit 0 (JHD) of parameter No. 7100 is set to 1, manual handle feed and incremental feed are enabled in handle/incremental feed mode. Manual handle feed and incremental feed, however, cannot be enabled at the same time. Manual handle feed is possible only when the tool is not being moved by incremental feed.

3.4 MANUAL HANDLE FEED

In the handle/incremental 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. 7113 and 7114).



Procedur for Manual Handle Feed

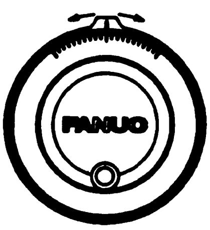


1 Press the HANDLE/INC switch, one of the mode selection switches.

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.



Manual pulse generator

Explanations

- **Availability of manual pulse generator in Jog mode (JHD)**

Parameter JHD (bit 0 of No. 7100) enables or disables the manual handle feed in the JOG mode.
When the parameter JHD(bit 0 of No. 7100) is set 1,both manual handle feed and incremental feed are enabled.

- **Availability of manual pulse generator in TEACH IN JOG mode (THD)**

Parameter THD (bit 1 of No. 7100) enables or disables the manual handle feed in the TEACH IN JOG mode.

- **A command to the MPG exceeding rapid traverse rate (HPF)**

Parameter HPF (No. 7117) specifies as follows:

SET VALUE 0 : The feedrate is clamped at the rapid traverse rate andgenerated pulses exceeding the rapid traverse rate are ignored.(The distance the tool is moved may not match the graduations on the manual pulse generator.)

SET VALUE 1 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC until the limit specified in parameter No. 7117 is reached. (No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

- **Movement direction of an axis to the rotation of MPG (HNGX)**

Parameter HNGx (No. 7102#0) switches the direction in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

Restrictions

- **Number of MPGs**

Up to four manual pulse generators can be connected.

Notes

1. 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.
2. 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.

3.5 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.

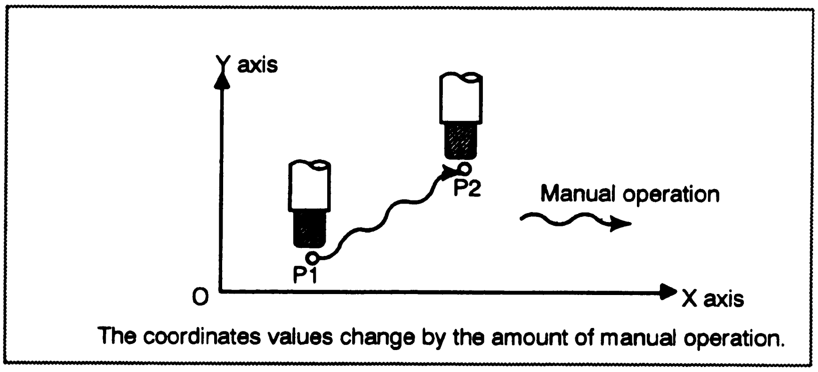


Fig. 3.5(a) Coordinates with the switch ON

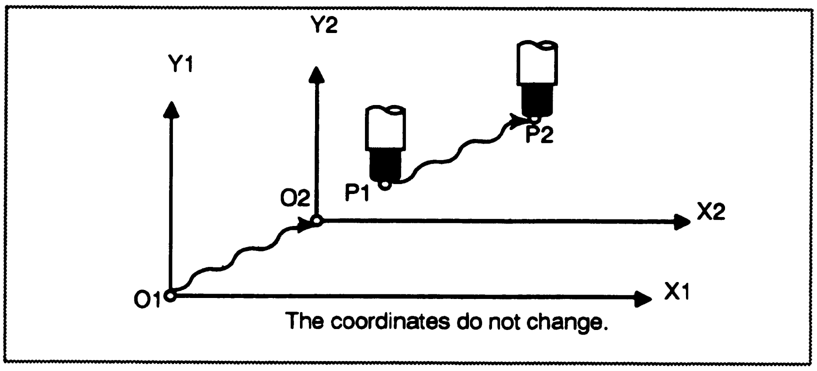


Fig. 3.5(b) Coordinates with the switch OFF

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 X100.0Y100.0F010;
       X200.0Y150.0    ;    [2]
       X300.0Y200.0    ;    [3]
```

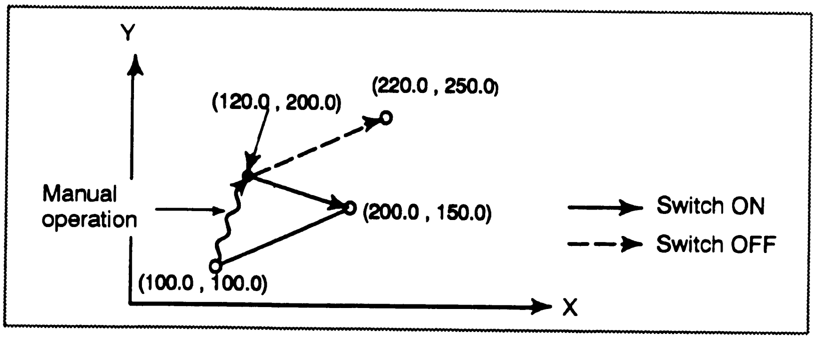
The subsequent figures use the following notation:

⟶ 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.

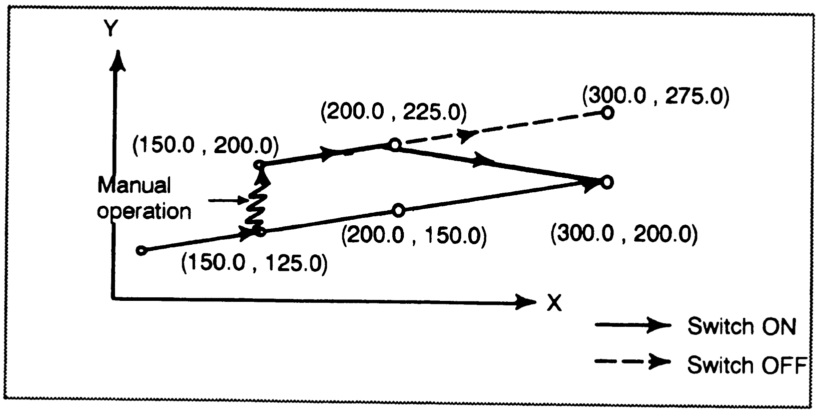
• Manual operation after the end of block

Coordinates when block [2] has been executed after manual operation (X–axis +20.0, Y–axis +100.0) at the end of movement of block .



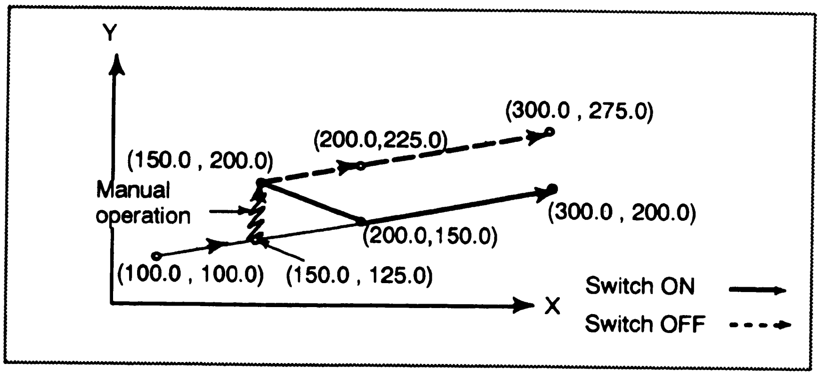
• Manual operation after a feed hold

Coordinates when the feed hold button is pressed while block [2] is being executed, manual operation (Y–axis + 75.0) is performed, and the cycle start button is pressed and released.



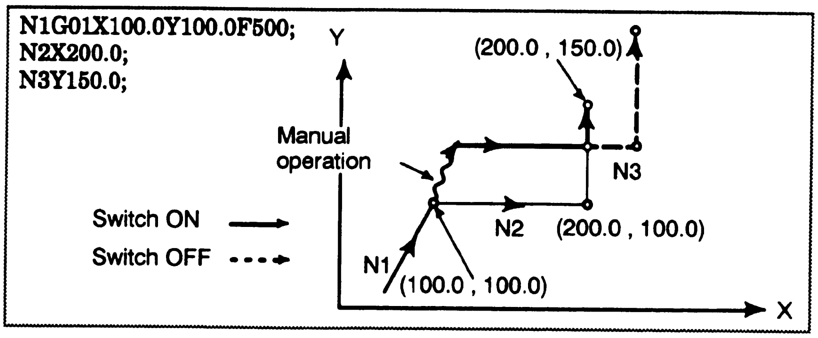
- **When reset after a manual operation following a feed hold**

Coordinates when the feed hold button is pressed while block [2] is being executed, manual operation (Y–axis +75.0) is performed, the control unit is reset with the RESET button, and block [2] 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.



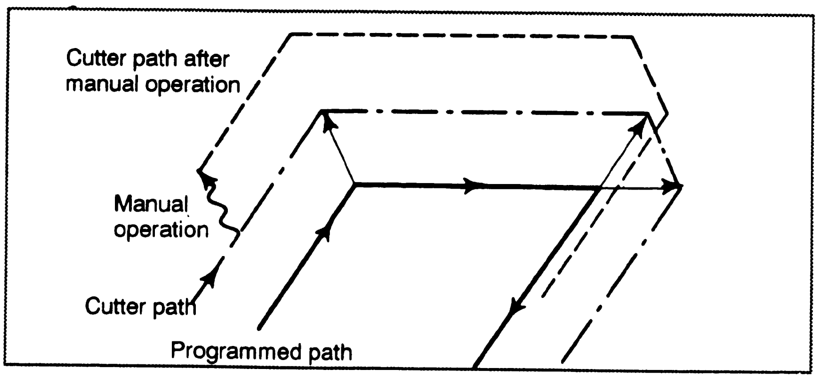
- **When the next move block is an incremental**

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.

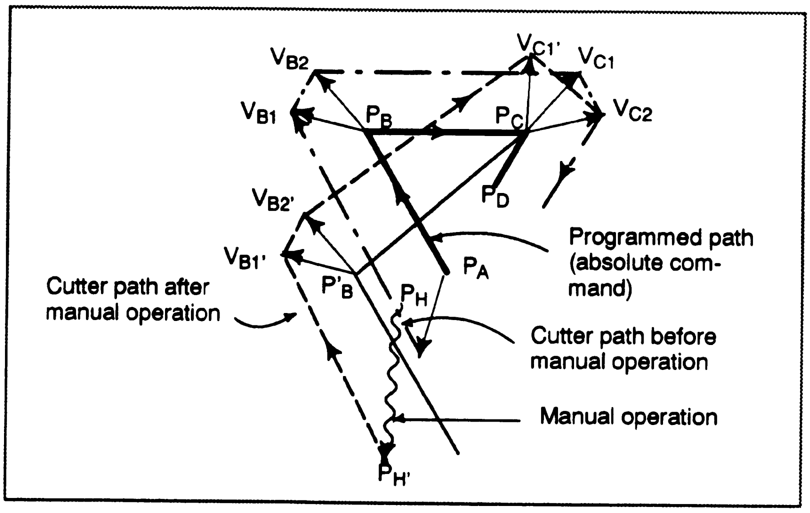


When the switch is ON during cutter compensation

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. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

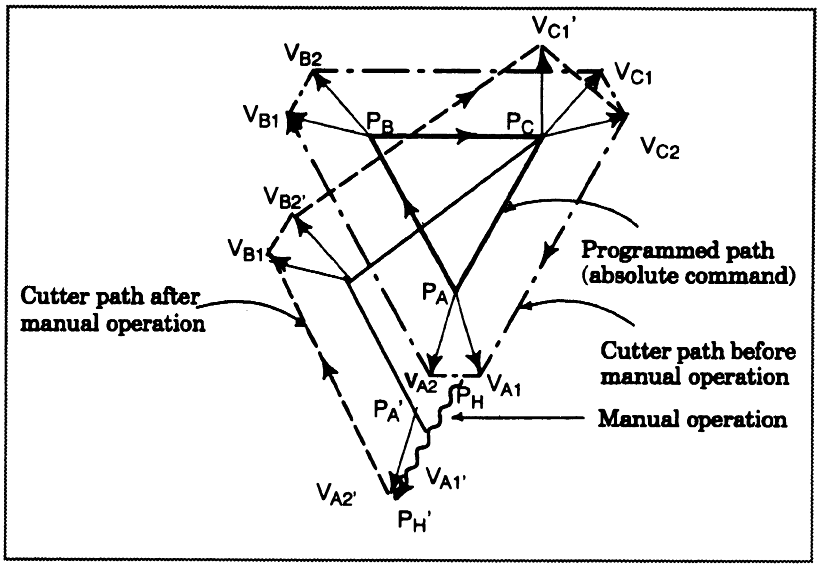
Manual operation performed in other than cornering

Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B, and P_C and that the tool was manually moved to $P_{H'}$. The block end point P_B moves to the point $P_{B'}$ by the amount of manual movement, and vectors V_{B1} and V_{B2} at P_B also move to $V_{B1'}$ and $V_{B2'}$. Vectors V_{C1} and V_{C2} between the next two blocks $P_B - P_C$ and $P_C - P_D$ are discarded and new vectors $V_{C1'}$ and $V_{C2'}$ ($V_{C2'}$ = V_{C2} in this example) are produced from the relation between $P_{B'} - P_C$ and $P_C - P_D$. However, since $V_{B2'}$ is not a newly calculated vector, correct offset is not performed at block $P_{B'} - P_C$. Offset is correctly performed after P_C.



Manual operation during cornering

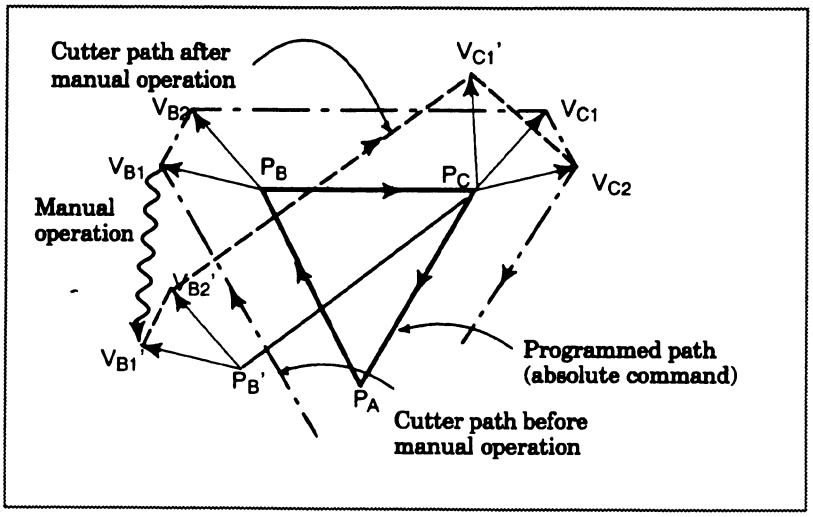
This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$, and $V_{B2'}$ are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct cutter compensation is performed for the blocks following Pc.



Manual operation after single block stop

Manual operation was performed when execution of a block was terminated by single block stop.

Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub–sequent processing is the same as case a described above. An MDI operation can also be interveneted as well as manual operation. The movement is the same as that by manual operation.



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
(GUIDANCE PROGRAMMING MDI OPERATION)
Operation by executing a program entered from the MDI panel

·DNC OPERATION
Function for operating a machine while a program from an external input/output unit.

·PROGRAM RESTART
Restarting a program for automatic operation from an intermediate point

·SCHEDULING FUNCTION
Scheduled operation by executing programs (files) registered in an external input/output device (Handy File, Floppy Cassette, or FA Card)

·SUBPROGRAM CALL FUNCTION
Function for calling and executing subprograms (files) registered in an external input/output device (Handy File, Floppy Cassette, or FA Card) during memory operation

·MANUAL HANDLE INTERRUPTION
Function for performing manual feed during movement executed by automatic operation

·MIRROR IMAGE
Function for enabling mirror–image movement along an axis during automatic operation

·MANUAL INTERVENTION AND RETURN
Function restarting automatic operation by returning the tool to the position where manual intervention was started during automatic operation

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.

When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.

When the reset switch on the CRT/MDI panel is pressed, automatic operation terminates and the reset state is entered.

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

Procedure for Memory Operation

Procedure

1 Select a program to be executed according to either of the following procedure :

1–1 Selecting a program in **EDIT** mode

a Press the **EDIT** mode selection switch.

b Select the program to be executed from the registered programs, as follows :

(i) Press PROG to display the program list screen.

(ii) Position the cursor to the program to be executed by using the ↑ and ↓ cursor keys.

(iii) Instead of (ii), you can also enter the program number directly by using the numeric keys.

(iv) Press the **[EDIT]** soft key.

c Press the MEMORY mode selection switch.

1–2 Selecting a program in **MEMORY** mode

a Press the **MEMORY** mode selection switch.

b Select the program to be executed from the registered programs, as follows:

(i) Press PROG to display the program screen.

(ii) Press the O address key.

(iii) Enter the program number by using the numeric keys.

(iv) Press the **[O SRH]** soft key.

2 Press the cycle start switch on the machine operator's panel. Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.

3 To stop or cancel memory operation midway through, follow the steps below.

a. Stopping memory operation
Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:
(i) When the machine was moving, feed operation decelerates and stops.
(ii) When dwell was being performed, dwell is stopped.
(iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.
When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.

b. Terminating memory operation
Press the RESET key on the CRT/MDI panel.
Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Explanation

Memory operation

After memory 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, memory operation can be executed by repeating the steps (4) to.(6)

Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

– The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).
– There are two keys to stop memory operation: The feed hold key and reset key.

• Program stop (M00)

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

• Optional stop (M01)

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

• Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered.
In some machines, M30 returns control to the top of the program. For details, refer to the manual supplied by the machine tool builder.

• Feed hold

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

• Reset

Automatic operation can be stopped and the system can be made to the reset state by using RESET key on the CRT/MDI panel or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

• Optional block skip

When the optional block skip switch on the machine operator's panel is turned on, blocks containing a slash (/) are ignored.

Calling a subprogram stored in an external input/output device

A file (subprogram) in an external input/output device such as a Floppy Cassette can be called and executed during memory operation. For details, see Section 4.6.

4.2 MDI OPERATION (GUIDANCE PROGRAMMING MDI OPERATION)

When the guidance programming function option is supported, you can create and execute a program by guidance programming in MDI mode. During guidance programming MDI operation, you can add a program that is executed by MDI operation to a program that is already registered in memory.
The following procedure is given as an example. For a full explanation of the actual operation, refer to the relevant manual supplied by the machine tool builder.

Procedure for MDI Operation

Procedure

1 Press the **MDI** mode selection switch.

2 Press the PROG or **[PRGRM]** key to display the program list screen.
Pressing the right–most soft key on the program list screen changes the soft keys. Pressing the **[COMENT]** soft key hides the program names. To display the program names, press the **[COMENT]** soft key again (see Section 10.1.3 for details).

3 In response to the message "INPUT NO. OR SELECT MENU BY CURSOR" on the screen, enter the program number.

a **When creating a new program**
Enter a program number of up to four digits, by using the numeric keys.

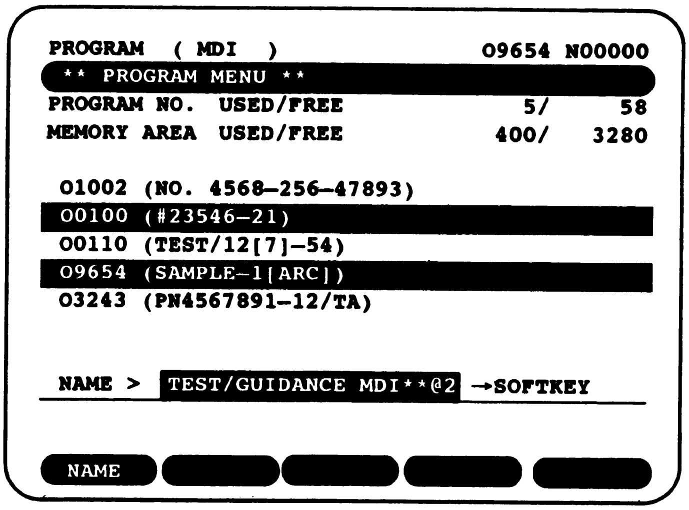
b **When adding an MDI operation program to a registered program**
Enter the program number using the numeric keys, or select the program using the ↑ and ↓ cursor keys.

Example) When a new program number is entered

```
PROGRAM  ( MDI  )                          O9654 N00000
 ** PROGRAM MENU **
PROGRAM NO.   USED/FREE                      5/      58
MEMORY AREA   USED/FREE                    480/    3280

 O1002 (NO. 4568–256–47893)
 O0100 (#23546–21)
 O0110 (TEST/12[7]–54)
 O9654 (SAMPLE–1[ARC])
 O3243 (PN4567891–12/TA)


 PROGRAM NO.    > 235
INPUT NO. OR SELECT MENU BY CURSOR                 1/2
 EDIT
```

4 Press the **[EDIT]** soft key.

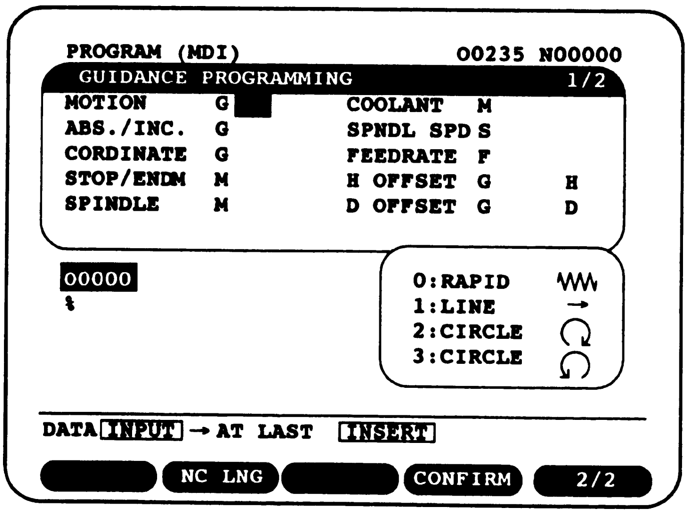
5 The **[NAME]** soft key and prompt to enter the program name appear. Enter a program name of up to 23 characters, using the alphanumeric keys, then press the **[NAME]** soft key. Note that the program name cannot include a semicolon (EOB).



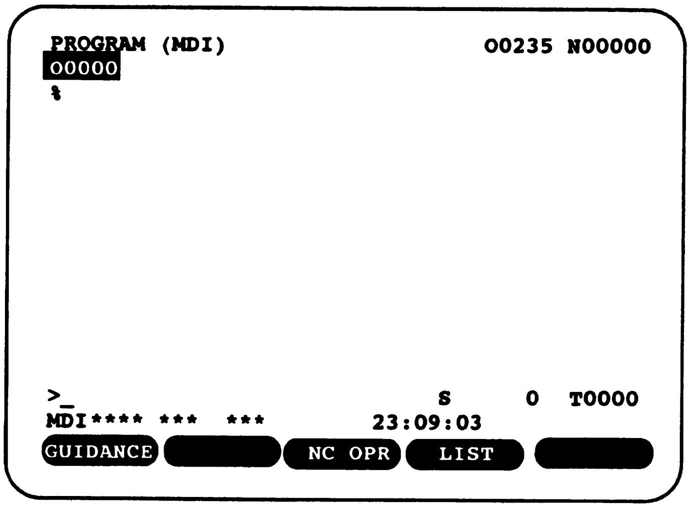
When a program name need not be registered, press the **[NAME]** soft key without entering any characters.
If you enter a previously registered program number in step 3, the prompt to enter a program name does not appear.

6 The guidance programming screen or CNC language screen appears, depending on the following parameter setting:
Bit 0 (NCE) of parameter No. 9320
0 : Displays the guidance programming screen once the program number has been registered.
1 : Displays the CNC language screen once the program number has been registered.

a When the NCE bit is set to 0, the guidance programming screen shown below appears. You can subsequently display the CNC language screen by pressing the **[NC LNG]** soft key.



b When the NCE bit is set to 1, the CNC language screen shown below appears. You can subsequently display the guidance programming screen by pressing the **[GUIDANCE]** soft key.



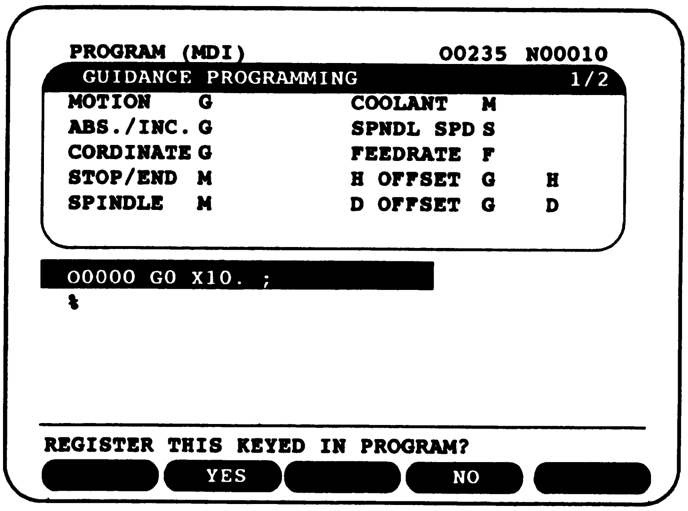
MDI operation program O0000 is displayed on the screen. The program number entered in step 3 is displayed at the top right of the screen. To return to the program list screen, press the **[LIST]** soft key.

7 Create a program by performing guidance programming. See Section 10.1.4 for details.
The guidance programming MDI operation allows up to nine blocks to be created at one time. When registering a program, created during guidance programming MDI operation, into memory, do not use M02 or M30 at the end of the program.

8 To execute the program displayed on the guidance programming screen, press the cycle start button on the machine operator's panel. The program is started.

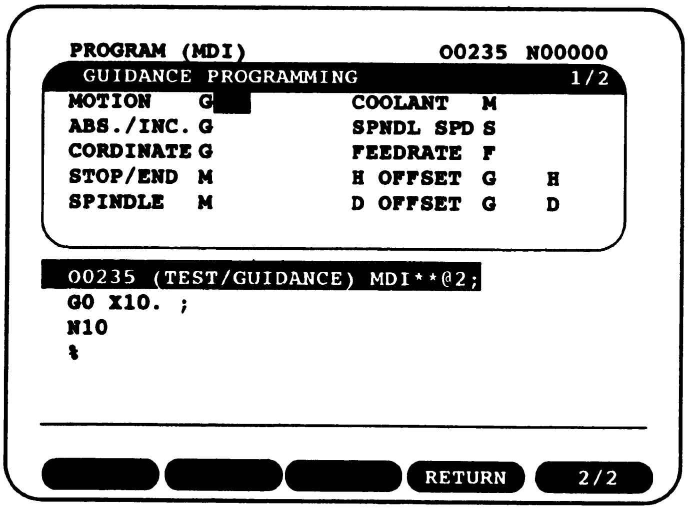
9 You can register the program executed by MDI operation into memory.

a Once the program has been executed, the message "REGISTER THIS KEYED IN PROGRAM?" appears, provided the memory protect switch on the machine operator's panel is released.

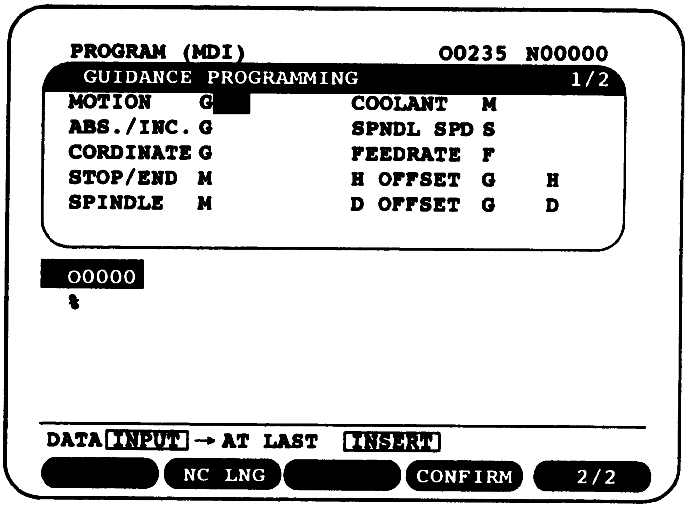


Pressing the **[YES]** soft key adds the program executed by the guidance programming MDI operation to the end of the program having the number entered in step 3. Pressing the **[NO]** soft key abandons registration of the program.

b You can display the contents of the program having the number entered in step 3 by pressing the **[CONFIRM]** soft key.
The cursor, used to indicate a block, can be moved with the page keys.



Pressing the **[RETURN]** soft key displays the contents of program O0000, the original program for guidance programming MDI operation.



c To continue MDI operation after registering the program into memory, repeat step 7 and subsequent steps.

10 To erase all programs created by MDI, press the RESET key. (In this case, bit 7 (MCL) of parameter No. 3203 must be set to 1.)

11 When stopping or terminating MDI operation, perform the following:

a **Stopping MDI operation**
Pressing the feed hold switch on the machine operator's panel turns on the feed hold LED and turns off the cycle start LED. The machine enters the following state:
(i) When the machine is moving the tool, the tool is decelerated until it stops.
(ii) During dwell, dwell is paused.
(iii) When the machine is performing an M, S, or T operation, the machine stops once the operation has been completed.
Pressing the cycle start switch on the machine operator's panel restarts the machine.

b **Terminating MDI operation**
Pressing the RESET key on the CRT/MDI panel terminates automatic operation and places the system in the reset state. When the system is reset while a tool is being moved, the tool is decelerated until it stops.

12 Pressing the **[NC OPR]** soft key enables programming with the CNC language. See Sections 9.2 to 9.7 for details of programming with the CNC language.

13 To execute the program displayed on the CNC language screen, position the cursor to the start of the program (or an arbitrary position within the program). Then, press the cycle start button on the machine operator's panel. The created program is started.
When a program end block (M02 or M03) or ER (%) is executed, the created program is automatically deleted, thus ending the operation.
When M99 is executed, execution returns to the start of the created program.

```
PROGRAM (MDI)                                  O0001 N00003
  O0000 G00 X100.0 Z200.;
  M03;
  G01 Z120.0 F500;
  M93 P9010;
  G00 Z0.0;
  %

  G00  G69 G21 G25 G80 G54
  G97  G99 G40 G22 G67 G18
                 B       H    M
       T                 D
       F             S
>_
MDI **** *** ***                        12:42:39
[ PRGRM ][ MDI ][ CURRNT ][ NEXT ][ (OPRT) ]
```

Explanations

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation, M30 does not return control to the beginning of the program (M99 performs this function).

• Erasing the program

Programs prepared in the **MDI** mode will be erased in the following cases:

- In MDI operation, if M02, M30 or ER(%) is executed.
  (If bit 6 (MER) of parameter No.3203 is set to 1, however, the program is erased when execution of the last block of the program is completed by single–block operation.)
- In **MEMORY** mode, if memory operation is performed.
- In **EDIT** mode, if any editing is performed.
- Background editing is performed.
- Upon reset when bit 7 (MCL) of parameter No. 3203 is set to 1

• Restart

If the program is restarted during editing on the CNC language screen while the MDI operation is stopped, it is restarted from the line to which the cursor is positioned.
On the guidance programming screen, the program is always restarted from the first line.

Limitations

• Program registration

On the CNC language screen, you cannot register a program created by an MDI operation.

- **Number of lines in a program**

1. **On the guidance programming screen**
   A program of up to nine lines can be created.
2. **On the CNC language screen**
   A program consisting of up to six lines can be created. When parameter MDL (No. 3107 #7) is set to 0 to specify a mode that suppresses the display of continuous–state information, a program of up to 10 lines can be created.
   If the created program exceeds the specified number of lines, % (ER) is deleted (prevents insertion and modification).

- **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. In addition to the main program executed by automatic operation, up to two levels of subprogram nesting are allowed (when the custom macro option is provided, up to four levels are allowed).

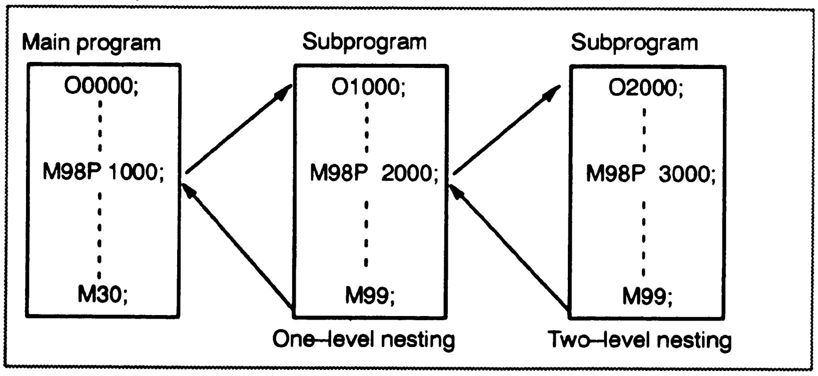


Fig. 4.2(a)Nesting Level of Subprograms Called from the MDI Program

- **Macro call**

When the custom macro option is provided, macro programs can also be created, called, and executed in the **MDI** mode. However, macro call commands cannot be executed when the mode is changed to **MDI** mode after memory operation is stopped during execution of a subprogram.

- **Memory area**

When a program is created in the **MDI** mode, an empty area in program memory is used. If program memory is full, no programs can be created in the **MDI** mode.

- **Editing a program during a guidance programming MDI operation**

Programs registered in memory (which can be displayed by pressing the **[CONFIRM]** soft key) cannot be edited during a guidance programming MDI operation. If you attempt to edit such a program, the message "CANNOT EDIT" is displayed.

Notes

If a program executed by a guidance programming MDI operation includes a miscellaneous function, such as M02 or M30, programs created in MDI mode may be cleared if the miscellaneous function resets the CNC. Whether programs created in MDI mode are cleared with a reset signal can be specifiedwith the following parameter. When the parameter is set to clear programs, programs cannot be registered into memory.
Bit 7 (MCL) of parameter No. 3203
0 : Programs created in MDI mode are not cleared upon reset.
1 : Programs created in MDI mode are cleared upon reset.

4.3 DNC OPERATION

DNC operation is mode of machine operation is which the machine runs on a program being read directly from an external I/O device. The program need not reside in the CNC memory. This mode is useful when the program is too large to fit in the CNC memory.

Procedure for MDI Operation

Procedure

1 Select the MDI mode, and specify an I/O device channel connected to the seting data (I/O channel) on the setting screen.

2 Press the MEMORY switch (one of the mode selection switches).

3 Locate the beginning of the program on the I/O device.

4 Press the REMOTE switch on the machine operator's panel to select the RMT mode.

5 Press the CYCLE START button.

Now the DNC operation is activated. It can be stopped and restarted in the same way as memory operation.

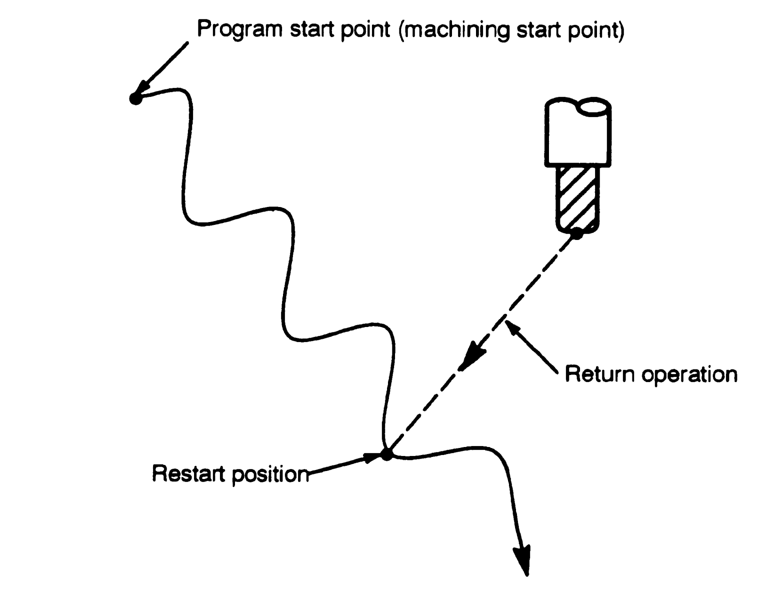
Explanations

- Subprograms in memory can be called during DNC operation.
- During DNC operation, custom macros can be issued, but repetition or branch commands cannot be used.
- Macro programs in memory can be called during DNC operation.
- During DNC operation, a subprogram or macro program cannot make a return to the calling program by issuing a return command with a sequence number specified (M99P_).

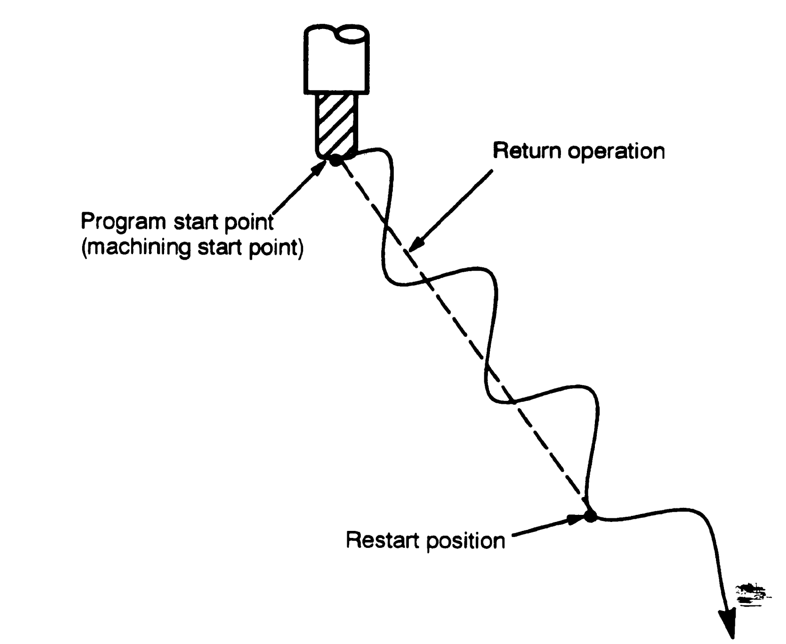
4.4 PROGRAM RESTART

This function specifies Sequence No. of a block 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.



Q TYPE	Before operation can be restarted, the machine must be moved to the programmed start point (machining start point)



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

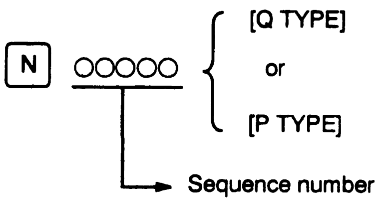
[COMMON TO P TYPE / Q TYPE]

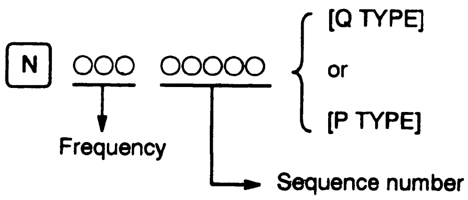
1 Turn the program restart switch on the machine operator's panel ON.

2 Press PROG on the CRT/MDI panel to display the desired program.

3 Find the program head.

4 Enter the sequence number of the block to be restarted, then press the **[P TYPE]** or **[Q TYPE]** soft key.





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.

5 The sequence number is searched for, and the program restart screen appears on the CRT display.

```
PROGRAM RESTART                              O0002 N01000

  DESTINATION              M          1            2
    X      57. 096                    1            2
    Y      56. 877                    1            2
    Z      56. 943                    1            2
                                      1            2
                                      1 * * * * * * * *
  DISTANCE TO GO           * * * * * * * * * * * * * * * *
    X       1. 459
    Y      10. 309         T * * * * * * * *   * * * * * * * *
    Z       7. 320         S * * * * *

                                          S    0   T0000
MEM * * * *   * * *   * * *        10 : 10 : 40
[ RSTR ][       ][ FL.SDL ][       ][ (OPRT) ]
```

DESTINATION shows the position at which machining is to restart. DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter setting) along which the tool moves to the restart position.

M: Fourteen most recently specified M codes
T: Two most recently specified T codes
S: Most recently specified S code
B: Most recently specified 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.

6 Turn the program re–start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the 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.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter settings (No. 7310). 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.

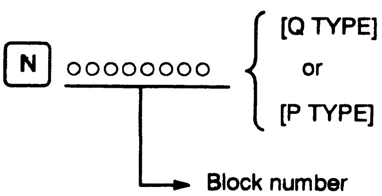
3 If necessary, modify the offset amount.

Procedure 2

[COMMON TO P TYPE / Q TYPE]

1 Turn the program restart switch on the machine operator's panel ON.

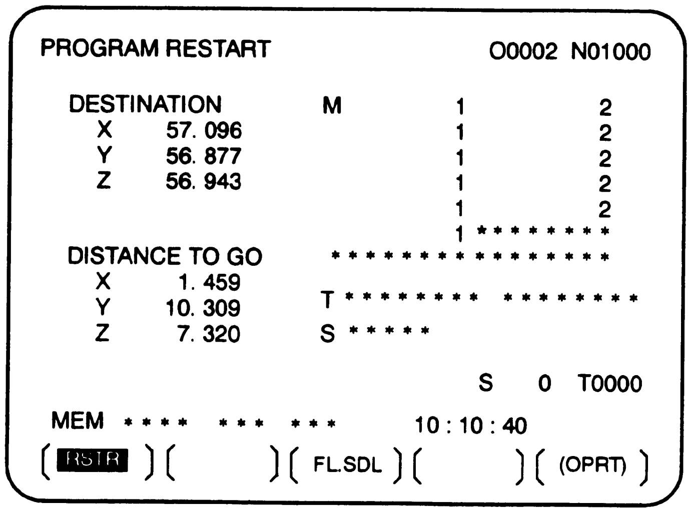
2 Press PROG on the CRT/MDI panel to display the desired program.



3 Find the program head. Press function RESET key.

4 Enter the number of the block to be restarted then press the **[P TYPE]** or **[Q TYPE]** soft key. The block number cannot exceed eight digits.

5 The block number is searched for, and the program restart screen appears on the CRT display.



DESTINATION shows the position at which machining is to restart. DISTANCE TO GO shows the distance from the current tool position to the position where machining is to restart. A number to the left of each axis name indicates the order of axes (determined by parameter

setting) along which the tool moves to the restart position.
M: Fourteen most recently specified M codes
T: Two most recently specified T codes
S: Most recently specified S code
B: Most recently specified 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.

6 Turn the program re–start switch OFF. At this time, the figure at the left side of axis name DISTANCE TO GO blinks.

7 Check the screen for the 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.

8 Check that the distance indicated under DISTANCE TO GO is correct. Also check whether there is the possibility that the tool might hit a workpiece or other objects when it moves to the machining restart position. If such a possibility exists, move the tool manually to a position from which the tool can move to the machining restart position without encountering any obstacles.

9 Press the cycle start button. The tool moves to the machining restart position at the dry run feedrate sequentially along axes in the order specified by parameter settings (No. 7310). Machining is then restarted.

Explanations

• 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. Z–120. 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.

- **Block number exceeding eight digits**

When the block number displayed on the program screen exceeds eight digits, the block number is reset to 0 and counting continues.

Limitations

- **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)

- **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.

• 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 intervention can be used to perform a return operation for an axis if it has not yet been performed for the axis. A return operation cannot be done further on axes for which a return has already been completed.

• 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.

• 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.

Alarm

Alarm No.	Contents
071	The specified block number for restarting the program is not found.
094	After interruption, a coordinate system was set, then P–type restart was specified.
095	After interruption, the coordinate system shift was changed, then P–type restart was specified.
096	After interruption, the coordinate system was changed, then P–type restart was specified.
097	When automatic operation has not been performed since the power was turned on, emergency stop was released, or alarm No. 094 to 097 was reset, P–type restart was specified.
098	After the power was turned on, restart operation was performed without reference position return, but a G28 command was found in the program.
099	A move command was specified from the MDI panel during a restart operation.
5020	Program restart parameter setting error.

Notes

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.
- Manual operation is performed when the machine is locked.
- When the mirror image is used.
- 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.

4.5 SCHEDULING FUNCTION

The schedule function allows the operator to select files (programs) registered on a floppy–disk in an external input/output device (Handy File, Floppy Cassette, or FA Card) and specify the execution order and number of repetitions (scheduling) for performing automatic operation. It is also possible to select only one file from the files in the external input/output device and execute it during automatic operation.

FILE DIRECTORY

FILE NO.	FILE NAME
0001	O0010
0002	O0020
0003	O0030
0004	O0040

List of files in an external input/output device

⇩ Set file number and number of repetitions.

ORDER	FILE NO	REPETITION
01	0002	2
02	0003	1
03	0004	3
04	0001	2

Scheduling screen

⇩

Executing automatic operation

Procedure for Scheduling Function

Procedure

·Procedure for executing one file

1 Press the **MEMORY** switch on the machine operator's panel, then press the PROG function key on the MDI panel.

2 Press the rightmost soft key (continuous menu key), then press the **[FL. SDL]** soft key. A list of files registered in the Floppy Cassette is displayed on screen No. 1. To display more files that are not displayed on this screen, press the page key on the MDI panel. Files registered in the Floppy Cassette can also be displayed successively.

```
FILE DIRECTORY                              O0001 N00000
 CURRENT SELECTED : SCHEDULE
  NO.     FILE NAME               (METER)    VOL
  0000    SCHEDULE
  0001    PARAMETER                58.5
  0002    ALL PROGRAM              11.0
  0003    O0001                    1.9
  0004    O0002                    1.9
  0005    O0010                    1.9
  0006    O0020                    1.9
  0007    O0040                    1.9
  0008    O0050                    1.9

 MEM  ****  ***  ***           19 : 14 : 47
[PRGRM ][      ][ DIR  ][SCHDUL][(OPRT) ]
```

Screen No.1

3 Press the **[(OPRT)]** and **[SELECT]** soft keys to display "SELECT FILE NO." (on screen No. 2). Enter a file number, then press the **[F SET]** and **[EXEC]** soft keys. The file for the entered file number is selected, and the file name is indicated after "CURRENT SELECTED:".

```
FILE DIRECTORY                              O0001 N00000
 CURRENT SELECTED:O0040
  NO.     FILE NAME               (METER)      VOL
  0000    SCHEDULE
  0001    PARAMETER                58.5
  0002    ALL PROGRAM              11.0
  0003    O0001                    1.9
  0004    O0002                    1.9
  0005    O0010                    1.9
  0006    O0020                    1.9
  0007    O0040                    1.9
  0008    O0050                    1.9
  SELECT FILE NO.=7

>_
MEM  ****  ***  ***            19 : 17 : 10
[F SET ][      ][      ][      ][ EXEC ]
```

Screen No.2

4 Press the **REMOTE** switch on the machine operator's panel to enter the **RMT** mode, then press the cycle start switch. The selected file is executed. For details on the **REMOTE** switch, refer to the manual supplied by the machine tool builder. The selected file number is indicated at the upper right corner of the screen as an F number (instead of an O number).

```
FILE DIRECTORY                          F0007  N00000
 CURRENT SELECTED:O0040








RMT  ****  ***  ***        13 : 27 : 54
[ PRGRM ][       ][ DIR  ][ SCHDUL ][ (OPRT) ]
```

Screen No.3

- **Procedure for executing the scheduling function**

1 Display the list of files registered in the Floppy Cassette. The display procedure is the same as in steps 1 and 2 for executing one file.

2 On screen No. 2, press the **[(OPRT)]** and **[SELECT]** soft keys to display "SELECT FILE NO."

3 Enter file number 0, and press the **[F SET]**, and **[EXEC]** soft keys. "SCHEDULE" is indicated after "CURRENT SELECTED:".

4 Press the leftmost soft key (return menu key) and the **[SCHDUL]** soft key. Screen No. 4 appears.

```
FILE DIRECTORY                          F0000  N02000
   ORDER  FILE NO.              REQ.REP       CUR.REP
    01
    02
    03
    04
    05
    06
    07
    08
    09
    10

>_
MEM  ****  ***  ***        22 : 07 : 00
[ PRGRM ][       ][ DIR  ][ SCHDUL ][ (OPRT) ]
```

Screen No.4

Move the cursor and enter the file numbers and number of repetitions in the order in which to execute the files. At this time, the current number of repetitions "CUR.REP" is 0.

5 Press the **REMOTE** switch on the machine operator's panel to enter the **RMT** mode, then press the start switch. The files are executed in the specified order. When a file is being executed, the cursor is positioned at the number of that file.
The current number of repetitions CUR.REP is increased when M02 or M30 is executed in the program being run.

```
FILE DIRECTORY                               O0000  N02000
    ORDER  FILE NO.              REQ.REP        CUR.REP
     01      0007                   5               5
     02      0003                  23              23
     03      0004                9999             156
     04      0005                LOOP               0
     05
     06
     07
     08
     09
     10

 RMT  ****   ***   ***        10 : 10 : 40
[ PRGRM ][       ][ DIR   ][ SCHDUL ][ (OPRT) ]
```

Screen No.5

Explanations

- **Specifying no file number**

If no file number is specified on screen No. 4 (the file number field is left blank), program execution is stopped at that point. To leave the file number field blank, press numeric key [0] then [INPUT].

- **Endless repetition**

If a negative value is set as the number of repetitions, **<LOOP>** is displayed, and the file is repeated indefinitely.

- **Clear**

When the **[(OPRT)]**, **[CLEAR]**, and **[EXEC]** soft keys are pressed on screen No. 4, all data is cleared. However, these keys do not function while a file is being executed.

- **Return to the program screen**

When the **[PRGRM]** soft key is pressed on screen No. 1, 2, 3, 4, or 5, the program screen is displayed.

Restrictions

- **Number of repetitions**

Up to 9999 can be specified as the number of repetitions. If 0 is set for a file, the file becomes invalid and is not executed.

- **Number of files registered**

By pressing the page key on screen No. 4, up to 20 files can be registered.

- **M code**

When M codes other than M02 and M30 are executed in a program, the current number of repetitions is not increased.

- **Displaying the floppy disk directory during file execution**

During the execution of file, the floppy directory display of background editing cannot be referenced.

- **Restarting automatic operation**

To resume automatic operation after it is suspended for scheduled operation, press the reset button.

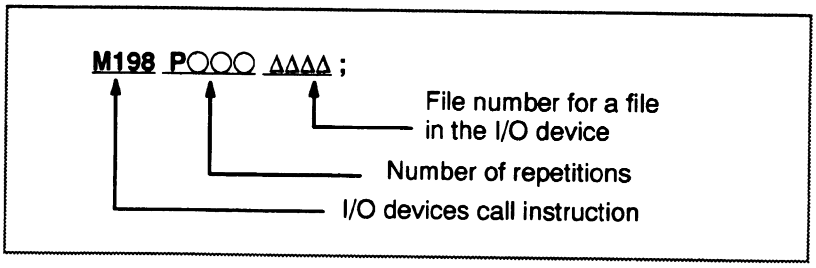
Alarm

Alarm No.	Description
086	An attempt was made to execute a file that was not registered in the floppy disk.
210	M198 and M99 were executed during scheduled operation, or M198 was executed during DNC operation.

4.6 SUBPROGRAM CALL FUNCTION

The subprogram call function is provided to call and execute subprogram files stored in an external input/output device(Handy File, FLOPPY CASSETTE, FA Card)during memory operation.
When the following block in a program in CNC memory is executed, a subprogram file in the external input/output device is called:

Format



Explanation

The subprogram call function is enabled when parameter No.0102 for the input/output device is set to 3. When the custom macro option is provided, either format 1 or 2 can be used. A different M code can be used for a subprogram call depending on the setting of parameter No.6030. In this case, M198 is executed as a normal M code. The file number is specified at address P. If the SBP bit (bit 2) of parameter No.3404 is set to 1, a program number can be specified. When a file number is specified at address P, Fxxxx is indicated instead of Oxxxx.

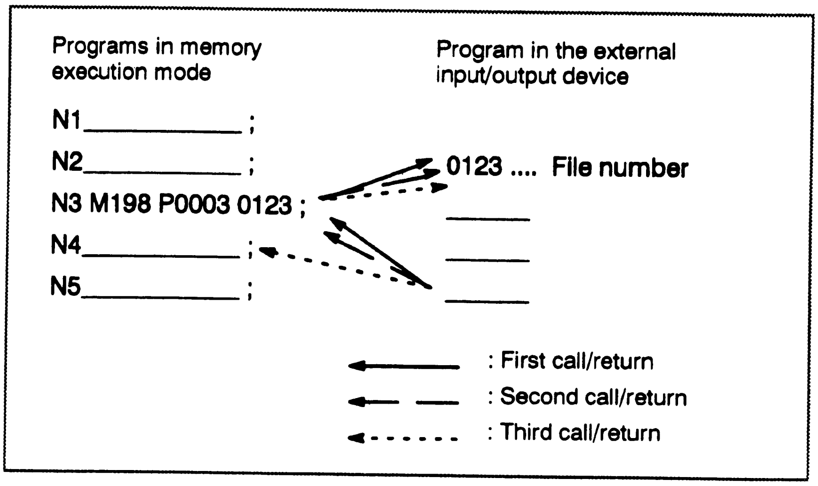


Fig.4.6 (a) Program Flow When M198 is Specified

Notes

1. When M198 in the program of the file saved in a floppy cassette is executed, a P/S alarm (No.210) is given. When a program in the memory of CNC is called and M198 is executed during execution of a program of the file saved in a floppy cassette, M198 is changed to an ordinary M–code.
2. When MDI is intervened and M198 is executed after M198 is commanded in the memory mode, M198 is changed to an ordinary M–code. When the reset operation is done in the MDI mode after M198 is commanded in the MEMORY mode, it does not influence on the memory operation and the operation is continued by restarting it in the MEMORY mode.

4.7 MANUAL HANDLE INTERRUPTION

The movement by manual handle operation can be done by overlapping it with the movement by automatic operation in the automatic operation mode.

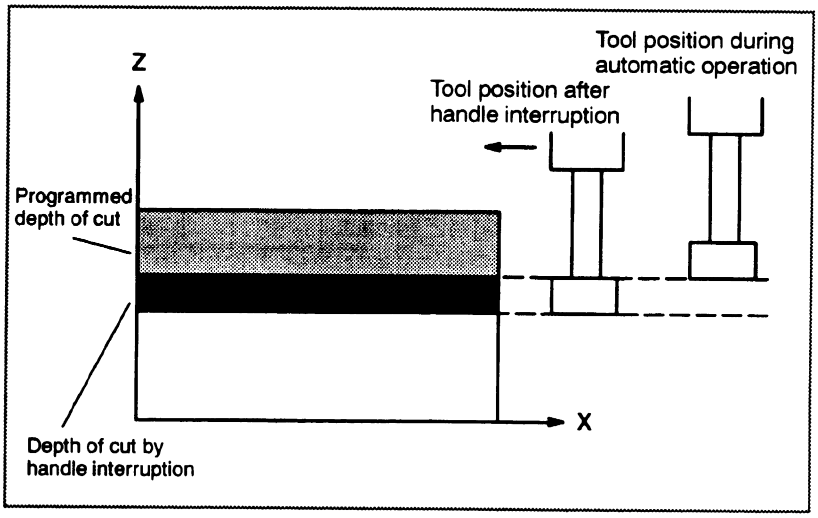


Fig 4.7 (a) Manual Handle Interruption

· Handle interruption axis selection signals
  For the handle interruption axis selection signals, refer to the manual supplied by the machine tool builder.

During automatic operation, handle interruption is enabled for an axis if the handle interruption axis selection signal for that axis is on. Handle interruption is performed by turning the handle of the manual pulse generator.

Notes

1. The travel distance by handle interruption is determined according to the amount by which the manual pulse generator is turned and the handle feed magnification (x1, x10, xM, xN). Since this movement is not accelerated or decelerated, it is very dangerous to use a large magnification value for handle interruption.
   The move amount per scale at x1 magnification is 0.001 mm (metric output) or 0.0001 inch (inch output).
2. Handle interruption is disabled when the machine is locked during automatic operation.

Explanations

• Relation with other functions

The following table indicates the relation between other functions and the movement by handle interrupt.

Signal	Relation
Machine lock	Machine lock is effective. When the machine lock signal is on, handle interrupt is ignored.
Interlock	Interlock is effective. The tool does not move even when this signal turns on.
Mirror image	Mirror image is not effective. Interrupt functions on the plus direction by plus direction command, even if this signal turns on.

• Position display

The following table shows the relation between various position display data and the movement by handle interrupt.

Display	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Handle interruption does not change relative coordinates.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

• Travel distance display

Press the [POS] function key, then press the HNDL chapter selection soft key. The distance moved by the tool as a result of the handle interrupt is displayed. The following 4 kinds of data are displayed concurrently.

```
HANDLE INTERRUPTION                          O0000 N02000
   (INPUT UNIT)                           (OUTPUT UNIT)
   X       69.594                         X       69.594
   Y      137.783                         Y      137.783
   Z      –61.439                         Z      –61.439

   (RELATIVE)                             (DISTANCE TO GO)
   X        0.000                         X        0.000
   Y        0.000                         Y        0.000
   Z        0.000                         Z        0.000
                                PART COUNT      287
RUN TIME             1H 12M CYCLE TIME      0H 0M 0S

MDI  ****  ***  ***              10 : 29 : 51
[ ABS ][ REL ][ ALL ][ HNDL ][ (OPRT) ]
```

(a) INPUT UNIT : Handle interrupt move amount in input unit system Indicates the travel distance specified by handle interruption according to the least input increment.

(b) OUTPUT UNIT : Handle interrupt move amount in output unit system
Indicates the travel distance specified by handle interruption according to the least command increment.

(c) RELATIVE : Position in relative coordinate system
These values have no effect on the travel distance specified by handle interruption.

(d) DISTANCE TO GO : The remaining travel distance in the current block has no effect on the travel distance specified by handle interruption.

The handle interrupt move amount display is cleared when the manual reference position return.

4.8 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. To use this function, set the mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the CRT/MDI panel.

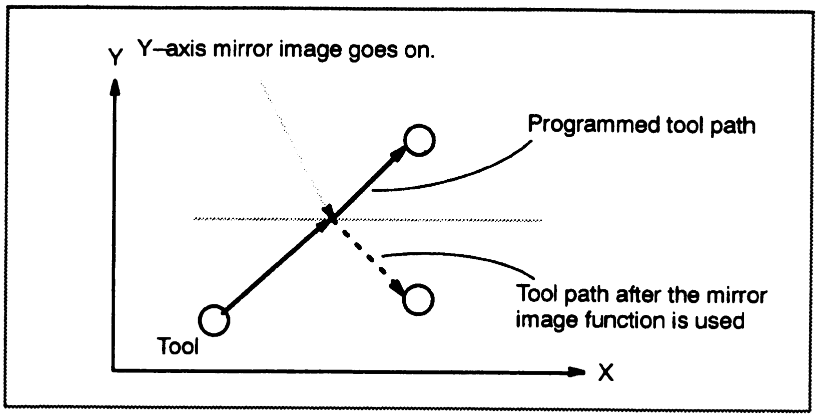


Fig 4.8 (a) Mirror Image

Procedure

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

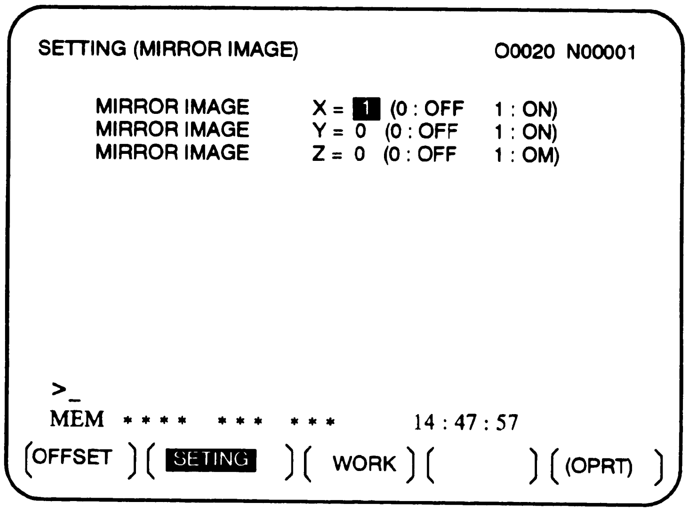
1 Press the single block switch to stop automatic operation. When the mirror image function is used from the begining of operation, this step is omitted.

2 Press the mirror image switch for the target axis on the machine operator's panel.
Alternatively, turn on the mirror image setting by following the steps below:

2–1 Set the **MDI** mode.

2–2 Press the OFFSET SETTING function key.

2–3 Press the **[SETING]** soft key for chapter selection to display the setting screen.



2–4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (memory mode or MDI mode), then press the cycle start button to start automatic operation.

Explanations

· The mirror image function can also be turned on and off by setting parameter MIR (No. 0012#0) to 1 (on) or 0 (off).

· For the mirror image switches, refer to the manual supplied by the machine tool builder.

Limitations

The direction of movement during manual operation, the direction of movement from an intemidiate 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.

4.9 MANUAL INTERVENTION AND RETURN

In cases such as when tool movement along an axis is stopped by feed hold during automatic operation so that manual intervention can be used to replace the tool: When automatic operation is restarted, this function returns the tool to the position where manual intervention was started.
To use the conventional program restart function and tool withdrawal and return function, the switches on the operator's panel must be used in conjunction with the MDI keys. This function does not require such operations.
To use manual intervention and return function, parameter MIN (No. 7001#0) is required to be set.

Explanations

• Manual absolute on/off

In manual absolute off mode, the tool does not return to the stop position, but instead operates according to the manual absolute on/off function.

• Override

For the return operation, the dry run feedrate is used, and the jog feedrate override function is enabled.

• Return operation

Return operation is performed according to positioning based on nonlinear interpolation.

• Single block

If the single block stop switch is on during return operation, the tool stops at the stop position and restarts movement when the cycle start switch is pressed.

• Cancellation

If a reset occurs or an alarm is issued during manual intervention or the return operation, this function is cancelled.

• MDI mode

This function can be used in the MDI mode as well.

Limitations

• Enabling and disabling manual intervention and return

This function is enabled only when the automatic operation hold LED is on. When there is no travel distance remaining, this function has no effect even if a feed hold stop is performed with the automatic operation hold signal *SP (bit 5 of G008).

• Offset

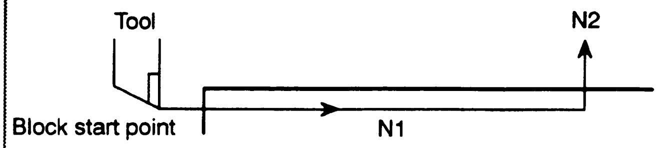
When the tool is replaced using manual intervention for a reason such as damage, the tool movement cannot be restarted by a changed offset in the middle of the interrupted block.

• Machine lock, mirror image, and scaling

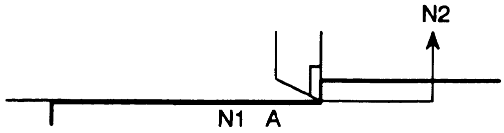
When performing manual intervention, never use the machine lock, mirror image, or scaling functions.

Example

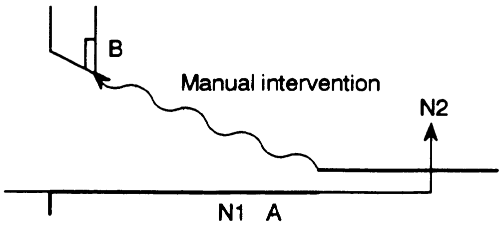
1. The N1 block cuts a workpiece



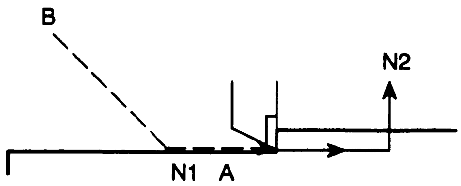
2. The tool is stopped by pressing the feed hold switch in the middle of the N1 block (point A).



3. After retracting the tool manually to point B, tool movement is restarted.



4. After automatic return to point A at the dry run feedrate, the remaining move command of the N1 block is executed.



Note

When performing manual intervention, pay particular attention of machining and the shape of the workpiece so that the machine and tool are not damaged.

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

1. **Machine Lock and Auxiliary Function Lock**
2. **Feedrate Override**
3. **Rapid Traverse Override**
4. **Dry Run**
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.

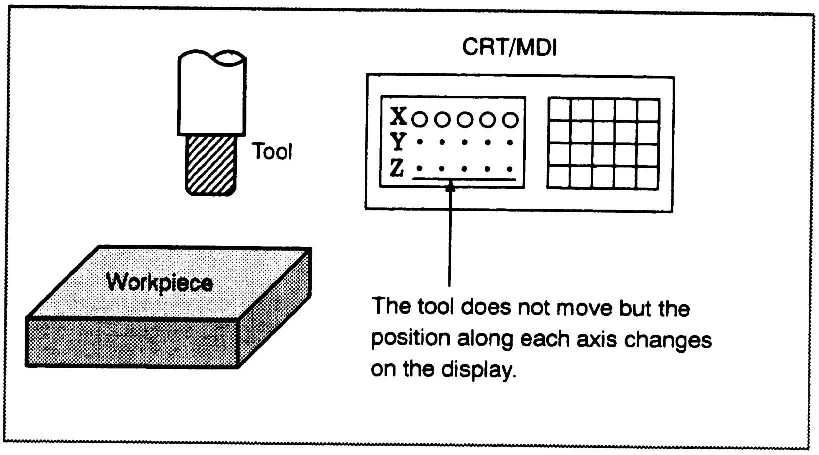


Fig. 5.1 Machine lock

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.
The position relationship between the workpiece coordinates and machine coordinates may change after automatic operation by the machine lock function has been executed. If this occurs, reset the workpiece coordinate system by specifying the coordinate system setting command or by making a manual reference position return.

• Auxiliary Function Lock

Press the auxiliary function lock switch on the operator's panel. M, S, and T codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Restrictions

• M, S, T command by only machine lock

M, S, and T commands are executed in the machine lock state.

• Reference position return under Machine Lock

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.

5.2 FEEDRATE OVERRIDE

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial.This feature is used to check a program.
For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

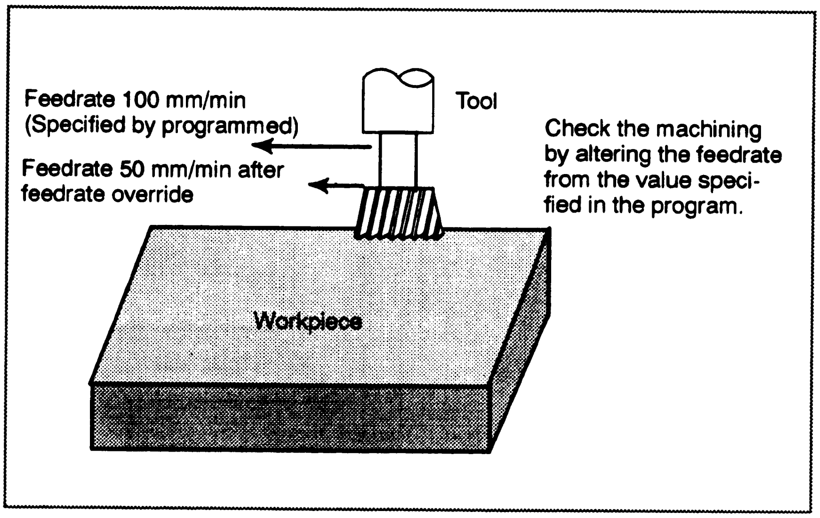
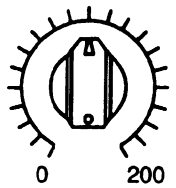


Fig. 5.2 Feedrate override

Procedure for Feedrate Override



Set the feedrate override dial to the desired percentage (%) on the machine operator's panel, before or during automatic operation.
On some machines, the same dial is used for the feedrate override dial and jog feedrate dial. Refer to the appropriate manual provided by the machine tool builder for feedrate override.

Restrictions

• Override Range

The override that can be specified ranges from 0 to 254%. For individual machines, the range depends on the specifications of the machine tool builder.

5.3 RAPID TRAVERSE OVERRIDE

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter (No. 1421).

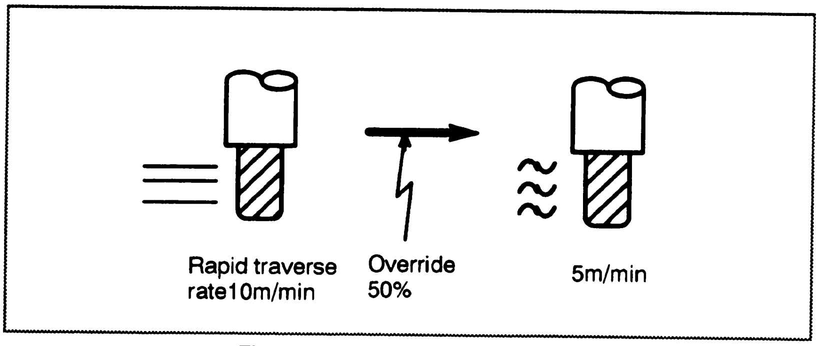
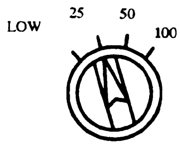


Fig. 5.3 Rapid traverse override

Rapid Traverse Override

Procedure



Rapid traverse override

Select one of the four feedrates with the rapid traverse override switch during rapid traverse. Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.

Explanation

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

1) Rapid traverse by G00.
2) Rapid traverse during a canned cycle.
3) Rapid traverse in G27, G28 and G30.
4) Manual rapid traverse.
5) Rapid traverse of manual reference position return.

5.4 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 taht the workpiece is removed from the table.

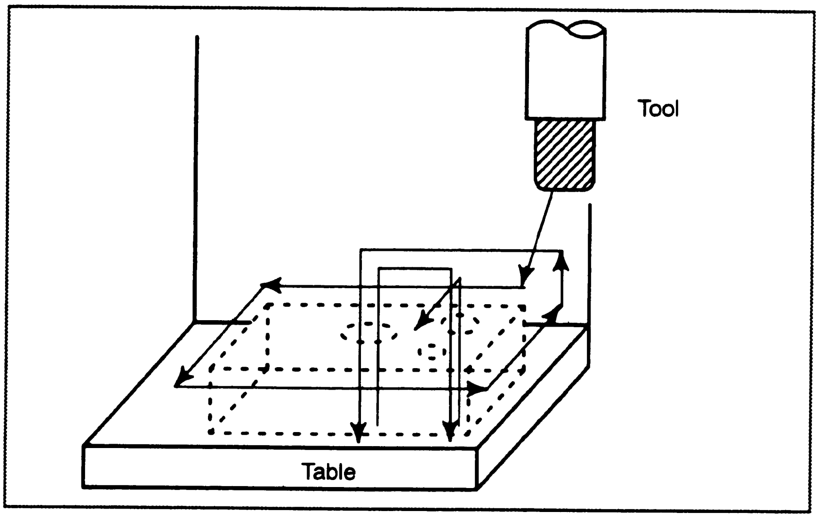


Fig. 5.4 Dry run

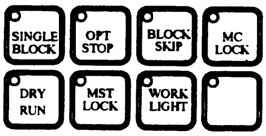
Procedure for Dry Run

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**



The dry run feedrate changes as shown in the table below according to the rapid traverse switch and parameters.

Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	Rapid traverse rate	Dry run feedrate x Max.JV *2)
OFF	Dry run speed x JV,or rapid traverse rate *1)	Dry run feedrate x JV *2)

Max. cutting feedrate . . . . . Setting by parameter No.1422

Rapid traverse rate . . . . Setting by parameter No.1420

Dry run feedrate . . . . Setting by parameter No.1410

JV: Jog feedrate override

*1: Dry run feedrate x JV when parameter RDR (bit 6 of No. 1401) is 1. Rapid traverse rate when parameter RDR is 0.

*2: Clamped at the maximum cutting feedrate.

JV max: Maximum value of jog feedrate override

5.5 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.

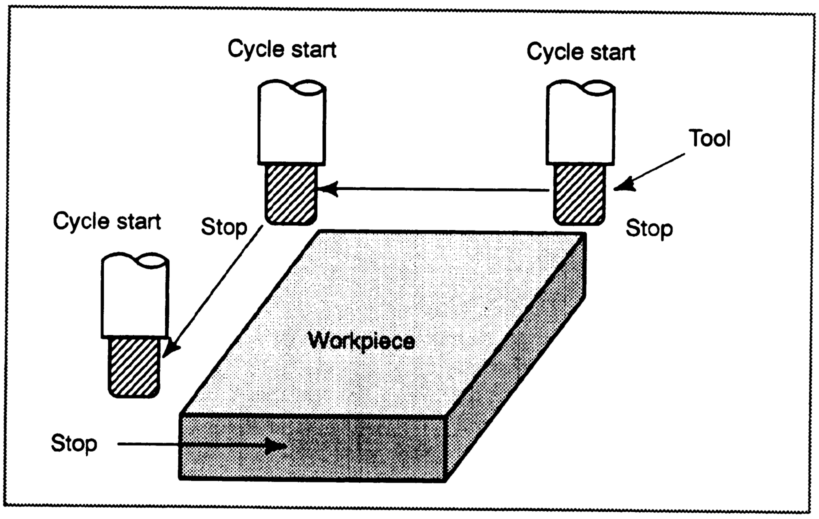


Fig. 5.5 (b) Single block

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.

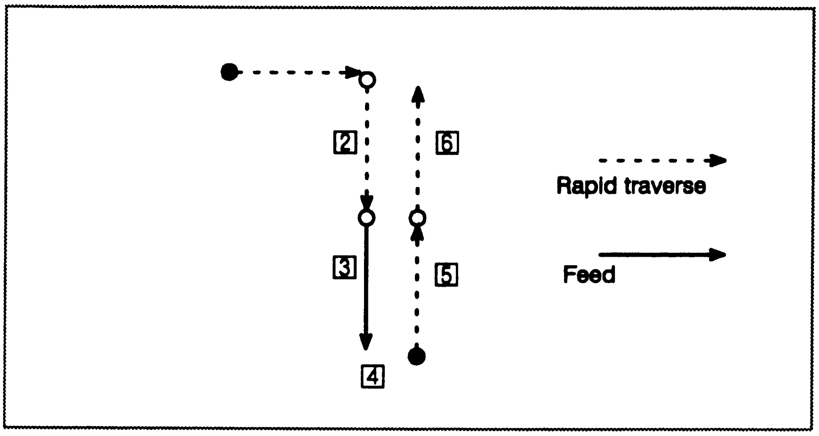


Fig. 5.5 (b) Single block during canned cycle

- **Subprogram call and single block**

Single block stop is not performed in a block containing M98P_;. M99; or G65.
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 or P.

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.

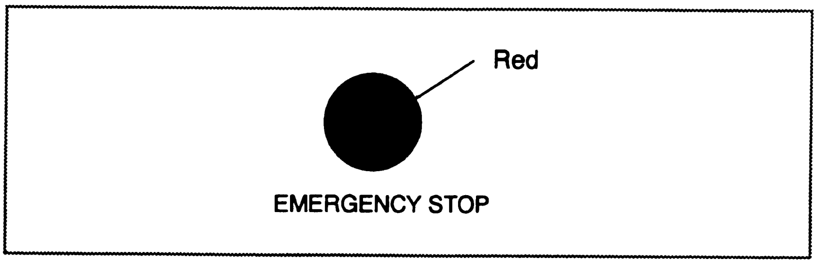


Fig. 6.1 Emergency stop

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

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.

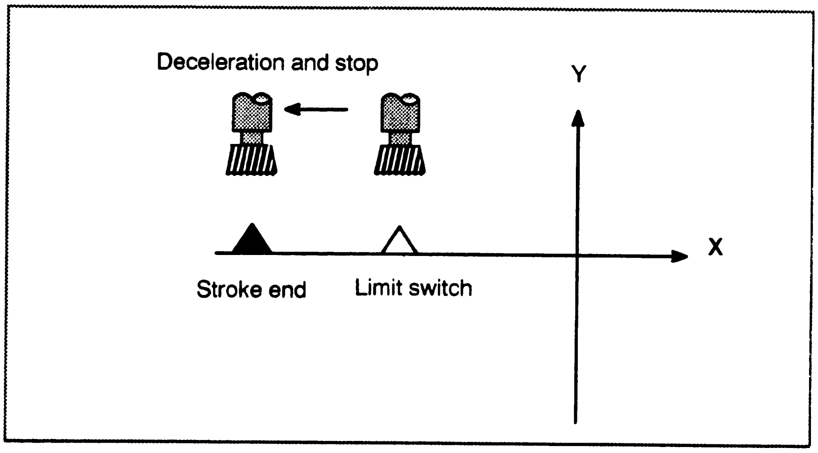


Fig. 6.2 Overtravel

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**

No.	Message	Description
506	Overtravel: +n	The tool has exceeded the hardware–specified overtravel limit along the positive nth axis (n: 1 to 3).
507	Overtravel: –n	The tool has exceeded the hardware–specified overtravel limit along the negative nth axis (n: 1 to 3).

6.3 STROKE CHECK

Two areas which the tool cannot enter can be specified with stored stroke limit 1 and stored stroke limit 2.

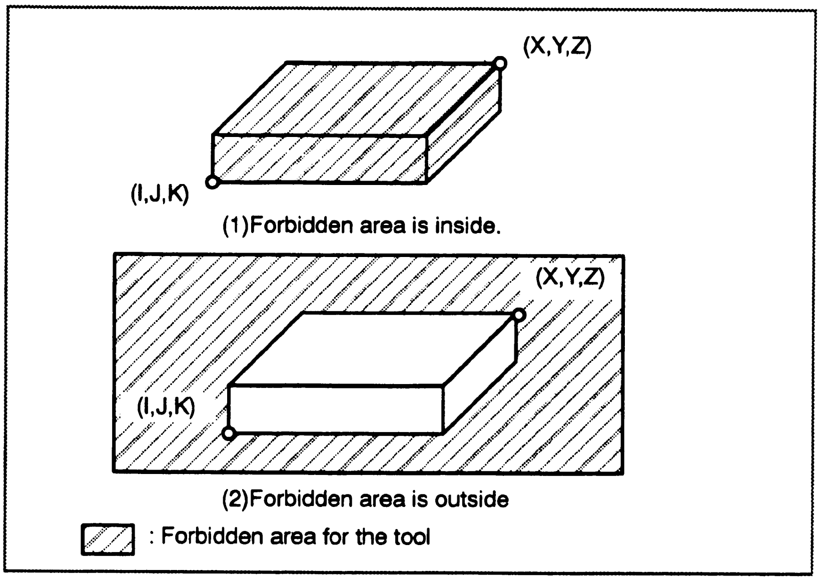


Fig. 6.3 (a) Stroke check

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 limit 1**

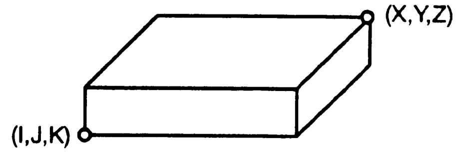
Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.

- **Stored stroke limit 2 (G22, G23)**

Parameters (Nos. 1322, 1323) or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter OUT (No. 1300#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:

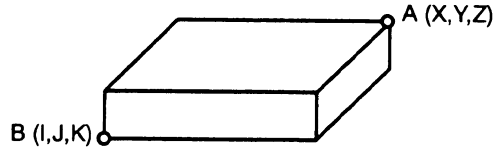
G 22X_Y_Z_I_J_K_;



X>I, Y>J, Z>K
X–I > 2000 (In least command increment)
Y–J > 2000 (In least command increment)
Z–K > 2000 ((In least command increment)

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.



X>I, Y>J, Z>K
X–I > 2000 (In least command increment)
Y–J > 2000 (In least command increment)
Z–K > 2000 (n least command increment)

Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

In limit 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 reference position in the least command increment. (Output increment)
If it is set by a G22 command, specify the data by the distance from the reference position 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.

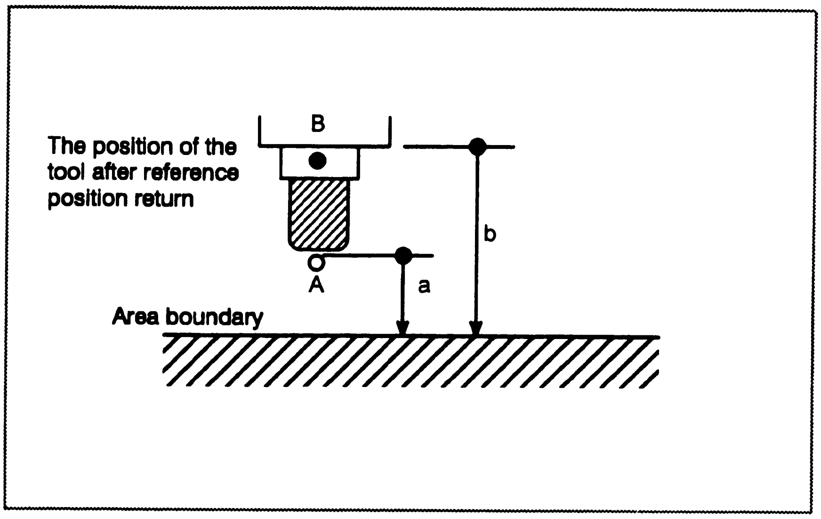


Fig. 6.3 (d) Setting the forbidden area

- **Forbidden area over–lapping**

Area can be set in piles.

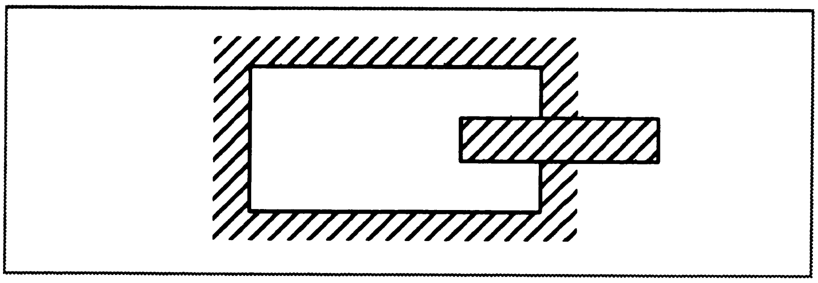


Fig. 6.3 (e) Setting the forbidden area over lapping

Unnecessary limits should be set beyond the machine stroke.

- **Effective time for a forbidden area**

Each limit becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed.
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 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.

Notes
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 limit 1, all areas are forbidden areas.
(2)When the forbidden area is limit 2, all areas are movable areas.

- **Overrun amount of stored stroke limit**

If the feed rate is F mm/minute, the maximum overrun amount, L mm, of the stored stroke limit is obtained from the following expression:

L mm=F/7500

The tool enters the specified inhibited area by up to L mm. The tool can be stopped at a position up to L mm in front of the inhibited area if parameter BFA (No. 1300#7) is specified accordingly. In this case, the tool will not enter the inhibited area.

Alarms

Number	Message	Contents
500	OVER TRAVEL: +n	Exceeded the n–th axis (1–3)+ side stored stroke limit I.
501	OVER TRAVEL: –n	Exceeded the n–th axis (1–3) – side stored stroke limit I.
502	OVER TRAVEL: +n	Exceeded the n–th axis (1–3)+ side stored stroke limit II.
503	OVER TRAVEL: –n	Exceeded the n–th axis (1–3) – side stored stroke limit II.

7 ALARM AND SELF–DIAGNOSIS FUNCTIONS

When an alarm occurs, the corresponding alarm screen appears to indicate the cause of the alarm. The causes of alarms are classified by error codes. Up to 25 previous alarms can be stored and displayed on the screen (alarm history display).
The system may sometimes seem to be at a halt, although no alarm is displayed. In this case, the system may be performing some processing. The state of the system can be checked using the self–diagnostic function.

7.1 ALARM DISPLAY

Explanations

• Alarm screen

When an alarm occurs, the alarm screen appears.

```
ALARM MESSAGE                                  O0000 N00000

      100    PARAMETER WRITE ENABLE
      510    OVER TRAVEL   :+1
      520    OVER TRAVEL   :+2
      530    OVER TRAVEL   :+3




                                             S    0 T0000
 MDI   ****   ***   *** ALM 18 : 52 : 05
[ARALM][ MSG ][ HISTRY ][       ][       ]
```

• Another method for alarm displays

In some cases, the alarm screen does not appear, but an ALM is displayed at the bottom of the screen.

```
PARAMETER (RS232C INTERFACE)            O1000 N00010

0100 ENS                       NCR        CTV
      0    0    0    0     0     0     0     0
0101 NFD                       ASI              SB2
      0    0    0    0     0     0     0     1
0102 DEVICE NUM. (CH0)                          2
0103 BAUDRATE (CH0)                            10
0111 NFD                       ASI              SB2
      0    0    0    0     0     0     0     0
0112 DEVICE NUM. (CH1)                          0
0113 BAUDRATE (CH1)                             0

)_                                   S        0 T0000
 MEM  ****   ***   *** ALM    08 : 41 : 27
[NO.SRH][ ON:1 ][ OFF:0 ][+INPUT ][ INPUT ]
```

In this case, display the alarm screen as follows:

1. Press the function key MESSAGE .
2. Press the chapter selection soft key **[ALARM]**.

• Reset of the alarm

Error codes and messages indicate the cause of an alarm. To recover from an alarm, eliminate the cause and press the reset key.

• Error codes

The error codes are classified as follows:
No. 000 to 255, 5000 to: Program errors(*)
No. 300 to 349: Absolute pulse coder (APC) alarms
No. 350 to 399: Serial pulse coder (SPC) alarms
No. 400 to 499: Servo alarms
No. 500 to 599: Overtravel alarms
No. 700 to 704: Overheat alarms
No. 750 to 799: Spindle alarms
No. 900 to 999: System alarms

*For an alarm (No. 000 to 255) that occurs in association with background operation, the indication "xxxBP/S alarm" is provided (where xxx is an alarm number). Only a BP/S alarm is provided for No. 140.
See the error code list in the appendix for details of the error codes.

7.2 ALARM HISTORY DISPLAY

Up to 25 of the most recent CNC alarms are stored and displayed on the screen.
Display the alarm history as follows:

Procedure for Alarm History Display

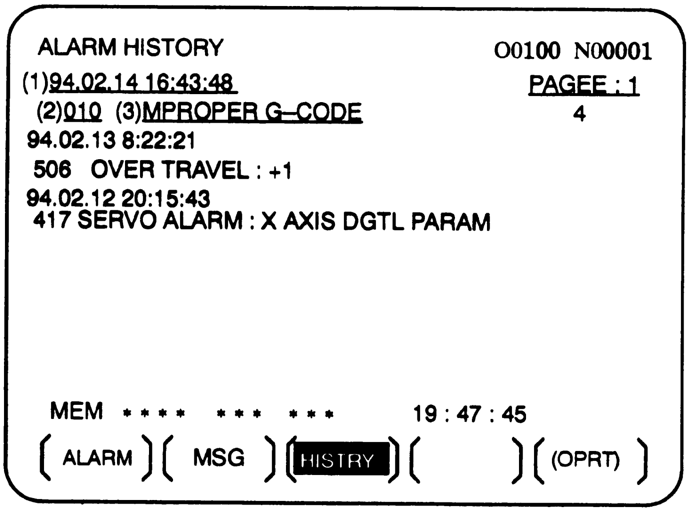
Procedure

1 Press the function key MESSAGE .

2 Press the chapter selection soft key **[HISTRY]**.
The alarm history appears.
The following information items are displayed.
(1)The date the alarm was issued
(2)Alarm No.
(3)Alarm message (some contains no message)

3 You can change the page by pressing a page key.

4 To delete the recorded information, press the softkey **[(OPRT)]** then the **[DELETE]** key.



(1)The date the alarm was issued
(2)Alarm No.
(3)Alarm message (some contains no message)
(4)Page No.

7.3 CHECKING BY SELF–DIAGNOSTIC SCREEN

The system may sometimes seem to be at a halt, although no alarm has occurred. In this case, the system may be performing some processing. The state of the system can be checked by displaying the self–diagnostic screen.

Procedure for Diagnois

1 Press the function key SYSTEM .

2 Press the chapter select key **[DGNOS]**.

3 The diagnostic screen has more than 1 pages. Select the screen by the following operation.
(1) Change the page by the 1–page change key.
(2) Method by soft key
 – Key input the number of the diagnostic data to be displayed.
 – Press **[N SRCH]**.

```
DIAGNOSTIC (GENERAL)                     O0000 N0000

000 WAITING FOR FIN SIGNAL                        :0
001 MOTION                                        :0
002 DWELL                                         :0
003 IN–POSITION CHECK                             :0
004 FEEDRATE OVERRIDE 0%                          :0
005 INTERLOCK/START–LOCK                          :0
006 SPINDLE SPEED ARRIVAL CHECK                   :0

)_
 EDIT **** *** ***          14 : 51 : 55
[ PARAM ][ DGNOS ][ PMC ][SYSTEM][ (OPRT) ]
```

Explanations

Diagnostic numbers 000 to 015 indicate states when a command is being specified but appears as if it were not being executed. The table below lists the internal states when 1 is displayed at the right end of each line on the screen.

Table 7.3 (a) Alarm displays when a command is specified but appears as if it were not being executed

No.	Display	Internal status when 1 is displayed
000	WAITING FOR FIN SIGNAL	M, S. T function being executed
001	MOTION	Move command in automatic operation being executed
002	DWELL	Dwell being executed
003	IN–POSITION CHECK	In–position check being executed
004	FEEDRATE OVERRIDE 0%	Cutting feed override 0%
005	INTERLOCK/START–LOCK	Interlock ON
006	SPINDLE SPEED ARRIVAL CHECK	Waiting for spindle speed arrival signal to turn on
010	PUNCHING	Data being output via reader puncher interface
011	READING	Data being input via reader puncher interface
012	WAITING FOR (UN) CLAMP	0 is always displayed. (unused)
013	JOG FEEDRATE OVERRIDE 0%	Jog override 0%
014	WAITING FOR RESET.ESP.RRW.OFF	Emergency stop, external reset, reset & rewind, or MDI panel reset key on
015	EXTERNAL PROGRAM NUMBER SEARCH	External program number searching

Table 7.3 (b) Alarm displays when an automatic operation is stopped or paused.

No.	Display	Internal status when 1 is displayed
020	CUT SPEED UP/DOWN	Set when emergency stop turns on or when servo alarm occurs
021	RESET BUTTON ON	Set when reset key turns on
022	RESET AND REWIND ON	Reset and rewind turned on
023	EMERGENCY STOP ON	Set when emergency stop turns on
024	RESET ON	Set when external reset, emergency stop, reset, or reset & rewind key turns on
025	STOP MOTION OR DWELL	A flag which stops pulse distribution. It is set in the following cases. (1)External reset turned on. (2)Reset & rewind turned on. (3)Emergency stop turned on. (4)Feed hold turned on. (5)The MDI panel reset key turned on. (6)Switched to the manual mode(JOG /HANDLE/INC). (7)Other alarm occurred. (There is also alarm which is not set.)

The table below shows the signals and states which are enabled when each diagnostic data item is 1. Each combination of the values of the diagnostic data indicates a unique state.

020	CUT SPEED UP/DOWN	1	0	0	0	1	0	0
021	RESET BUTTON ON	0	0	1	0	0	0	0
022	RESET AND REWIND ON	0	0	0	0	0	0	0
023	EMERGENCY STOP ON	1	0	0	0	0	0	0
024	RESET ON	1	1	1	1	0	0	0
025	STOP MOTION OR DWELL	1	1	1	1	1	1	0

Emergency stop signal input
External reset signal input
MDI reset button turned on
Reset & rewind input
Servo alarm generation
Changed to another mode or feed hold
Single block stop

Diagnostic numbers 030 and 031 indicate TH alarm states.

No.	Display	Meaning of data
030	CHARACTER NUMBER TH DATA	The position of the character which caused TH alarm is displayed by the number of characters from the beginning of the block at TH alarm
031	TH DATA	Read code of character which caused TH alarm

8 DATA INPUT/OUTPUT

NC data is transferred between the CNC and external input/output devices such as the Handy File.

The following types of data can be entered and output :

1.Program

2.Offset data

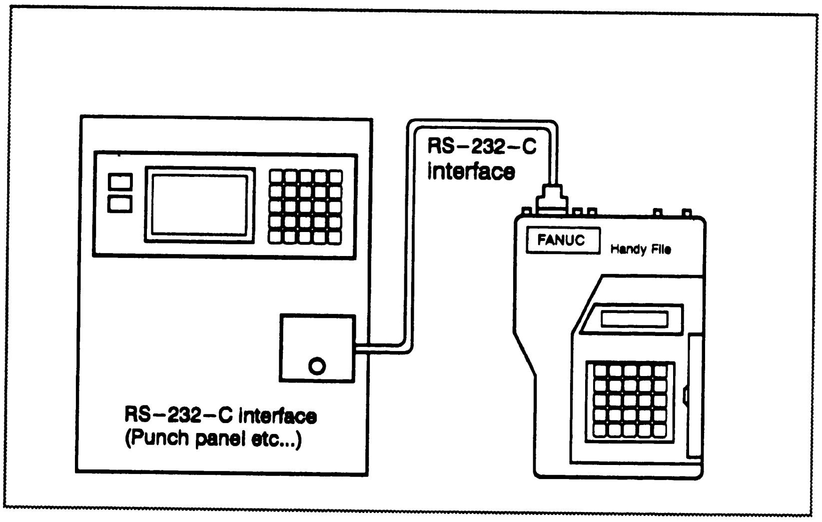
3.Parameter

4.Pitch error compensation data

5.Custom macro common variable

Before an input/output device can be used, the input/output related parameters must be set.

For how to set parameters, see Chapter 2 "**OPERATIONAL DEVICES**".



8.1 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 Hfilel. 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.

File 1	File 2	File 3		File n	Blank

- **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

File 1	File 2	File 3		File (k–1)	File k

Floppy 2

Continuation of file k	File (k+1)		File n	Blank

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.

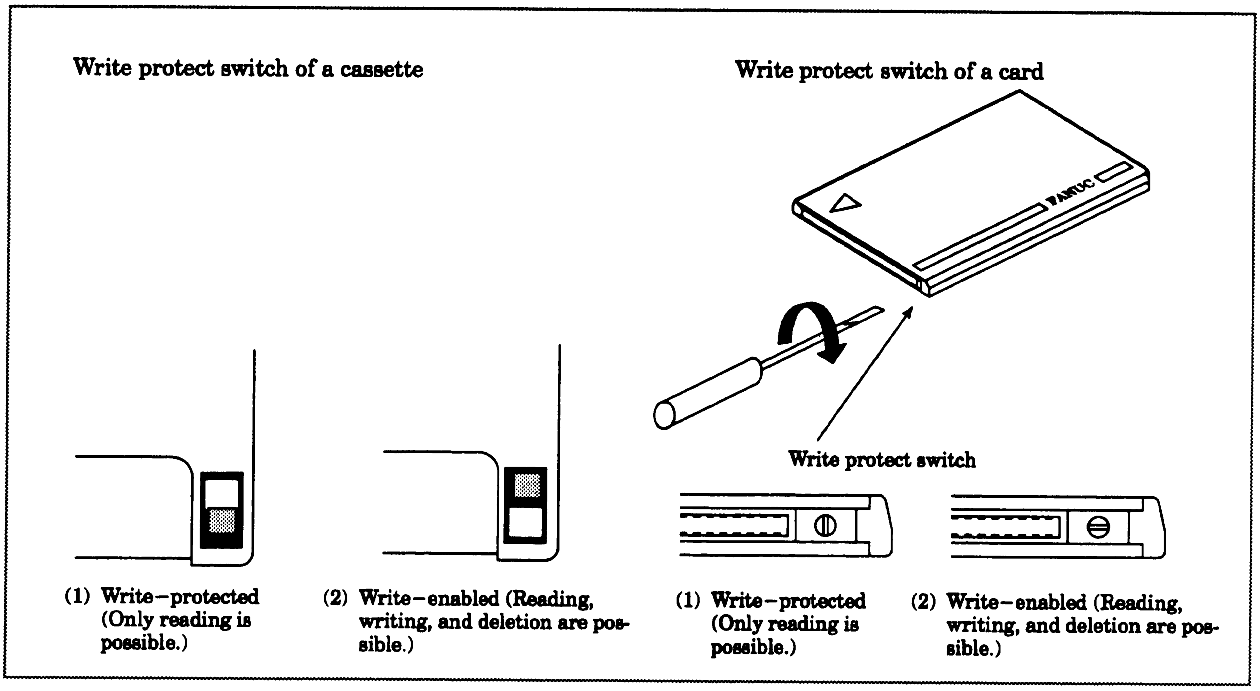


Fig. 8.1 Protect swtich

• Writing memo

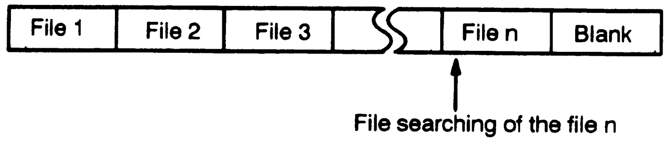
Once written in the cassette or card, data can subsequently be read out by correspondence between the data contents and file numbers. This correspondence cannot be verified, unless the data contents and file numbers are output to the CNC and displayed. The data contents can be displayed with display function for directory of floppy disk (See Section 8.8).
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	NC parameters
File 2	Offset data
File 3	NC program O0100
.	.
.	.
.	.
File (n–1)	NC program O0500
File n	NC program O0600

8.2 FILE SEARCH

When the program is input from the floppy, the file to be input first must be searched.
For this purpose, proceed as follows:



File heading

Procedure

1 Press the MEMORY switch on the machine operator's panel.

2 Press function key PROG .

3 Press soft key **[(OPRT)]**

4 Press the rightmost soft key ▷ (next–menu key).

5 Enter address N.

6 Enter the number of the file to search for.
·N0
The beginning of the cassette or card is searched.
·One of N1 to N9999
Of the file Nos. 1 to 9999, a designated file is searched.
·N–9999
The file next to that accessed just before is searched.
·N–9998
When N–9998 is designated, N–9999 is automatically inserted each time a file is input or output. This condition is reset by the designation of N1,N1 to 9999, or N – 9999 or reset.

7 Press soft keys **[F SRH]** and **[EXEC]**
The specified file is searched for.

Explanation

• File search by N–9999

The same result is obtained both by sequentially searching the files by specifying Nos. N1 to N9999 and by first searching one of N1 to N9999 and then using the N–9999 searching method. The searching time is shorter in the latter case.

Alarm

No.	Description
86	The ready signal (DR) of an input/output device is off. An alarm is not immediately indicated in the CNC even when an alarm occurs during head searching (when a file is not found, or the like). An alarm is given when the input/output operation is performed after that. This alarm is also raised when N1 is specified for writing data to an empty floppy. (In this case, specify N0.)

8.3 PROGRAM INPUT/OUTPUT

8.3.1 Inputting a Program

This section describes how to load a program into the CNC from a floppy or NC tape.

Inputting a program

Procedure

1 Make sure the input device is ready for reading.

2 Press the EDIT switch on the machine operator's panel.

3 When using a floppy, search for the required file according to the procedure in Section **8.2**.

4 Press function key PROG to display the program menu screen.

5 Press the rightmost soft key ▷ (next–menu key). Then the soft key **[READ]** is displayed.

6 Press soft keys **[READ]**.
The program is read and registered into the memory.

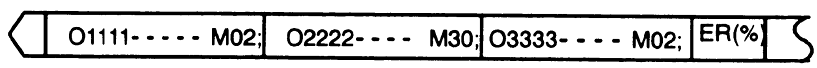
Explanations

• Collation

If a program is input while the data protect key on the machine operator's panel turns ON, the program loaded into the memory is verified against the contents of the floppy or NC tape.
If a mismatch is found during collation, the collation is terminated with an alarm (No. 079).
If the operation above is performed with the data protection key turns OFF, collation is not performed, but programs are registered in memory.

• Inputting multiple programs from an NC tape

When a tape holds multiple programs, the tape is read up to ER (or %).



- **Program numbers on a NC tape**

¢When a program is entered without specifying a program number.
·The O–number of the program on the NC tape is assigned to the program. If the program has no O–number, the N–number in the first block is assigned to the program.
·When the program has neither an O–number nor N–number, the previous program number is incremented by one and the result is assigned to the program.
·When the program does not have an O–number but has a five–digit sequence number at the start of the program, the lower four digits of the sequence number are used as the program number. If the lower four digits are zeros, the previously registered program number is incremented by one and the result is assigned to the program.

- **Program registration in the background**

The method of registration operation is the same as the method of foreground operation. However, this operation registers a program in the background editing area. As with edit operation, the operations described below are required at the end to register a program in foreground program memory.
Display the program menu screen.
[BG–END]

- **Registering a program having the same number as that of a previously registered program**

The following parameter can be used to specify whether any program having the same number as that of a registered program is to be registered:
Bit 2 (REP) of parameter No. 3201
0 : The program is not registered, causing P/S alarm 073.
1 : The program is registered after deletion of the already registered program. If editing is disabled, however, an alarm is issued without deleting the registered program.

Alarm

No.	Description
70	The size of memory is not sufficient to store the input programs
73	An attempt was made to store a program with an existing program number.
79	The verification operation found a mismatch between a program loaded into memory and the contents of the program on the floppy or NC tape.

8.3.2 Outputting a Program

A program stored in the memory of the CNC unit is output to a floppy or NC tape.

Outputting a program

Procedure

1 Make sure the output device is ready for output.

2 To output to an NC tape, specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key PROG to display the program menu screen.

5 Press the rightmost soft key ▷ (next–menu key) to display soft key **[PUNCH]**.

6 Enter a program number. If –9999 is entered, all programs stored in memory are output.

7 Press soft keys **[PUNCH]**
The specified program is output.
If skip the step 6, all programs in the memory are output.

Explanations (Output to a floppy)

• File output location

When output is conducted to the floppy, the program is output as the new file after the files existing in the floppy. New files are to be written from the beginning with making the old files invalid, use the above output operation after the N0 head searching.

• An alarm while a program is output

When an alarm No. 086 occurs during program output, the floppy is restored to the condition before the output.

• Outputting a program after file heading

When program output is conducted after N1 to N9999 head searching, the new file is output as the designated n–th position. In this case, 1 to n–1 files are effective, but the files after the old n–th one are deleted. If an alarm occurs during output, only the 1 to n–1 files are restored.

• Efficient use of memory

To efficiently use the memory in the cassette or card, output the program by setting parameter NFD (No. 0101#7,No. 0111#7 or 0121#7) to 1. This parameter makes the feed is not output, utilizing the memory efficiently.

• On the memo record

Head searching with a file No. is necessary when a file output from the CNC to the floppy is again input to the CNC memory or compared with the content of the CNC memory. Therefore, immediately after a file is output from the CNC to the floppy, record the file No. on the memo.

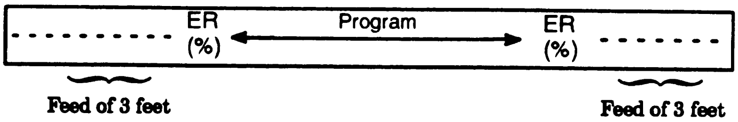
• Punching programs in the background

Punch operation can be performed in the same way as in the foreground. This function alone can punch out a program selected for foreground operation.
Display the program menu screen.
(Program No.) **[PUNCH]** : Punches out a specified program.
[PUNCH] : Punches out all programs.

Explanations
(Output to an NC tape)

• Format

A program is output to paper tape in the following format:



If three–feet feeding is too long, press the CAN key during feed punching to cancel the subsequent feed punching.

• TV check

A space code for TV check is automatically punched.

• ISO code

When a program is punched in ISO code, two CR codes are punched after an LF code.

- - - - - - - - - - - - - - LF CR CR

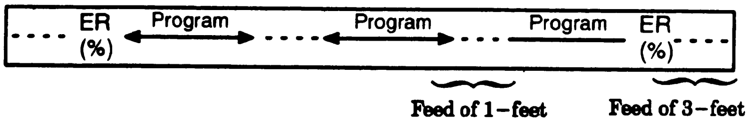
Bit 3 (NCR) of parameter No. 0100 can be set to display LF only.

• Stopping the punch

Press the RESET key to stop punch operation.

• Punching all programs

All programs are output to paper tape in the following format.



The sequence of the programs punched is undefined.

8.4 OFFSET DATA INPUT AND OUTPUT

8.4.1 Inputting Offset Data

Offset data is loaded into the memory of the CNC from a floppy or NC tape. The input format is the same as for offset value output. See section 8.4.2.

When an offset value is loaded which has the same offset number as an offset number already registered in the memory, the loaded offset data replaces existing data.

Inputting offset data

Procedure

1 Make sure the input device is ready for reading

2 Press the EDIT switch on the machine operator's panel.

3 When using a floppy, search for the required file according to the procedure in Section 8.2.

4 Press function key OFFSET SETTING to display the offset screen.

5 Press soft keys **[(OPRT)]**.

6 Press rightmost soft key ▷ (next menu key).

7 Press soft keys **[READ]** and **[EXEC]**.

8 The input offset data will be displayed on the screen after completion of input operation.

8.4.2 Outputting Offset Data

All offset data is output in a output format from the memory of the CNC to a floppy or NC tape.

Outputting offset data

Procedure

1 Make sure the output device is ready for output.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key [OFFSET SETTING] to display the offset screen.

5 Press soft key **[(OPRT)]**.

6 Press the rightmost soft key [▷] (next–menu key)

7 Press soft keys **[PUNCH]** and **[EXEC]**.
Offset data is output in the output format described below.

Explanations

• Output format

Output format is as follows:

Format

G10 L11 P_R_;

Where P_: Offset No.

R_: Tool compensation amount

The L1 command may be used instead of L11 for format compatibility of the conventional CNC.

• Output file name

When the floppy disk directory display function is used, the name of the output file is OFFSET.

8.5 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA

Parameters and pitch error compensation data are input and output from different screens, respectively. This chapter describes how to enter them.

8.5.1 Inputting Parameters

Parameters are loaded into the memory of the CNC unit from a floppy or NC tape. The input format is the same as the output format. See Section 8.6.2. When a parameter is loaded which has the same data number as a parameter already registered in the memory, the loaded parameter replaces the existing parameter.

Inputting parameters

Procedure

1 Make sure the input device is ready for reading.

2 When using a floppy, search for the required file according to the procedure in Section 8.2.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key OFFSET SETTING .

5 Press the soft key **[SETING]** for chapter selection to display the setting screen.

6 Enter 1 to "PARAMETER WRITE" in the setting data (PWE). Alarm No. 100 (indicating that parameters can be written) appears.

7 Press soft key SYSTEM .

8 Press chapter selection soft key **[PARAM]** to display the parameter screen.

9 Press soft key **[(OPRT)]**.

10 Press the rightmost soft key ▷ (next–menu key).

11 Press soft keys **[READ]** and **[EXEC]**.
Parameters are read into memory. Upon completion of input, the "INPUT" indicator at the lower–right corner of the screen disappears.

12 Press function key OFFSET SETTING .

13 Press soft key **[SETING]** for chapter selection.

14 Enter 0 to "PARAMETER WRITE" in the setting data.

15 Turn the power to the NC back on.

16 Release the EMERGENCY STOP button on the machine operator's panel.

8.5.2 Outputting Parameters

All parameters are output in the defined format from the memory of the CNC to a floppy or NC tape.

Outputting parameters

Procedure

1 Make sure the output device is ready for output.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key SYSTEM .

5 Press chapter selection soft key **[PARAM]** to display the parameter screen.

6 Press soft key **[(OPRT)]**.

7 Press rightmost soft key ▷ (next–menu key).

8 Press soft keys **[PUNCH]** and **[EXEC]**.
All parameters are output in the defined format.

Explanations

• Output format

Output format is as follows:
N .. P...... ;
N .. A1P ... A2P .. AnP ... ;
N .. P...... ;

N:Parameter No.
A:Axis No.(n is the number of control axis)
P:Parameter setting value .

• Output file name

When the floppy disk directory display function is used, the name of the output file is PARAMETER.

8.5.3 Inputting Pitch error compensation data

Pitch error compensation data are loaded into the memory of the CNC from a floppy or NC tape. The input format is the same as the output format. See Section 8.5.4. When a pitch error compensation data is loaded which has the corresponding data number as a pitch error compensation data already registered in the memory, the loaded data replaces the existing data.

Pitch error compensation data

Procedure

1 Make sure the input device is ready for reading.

2 When using a floppy, search for the required file according to the procedure in Section 8.2.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key [OFFSET SETTING].

5 Press the soft key **[SETING]** for chapter selection.

6 Enter 1 to "PARAMETER WRITE" in the setting data (PWE). Then P/S alarm No. 100 (indicating that parameters can be written) appears.

7 Press soft key [SYSTEM].

8 Press the rightmost soft key [▷](next–menu key)and press chapter selection soft key **[PITCH]**.

9 Press soft key **[(OPRT)]**.

10 Press the rightmost soft key[▷](next–menu key).

11 Press soft keys **[READ]** and **[EXEC]**.
Parameters are read into memory. Upon completion of input, the "INPUT" indicator at the lower–right corner of the screen disappears.

12 Press function key [OFFSET SETTING].

13 Press soft key **[SETING]** for chapter selection.

14 Enter 0 to "PARAMETER WRITE" in the setting data.

15 Turn the power to the CNC back on.

16 Release the EMERGENCY STOP button on the machine operator's panel.

Explanations

- **Pitch error compensation**

Parameters 3620 to 3624 and pitch error compensation data must be set correctly to apply pitch error compensation correctly
(See subsection 11.5.2)

8.5.4 Outputting Pitch Error Compensation Data

All pitch error compensation data are output in the defined format from the memory of the CNC to a floppy or NC tape.

Outputting Pitch Error Compensation Data

Procedure

1 Make sure the output device is ready for output.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key SYSTEM .

5 Press the rightmost soft key ▷ (next–menu key) and press chapter selection soft key **[PITCH]**.

6 Press soft key **[(OPRT)]**.

7 Press rightmost soft key ▷(next–menu key).

8 Press soft keys **[PUNCH]** and **[EXEC]**.
All parameters are output in the defined format.

Explanations

• Output format

Output format is as follows:
N 10000 . . . . . P . . . . . ;
N 11023 . . . . . P . . . . . ;
N:Pitch error compensation point No. +10000
P:Pitch error compensation data

• Output file name

When the floppy disk directory display function is used, the name of the output file is VPITCH ERRORW.

8.6 INPUTTING/OUTPUTTING CUSTOM MACRO COMMON VARIABLES

8.6.1 Inputting Custom Macro Common Variables

The value of a custom macro common variable (#500 to #531) is loaded into the memory of the CNC from a floppy or NC tape. The same format used to output custom macro common variables is used for input. See Section **8.6.2**. For a custom macro common variable to be valid, the input data must be executed by pressing the cycle start button after data is input. When the value of a common variable is loaded into memory, this value replaces the value of the same common variable already existing (if any) in memory.

Inputting custom macro common variables

Procedure

1 Register the program output in Section **8.6.2** in memory, according to the procedure given in Section **8.3.1**.

2 Press the MEMORY switch on the machine operator's panel upon completing input.

3 Press the cycle start button to execute the loaded program.

4 Display the macro vriable screen to chek whether the values of the common variables have been set correctly.

Display of the macro variable screen

·Press function key [OFFSET SETTING] .

· Press the rightmost soft key (next–menu key).
·Press soft key **[MACRO]**.
·Select a variable with the page keys or numeric keys and soft key **[NO.SRH]**.

Explanations

- **Common variables**

The common variables (#500 to #531) can be input and output.
Variables #100 to #149 can also be input and output when bit 3 (PV5) of parameter No. 6001 is set to 1.

8.6.2 Outputting Custom Macro Common Variable

Custom macro common variables (#500 to #531) stored in the memory of the CNC can be output in the defined format to a floppy or NC tape.

Outputting custom macro common variable

Procedure

1 Make sure the output device is ready for output.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key OFFSET SETTING.

5 Press the rightmost soft key ▷ (next–menu key), then press soft key **[MACRO]**.

6 Press soft key **[(OPRT)]**.

7 Press the rightmost soft key ▷ (next–menu key).

8 Press soft keys **[PUNCH]** and **[EXEC]**.
Common variables are output in the defined format.

Explanations

• Output format

The output format is as follows:

```
%
;
#500=[25283*65536+65536]/134217728 .... (1)
#501=#0; .........(2)
#502=0; ...........(3)
#503= ............. ;
.................... ;
.................... ;
#531= ............. ;
M02;
%
```

(1) The precision of a variable is maintained by outputting the value of the variable as <expression>.

(2) Undefined variable

(3) When the value of a variable is 0

• Output file name

When the floppy disk directory display function is used, the name of the output file is "**MACRO VAR**".

• Common variable

The common variables (#500 to #531) can be input and output.
Variables #100 to #149 can also be input and output when bit 3 (PV5) of parameter No. 6001 is set to 1.

8.7 DISPLAYING DIRECTORY OF FLOPPY DISK

On the floppy directory display screen, a directory of the FANUC Handy File, FANUC Floppy Cassette, or FANUC FA Card files can be displayed. In addition, those files can be loaded, output, and deleted.

```
DIRECTORY (FLOPPY)                       O0001 N00000
  NO.  FILE NAME                         (METER) VOL









 EDIT **** *** ***                    11 : 51 : 12
[ PRGRM ][        ][  DIR  ][        ][ (OPRT) ]
```

8.7.1 Displaying the Directory

Displaying the directory of floppy disk files

Procedure 1

Use the following procedure to display a directory of all the files stored in a floppy:

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press soft key **[EDIT]**.

4 Press soft key **[NC LNG]**.

5 Press soft key **[FLOPPY]**.

6 Press page key ↑PAGE or PAGE↓.

7 The screen below appears.

```
DIRECTORY (FLOPPY)                       O0001  N00000
 NO.   FILE NAME                         (METER) VOL
 0001    PARAMETER                          58.5
 0002    O0001                               1.9
 0003    O0002                               1.9
 0004    O0010                               1.3
 0005    O0040                               1.3
 0006    O0050                               1.9
 0007    O0100                               1.9
 0008    O1000                               1.9
 0009    O9500                               1.6

 EDIT ****   ***   ***              11 : 53 : 04
 ( F SRH )( READ )( PUNCH )( DELETE )(      )
```

Fig.8.7.1 (a)

8 Press a page key again to display another page of the directory.

Procedure 2

Use the following procedure to display a directory of files starting with a specified file number :

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press soft key **[EDIT]**.

4 Press soft key **[NC LNG]**.

5 Press soft key **[FLOPPY]**.

6 Press soft key **[(OPRT)]**.

7 Press soft key **[F SRH]**.

8 Enter a file number.

9 Press soft keys **[F SET]** and **[EXEC]**.

10 Press a page key to display another page of the directory.

11 Press soft key **[CAN]** to return to the soft key display shown in the screen of Fig 8.7.1(a).

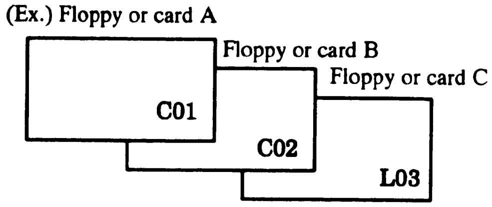
```
DIRECTORY (FLOPPY)                    O0001 N00000
 NO.   FILE NAME                      (METER) VOL




SEARCH
FILE NO. =
>_
EDIT * * * *   * * *   * * *          11 : 54 : 19
(F SET )(        )(        )( CAN )( EXEC)
```

Fig.8.7.1 (b)

Explanations

- **Screen fields and their meanings**

NO	: Displays the file number
FILE NAME	: Displays the file name.
(METER)	: Converts and prints out the file capacity to paper tape length.You can also produce H (FEET) I by setting the INPUT UNIT to INCH of the setting data.
VOL.	: When the file is multi–volume, that state is displayed.



C(number)means .... CONTINUE
L(number)means .... LAST
number .... number of floppies or cards

8.7.2 Reading Files

The contents of the specified file number are read to the memory of NC.

Reading files

Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press soft key **[EDIT]**.

4 Press soft key **[NC LNG]**.

5 Press soft key **[FLOPPY]**.

6 Press soft key **[(OPRT)]**.

7 Press soft key **[READ]**.

```
DIRECTORY (FLOPPY)                    O0001 N00000
 NO.   FILE NAME                      (METER) VOL




READ
 FILE NO. =                     PROGRAM NO. =
>_
EDIT  * * * *   * * *   * * *    11 : 55 : 04
[ F SET ][ O SET ][ STOP ][ CAN ][ EXEC ]
```

8 Enter a file number.

9 Press soft key **[F SET]**.

10 To modify the program number, enter the program number, then press soft key **[O SET]**.

11 Press soft key **[EXEC]**. The file number indicated in the lower–left corner of the screen is automatically incremented by one.

12 Press soft key **[CAN]** to return to the soft key display shown in the screen of Fig. 8.7.1.(a).

8.7.3 Outputting Programs

Any program in the memory of the CNC unit can be output to a floppy as a file.

Outputting programs

Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press soft key **[EDIT]**.

4 Press soft key **[NC LNG]**.

5 Press soft key **[FLOPPY]**.

6 Press soft key **[(OPRT)]**.

7 Press soft key **[PUNCH]**.

```
DIRECTORY (FLOPPY)                      O0002  N01000
 NO.   FILE NAME                        (METER) VOL





PUNCH
 FILE NO. =                        PROGRAM NO. =
>_
EDIT * * * *   * * *   * * *        11 : 55 : 26
( F SET  )( O SET  )( STOP  )(  CAN  )(  EXEC  )
```

8 Enter a program number. To write all programs into a single file, enter –9999 in the program number field. In this case, the file name "ALL.PROGRAM" is registered.

9 Press soft key **[O SET]**.

10 Press soft key **[EXEC]**. The program or programs specified in step 7 are written after the last file on the floppy. To output the program after deleting files starting with an existing file number, key in the file number, then press soft key **[F SET]** followed by soft key **[EXEC]**.

11 Press soft key **[CAN]** to return to the soft key display shown in the screen of Fig.8.7.1(a).

8.7.4 Deleting Files

The file with the specified file number is deleted.

Deleting files

Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press soft key **[EDIT]**.

4 Press soft key **[NC LNG]**.

5 Press soft key **[FLOPPY]**.

6 Press soft key **[(OPRT)]**.

7 Press soft key **[DELETE]**.

```
DIRECTORY (FLOPPY)                         O0001 N00000
 NO.   FILE NAME                           (METER) VOL





DELETE
 FILE NO. =       NAME=
>_
EDIT * * * *   * * *   * * *        11 : 55 : 51
[F SET  ][F NAME ][       ][  CAN  ][  EXEC  ]
```

8 Specify the file to be deleted.
When specifying the file with a file number, type the number and press soft key **[F SET]**. When specifying the file with a file name, type the name and press soft key **[F NAME]**.

9 Press soft key **[EXEC]**.
The file specified in the file number field is deleted. When a file is deleted, the file numbers after the deleted file are each decremented by one.

10 Press soft key **[CAN]** to return to the soft key display shown in the screen of Fig. **8.7.1(a)**.

Restrictions

- **Inputting file numbers and program numbers with keys**

If **[F SET]** or **[O SET]** is pressed without key inputting file number and program number, file number or program number shows blank. When 0 is entered for file numbers or program numbers, 1 is displayed.

• I/O devices

To use channel 0 ,set a device number in parameter 102.
Set the I/O device number to parameter No. 0112 when cannel 1 is used. Set it to No. 0122 when channel 2 is used.

• Significant digits

For the numeral input in the data input area with FILE NO. and PROGRAM NO., only lower 4 digits become valid.

• Collation

When the data protection key on the machine operator's panel is ON, no programs are read from the floppy. They are verified against the contents of the memory of the CNC instead.

ALARM

No.	Contents
71	An invalid file number or program number was entered. (Specified program number is not found.)
79	Verification operation found a mismatch between a program loaded into memory and the contents of the floppy
86	The dataset–ready signal (DR) for the input/output device is turned off. (The no file error or duplicate file error occurred on the input/output device because an invalid file number, program number, or file name was entered.

9 EDITING PROGRAMS

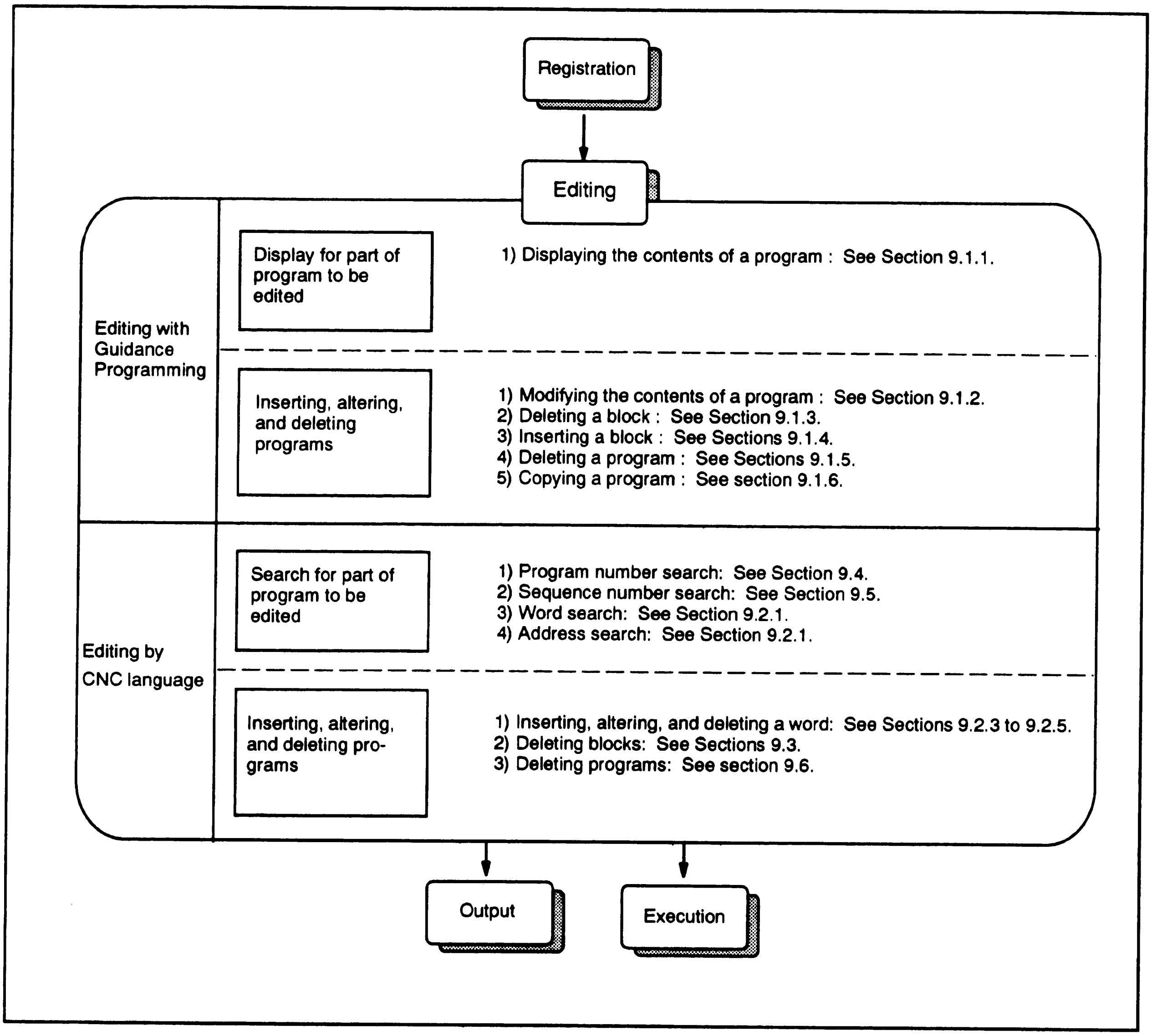
This chapter describes how to edit a program registered in the CNC. Programs can be edited using either of the following two methods:

- Editing by guidance programming
- Editing by the CNC language

1. Editing by guidance programming enables the modification of a program and the deletion and insertion of a block according to guidance, as well as the deletion and copying of the entire program.

2. Editing by the CNC language enables the insertion, modification, and deletion of a word, as well as the deletion of the entire program. This chapter also describes searching for a program number, sequence number, word, or address, prior to program editing.

The program editing function can be used when the guidance programming function option is supported.


Registration
Editing
Editing with Guidance Programming
Display for part of program to be edited
1) Displaying the contents of a program : See Section 9.1.1.
Inserting, altering, and deleting programs
1) Modifying the contents of a program : See Section 9.1.2.
2) Deleting a block : See Section 9.1.3.
3) Inserting a block : See Sections 9.1.4.
4) Deleting a program : See Sections 9.1.5.
5) Copying a program : See section 9.1.6.
Editing by CNC language
Search for part of program to be edited
1) Program number search: See Section 9.4.
2) Sequence number search: See Section 9.5.
3) Word search: See Section 9.2.1.
4) Address search: See Section 9.2.1.
Inserting, altering, and deleting programs
1) Inserting, altering, and deleting a word: See Sections 9.2.3 to 9.2.5.
2) Deleting blocks: See Sections 9.3.
3) Deleting programs: See section 9.6.
Output
Execution

9.1 EDITING WITH GUIDANCE PROGRAMMING

The contents of a registered program can be modified, deleted, or inserted into blocks by means of guidance programming.

9.1.1 Displaying the contents of a program

The contents of a program can be displayed by following the procedure below

Procedure

1 Select **EDIT** mode.

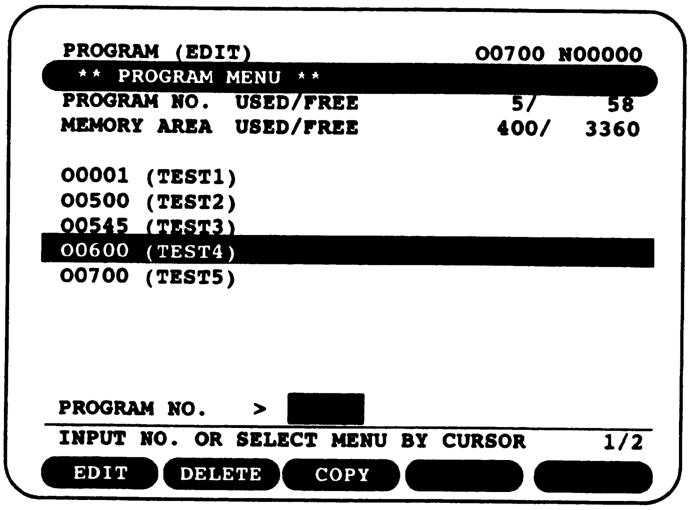
2 Press the PROG function key or **[PRGRM]** function selection soft key.

3 Press the ↑ or ↓ key to position the cursor to the number of the program to be edited.
Specify the number of the desired program by applying either of the following two methods:

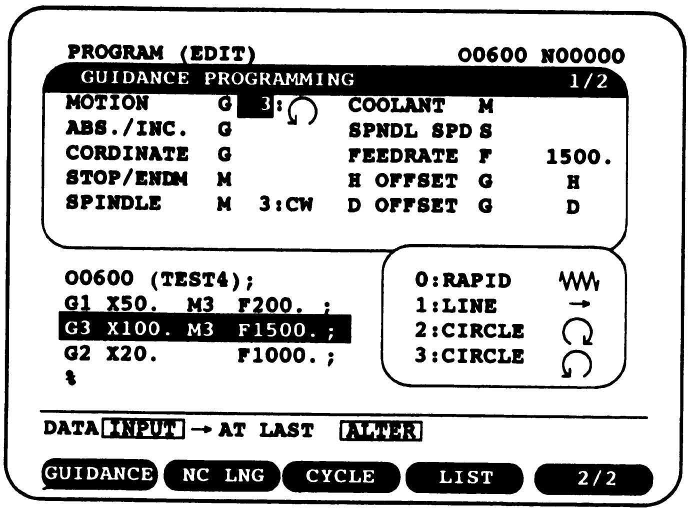
a. Using the numeric keypad, key in the desired program number.

```
PROGRAM (EDIT)                          O0700 N00000
 ** PROGRAM MENU **
PROGRAM NO.   USED/FREE                    5/    58
MEMORY AREA   USED/FREE                  400/  3360

O0001 (TEST1)
O0500 (TEST2)
O0545 (TEST3)
O0600 (TEST4)
O0700 (TEST5)



PROGRAM NO.    >  600
INPUT NO. OR SELECT MENU BY CURSOR           1/2
 EDIT    DELETE    COPY
```

b.Position the cursor to the desired program number.



4 Press the **[EDIT]** soft key.
The contents of the block selected by positioning the cursor are displayed.
For a block created by guidance programming, the program data is displayed on the guidance programming screen. For a block created on the cycle screen, the program data is displayed on the cycle screen.



5 To move the displayed block, press page key [PAGE ↑] or [PAGE ↓]

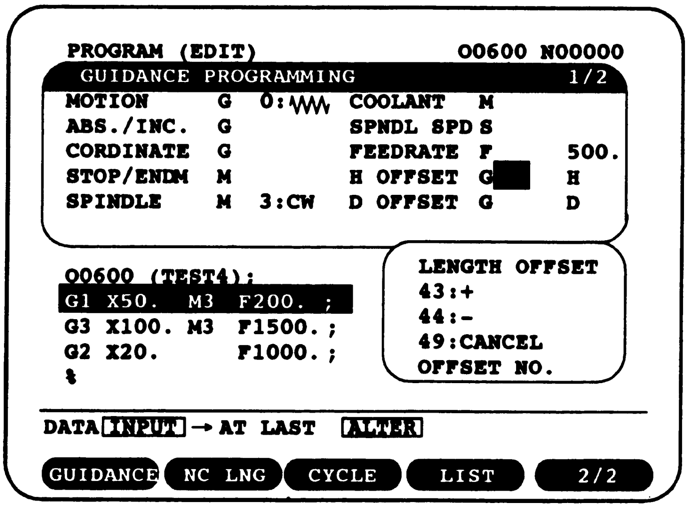
9.1.2 Modifying the contents of a program

The contents of a program can be modified by following the procedure below

Procedure

1 Press the [↑ PAGE] or [PAGE ↓] key to position the cursor to the block to be modified.

2 On the guidance programming screen, the contents of the block selected by positioning the cursor are displayed. Following the guidance given on the guidance programming screen, modify the data. For details of the data input method, see Section 10.1.4.



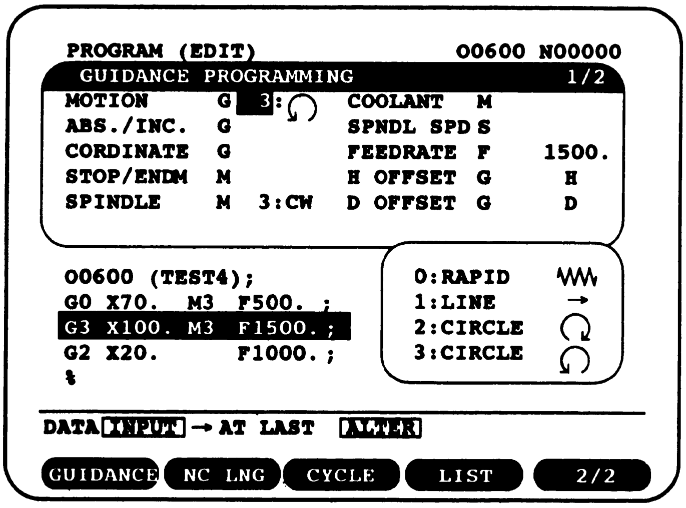
3 Press the [ALTER] key. The block is modified according to the input data.

9.1.3 Deleting a block

A block is deleted by following the procedure below

Procedure

1 Press the [↑ PAGE] or [PAGE ↓] key to position the cursor to the block to be deleted.



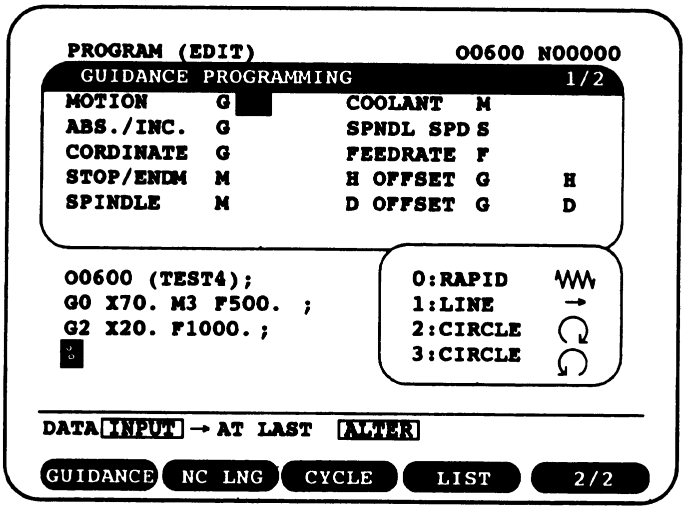
2 Press the [DELETE] key. The selected block is deleted.

9.1.4 Inserting a block

A block is inserted by following the procedure below

Procedure

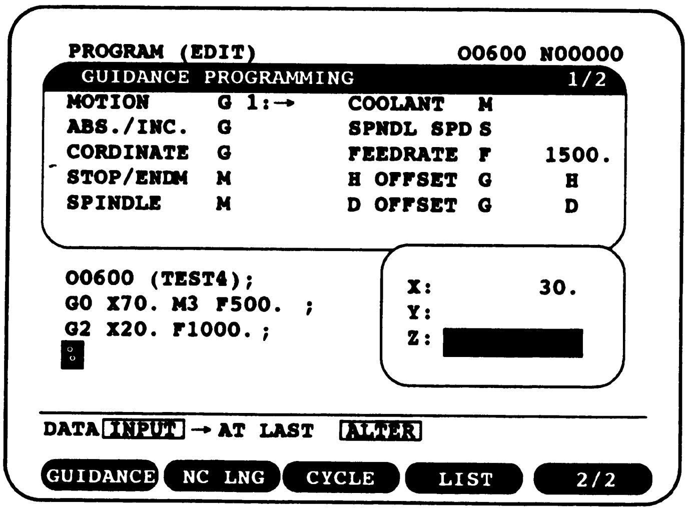
1 Press the [PAGE ↑] or [PAGE ↓] key to position the cursor to the last block prior to the position where the block is to be inserted.



2 Press the **[GUIDANCE]** soft key. The entire contents of the programming screen are initialized and the data of the block to be inserted can be entered.

3 Following the guidance displayed on the guidance programming screen, input the data of the block to be inserted.
For details of the data input method, see Section 10.1.4.

4 Press the [INSERT] key. The block is inserted.



9.1.5 Deleting a program

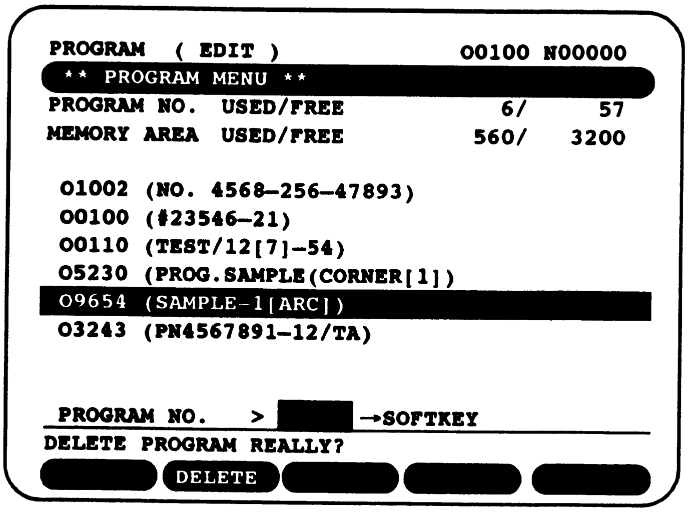
A program is deleted by following the procedure below

Procedure

1 Display the program menu screen.

2 Using the numeric keypad, enter the number of the program to be deleted. Alternatively, position the cursor to the desired program number.

3 Press the **[DELETE]** soft key.
Check that the data protect key on the machine operator's panel is released.
The message "DELETE PROGRAM REALLY?" is displayed.



4 To delete the program, press the **[DELETE]** soft key again. To abandon the deletion, press the leftmost soft key or the RESET key. The screen displayed in 1, above, reappears.

9.1.6 Copying a program

The program copy function is provided to enable part of a existing program to be modified to create another program.

A program is copied by following the procedure below

Procedure

1 Press the rightmost soft key. The **[COPY]** soft key is displayed.

2 Using the numeric keypad, enter the number of the program to be copied. Alternatively, position the cursor to the desired program number.

3 Press the **[COPY]** soft key.
Check that the data protect key on the machine operator's panel is released.
The message "INPUT NEW PROGRAM NO. FOR COPY" is displayed.

4 Using the numeric keypad, enter the new program number.

```
PROGRAM  ( EDIT )                      O0100 N00000
 ** PROGRAM MENU **
PROGRAM NO.  USED/FREE                    6/     57
MEMORY AREA  USED/FREE                  560/   3200

 O1002 (NO. 4568–256–47893)
 O0100 (#23546–21)
 O0110 (TEST/12[7]–54)
 O5230 (PROG.SAMPLE(CORNER[1])
 O9654 (SAMPLE–1[ARC])
 O3243 (PN4567891–12/TA)

 PROGRAM NO.   > 2300  →SOFTKEY
INPUT NEW PROGRAM NO. FOR COPY
                  COPY
```

5 Press the **[COPY]** soft key once again. The program is registered with the new program number. If the specified new program number has already been registered using the program menu screen, the message "PROGRAM NUMBER ALREADY EXISTS" is displayed. The program cannot be copied.

9.1.7 Renaming a program

A program name, registered using the program menu screen, can be changed.

A program is renamed by following the procedure below

Procedure

1 Display the program menu screen.

2 Using the numeric keypad, enter the number of the program to be renamed. Alternatively, position the cursor to the desired program number.

3 Press the **[EDIT]** soft key.

4 Press the rightmost soft key. The **[NAME]** soft key is displayed. Note that the **[NAME]** soft key is displayed only when the memory protect switch is released.

5 Using the address and numeric keys, enter a new program name of up to 23 characters.
Do not include the EOB (;) in a program name.

6 Press the **[NAME]** soft key. The program is renamed.

9.1.8 Editing in CNC language

When the **[NC LNG]** soft key is pressed on the guidance programming screen, the CNC language editing screen is displayed. Using this screen, programs can be edited in CNC language.

Editing in the CNC language is done by following the procedure below

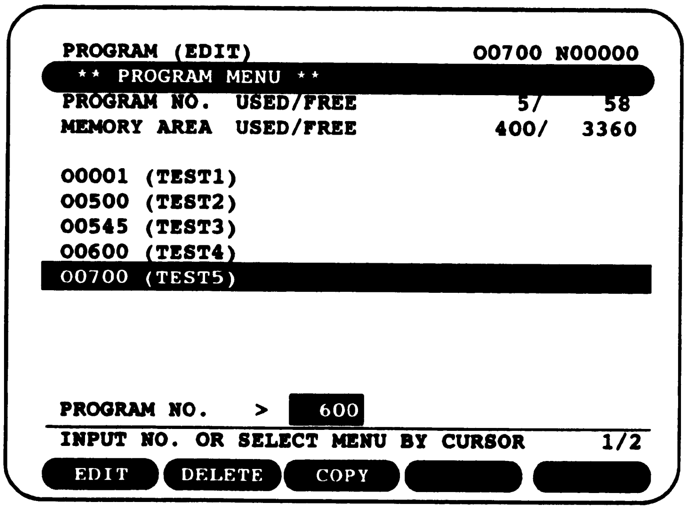
Procedure

1 Select **EDIT** mode.

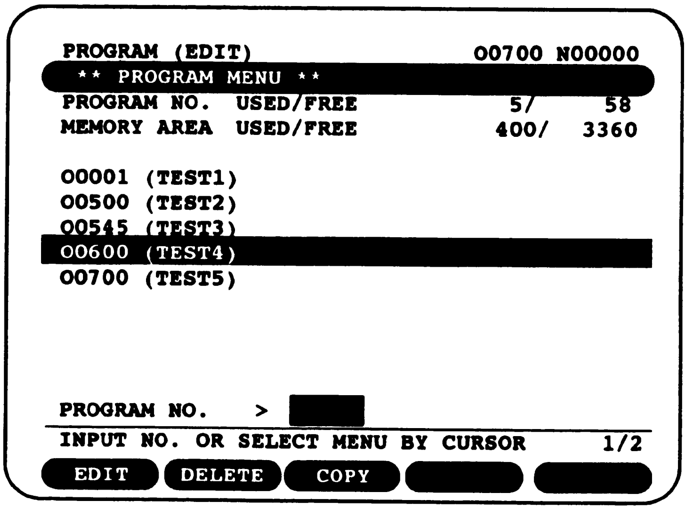
2 Press the [PROG] function key or the **[PRGRM]** soft key.

3 Press the [↑] or [↓] key to position the cursor to the number of the program to be edited.
Specify the desired program number by means of either of the following two methods:

a. Using the numeric keypad, enter the desired program number.



b. Position the cursor to the desired program number.



4 Press the **[EDIT]** soft key.

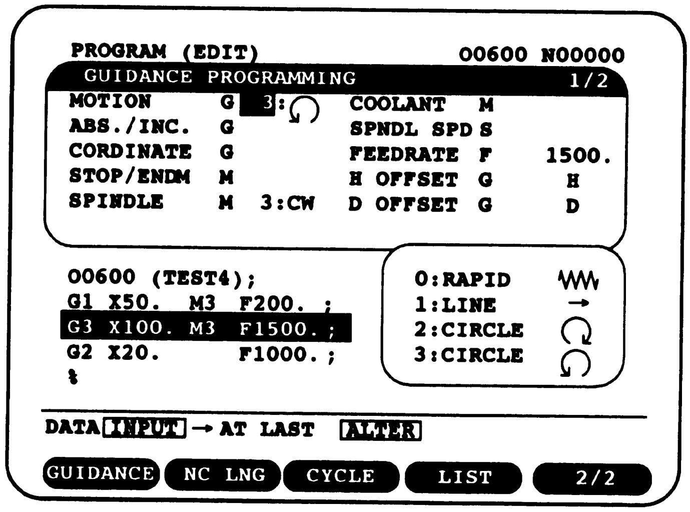
5 The guidance programming screen or CNC language screen is displayed. The screen displayed depends on the following this parameter:

Bit 0 of Parameter 9320 (NCE)

0 : Displays the guidance programming screen after a program number is registered.

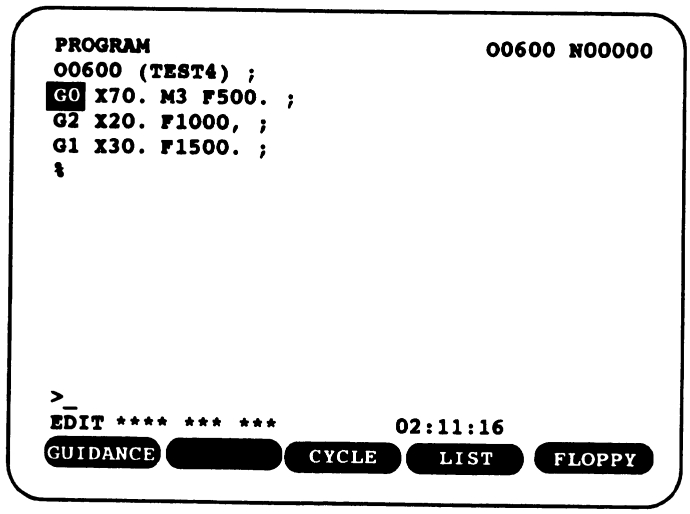
1 : Displays the CNC language screen after a program number is registered.

If the NCE bit is set to 0, the following screen is displayed :



To display the CNC language screen, press the **[NC LNG]** soft key.

If the NCE bit is set to 1, the CNC language screen is displayed as shown below:



To display the guidance programming screen, press the **[GUIDANCE]** soft key.

6 On the CNC language screen, edit a program, as explained in Sections 9.2 to 9.7.

9.2 INSERTING , ALTERING AND DELETING A WORD

This section outlines the procedure for inserting, modifying, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

1 Select **EDIT** mode.

2 Display the CNC language screen according to the procedure for editing by the CNC language as described in Section 9.1.8.

3 Search for a word to be modified.
·Scan method
·Word search method

4 Perform an operation such as altering, inserting, or deleting a word.

Explanation

- **Concept of word and editing unit**

A word is an address followed by a number. With a custom macro, the concept of word is ambiguous.
So the editing unit is considered here.
The editing unit is a unit subject to alteration or deletion in one operation.
In one scan operation, the cursor indicates the start of an editing unit.
An insertion is made after an editing unit.
Definition of editing unit
(i) Program portion from an address to immediately before the next address
(ii) An address is an alphabet, **IF, WHILE, GOTO, END, DO=**,or ; **(EOB)**.
According to this definition, a word is an editing unit.
The word "word," when used in the description of editing, means an editing unit according to the precise definition.

> **Note**
> The user cannot continue program execution after altering, inserting, or deleting data of the program by suspending machining in progress by means of an operation such as a single block stop or feed hold operation during program execution. If such a modification is made, the program may not be executed exactly according to the contents of the program displayed on the screen after machining is resumed. So, when the contents of memory are to be modified by part program editing, be sure to enter the reset state or reset the system upon completion of editing before executing the program.

9.2.1 Word Search

A word can be searched for by merely moving the cursor through the text (scanning), by word search, or by address search.

Procedure for scanning a program

1 Press the cursor key [→]

The cursor moves forward word by word on the screen; the cursor is displayed at a selected word.

2 Press the cursor key [←]

The cursor moves backward word by word on the screen; the cursor is displayed at a selected word.

Example) When Z1250.0 is scanned

```
Program                              O0050 N01234
 O0050 ;
 N01234 X100.0 Z1250.0 ;
 S12 ;
 N56789 M03 ;
 M02 ;
 %
```

3 Holding down the cursor key [→] or [←] scans words continuously.

4 The first word of the next block is searched for when the cursor key [↓] is pressed.

5 The first word of the previous block is searched for when the cursor key [↑] is pressed.

6 Holding down the cursor key [↓] or [↑] moves the cursor to the head of a block continuously.

7 Pressing the page key [PAGE ↓] displays the next page and searches for the first word of the page.

8 Pressing the page key [↑ PAGE] displays the previous page and searches for the first word of the page.

9 Holding down the page key [PAGE ↓] or [↑ PAGE] displays one page after another.

Procedure for searching a word

Example) of Searching for S12

```
PROGRAM                    O0050 N01234
O0050 ;
N01234 X100.0 Z1250.0 ; <-------------- N01234 is being searched for/scanned currently.
S12 ; <-------------------------------- S12 is searched for.
N56789 M03 ;
M02 ;
%
```

1 Key in address S .

2 Key in 1 2 .
·S12 cannot be searched for if only S1 is keyed in.
·S09 cannot be searched for by keying in only S9.
To search for S09, be sure to key in S09.

3 Pressing the **[SRH↓]** key starts search operation.
Upon completion of search operation, the cursor is displayed at S12.
Pressing the **[SRH↑]** key rather than the **[SRH↓]** key performs search operation in the reverse direction.

Procedure for searching an address

Example) of Searching for M03

```
PROGRAM                    O0050 N01234
O0050 ;
N01234 X100.0 Z1250.0 ; <-------------- N01234 is being searched for/scanned currently.
S12 ;
N56789 M03 ; <------------------------- M03 is searched for.
M02 ;
%
```

1 Key in address M .

2 Press the **[SRH↓]** key.
Upon completion of search operation, the cursor is displayed at M03.
Pressing the **[SRH↑]** key rather than the **[SRH↓]** key performs search operation in the reverse direction.

Alarm

Alarm number	Description
71	The word or address being searched for was not found.

9.2.2 Heading a Program

The cursor can be jumped to the top of a program. This function is called heading the program pointer. This section describes the three methods for heading the program pointer.

Procedure for Heading a Program

Method 1

1 Press RESET when the program screen is selected in EDIT mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2

Search for the program number.

1 Press address O ,when a program screen is selected in the **MEMORY** or **EDIT** mode.
2 Input a program number.
3 Press the soft key **[O SRH]**.

Method 3

1 Select **[MEMORY]** or **[EDIT]** mode.
2 Press PROG .
3 Press the **[(OPRT)]** key.
4 Press the **[REWIND]** key.

9.2.3 Inserting a Word

Procedure for inserting a word

1 Search for or scan the word immediately before a word to be inserted.

2 Key in an address to be inserted.

3 Key in data.

4 Press the [INSERT] key.

Example of Inserting T15

Procedure

1 Search for or scan Z1250.

```
Program                              O0050  N01234
 O0050 ;
 N01234 X100.0 Z1250.0 ;   <-------- Z1250.0 is searched for/scanned.
 S12 ;
 N56789 M03 ;
 M02 ;
 %
```

2 Key in [T] [1] [5] .

3 Press the [INSERT] key.

```
Program                              O0050  N01234
 O0050 ;
 N01234 X100.0 Z1250.0 T15 ;   <---- T15 is inserted.
 S12 ;
 N56789 M03 ;
 M02 ;
 %
```

9.2.4 Altering a Word

Procedure for altering a word

1 Search for or scan a word to be altered.

2 Key in an address to be inserted.

3 Key in data.

4 Press the [ALTER] key.

Example of changing T15 to M15

Procedure

1 Search for or scan T15.

```
Program                          O0050 N01234
 O0050 ;
 N01234 X100.0 Z1250.0 T15 ;
 S12 ;
 N56789 M03 ;
 M02 ;
 %
```

T15 is searched for/ scanned.

2 Key in [M] [1] [5] .

3 Press the [ALTER] key.

```
Program                          O0050 N01234
 O0050 ;
 N1234 X100.0 Z1250.0 M15 ;
 S12 ;
 N5678 M03 ;
 M02 ;
 %
```

T15 is changed to M15.

9.2.5 Deleting a Word

Procedure for deleting a word

1 Search for or scan a word to be deleted.

2 Press the DELETE key.

Example of deleting X100.0

Procedure

1 Search for or scan X100.0.

```
Program                                O0050  N01234
 O0050 ;
 N01234 X100.0  Z1250.0  M15 ;   <----  X100.0 is searched for/scanned.
 S12 ;
 N56789  M03 ;
 M02 ;
 %
```

2 Press the DELETE key.

```
Program                                O0050  N01234
 O0050 ;
 N01234  Z1250.0  M15 ;   <----  X100.0 is deleted.
 S12 ;
 N56789  M03 ;
 M02 ;
 %
```

9.3 DELETING BLOCKS

A block or blocks can be deleted in a program.

9.3.1 Deleting a Block

The procedure below deletes a block up to its EOB code; the cursor advances to the address of the next word.

Procedure for deleting a block

1 Search for or scan address N for a block to be deleted.

2 Key in [EOB].

3 Press the [DELETE].

Example of deleting a block of No. 01234

Procedure

1 Search for or scan N01234.

```
Program                        O0050 N01234
O0050 ;
N01234 Z1250.0 M15 ;
S12 ;
N56789 M03 ;
M02 ;
%
```

N01234 is searched for/scanned.

2 Key in [EOB].

3 Press the [DELETE] key.

```
Program                        O0050 N01234
O0050 ;
S12 ;
N56789 M03 ;
M02 ;
%
```

Block containing N01234 has been deleted.

9.3.2 Deleting Multiple Blocks

The blocks from the currently displayed word to the block with a specified sequence number can be deleted.

Procedure for deleting multiple blocks

1 Search for or scan a word in the first block of a portion to be deleted.

2 Key in address N .

3 Key in the sequence number for the last block of the portion to be deleted.

4 Press the DELETE key.

Example of deleting blocks from a block containing N01234 to a block containing N56789

Procedure

1 Search for or scan N01234.

```
Program                              O0050  N01234
 O0050 ;
 N01234  Z1250.0  M15 ;
 S12 ;
 N56789  M03 ;
 M02 ;
 %
```

N01234 is searched for/ scanned.

2 . Key in N 5 6 7 8 9 .

```
Program                              O0050  N01234
 O0050 ;
 N01234  Z1250.0  M15 ;
 S12 ;
 N56789  M03 ;
 M02 ;
 %
```

Underlined part is deleted.

3 Press the DELETE key.

```
Program                              O0050  N01234
 O0050 ;
 M02 ;
 %
```

Blocks from block containing N01234 to block containing N56789 have been deleted.

9.4 PROGRAM NUMBER SEARCH

When memory holds multiple programs, a program can be searched for. There are three methods as follows.

Procedure for program number search

Method 1

1 Select **EDIT** or **MEMORY** mode.

2 Press PROG to display the program screen.

3 Key in address O .

4 Key in a program number to be searched for.

5 Press the **[O SRH]** key.

6 Upon completion of search operation, the program number searched for is displayed in the upper–right corner of the CRT screen.
If the program is not found , P/S alarm No. 71 occurs.

Method 2

1 Select **EDIT** or **MEMORY** mode.

2 Press PROG to display the program screen.

3 Press the **[O SRH]** key.
In this case, the next program in the directory is searched for .

Method 3

This method searches for the program number (0001 to 0015) corresponding to a signal on the machine tool side to start automatic operation. Refer to the relevant manual prepared by the machine tool builder for detailed information on operation.

1 Select **MEMORY** mode.

2 Set the reset state(*1)

- The reset state is the state where the LED for indicating that automatic operation is in progress is off. (Refer to the relevant manual of the machine tool builder.)

3 Set the program number selection signal on the machine tool side to a number from 01 to 15.

- If the program corresponding to a signal on the machine tool side is not registered, P/S alarm (No. 059) is raised.

4 Press the cycle start button.

- When the signal on the machine tool side represents 00, program number search operation is not performed.

Alarm

No.	Contents
59	The program with the selected number cannot be searched during external program number search.
71	The specified program number was not found during program number search.

9.5 SEQUENCE NUMBER SEARCH

Sequence number search operation is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number.

Example) Sequence number 02346 in a program (O0002) is searched for.

```
                               Program
                               O0001 ;
                               N01234 X100.0 Z100.0 ;
                               S12 ;
                                :
Selected program  ──►          O0002 ;                    ⎫ This section is
                               N02345 X20.0 Z20.0 ;       ⎬ searched starting at
Target sequence   ──►          N02346 X10.0 Y10.0 ;       ⎭ the beginning.
number is found.                :                           (Search operation is
                               O0003 ;                      performed only within a
                                :                           program.)
```

Procedure for sequence number search

1 Select **MEMORY** mode.

2 Press PROG .

3 · If the program contains a sequence number to be searchedfor, perform the operations 4 to 7 below.

· If the program does not contain a sequence number to be searched for,select the program number of the program that contains the sequence number to be searched for.

4 Key in address N .

5 Key in a sequence number to be searched for.

6 Press the [N SRH] key.

7 Upon completion of search operation, the sequence number searched for is displayed in the upper–right corner of the CRT screen. If the specified sequence number is not found in the program currently selected, P/S alarm (No, 060) occurs.

Explanations

• Operation during Search

Those blocks that are skipped do not affect the CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values.
So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify M, S, and T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and CNC states at that point.

• Checking during search

During search operation, the following checks are made:
·Optional block skip

Limitations

• Searching in sub–program

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm (No.060) is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.

```
Main program          Subprogram
O1234                 O5678
  :                     :
  :                   N88888
M98 P5678 ;             :
  :                   M99 ;
  :                     :
```

If an attempt is made to search for N8888 in the example above, an alarm is raised.

Alarm

Number	Contents
60	Command sequence number was not found in the sequence number search.

9.6 DELETING PROGRAMS

Programs registered in memory can be deleted,either one program by one program or all at once. Also, More than one program can be deleted by specifying a range.

9.6.1 Deleting One Program

A program registered in memory can be deleted.

Procedure for deleting one program

1 Select the **EDIT** mode.

2 Press [PROG] to display the program screen.

3 Key in address [O].

4 Key in a desired program number.

5 Press the [DELETE] key.
The program with the entered program number is deleted.

9.6.2 Deleting All Programs

All programs registered in memory can be deleted.

Procedure for deleting all programs

1 Select the **EDIT** mode.

2 Press [PROG] to display the program screen.

3 Key in address [O].

4 Key in –9999.

5 Press edit key [DELETE] to delete all programs.

9.6.3 Deleting More Than One Program by Specifying a Range

Programs within a specified range in memory are deleted.

Procedure for deleting more than one program by specifying a range

1 Select the **EDIT** mode.

2 Press PROG to display the program screen.

3 Enter the range of program numbers to be deleted with address and numeric keys in the following format:
OXXXX,OYYYY
where XXXX is the starting number of the programs to be deleted and YYYY is the ending number of the programs to be deleted.

4 Press edit key DELETE to delete programs No. XXXX to No. YYYY.

9.7 EDITING OF CUSTOM MACROS

Unlike ordinary programs, custom macro programs are modified, inserted, or deleted based on editing units.
Custom macro words can be entered in abbreviated form.
Comments can be entered in a program.
Refer to the section 10.1 for the comments of a program.

Explanations

• Editing unit

When editing a custom macro already entered, the user can move the cursor to each editing unit that starts with any of the following characters and symbols:

(a) Address
(b) # located at the start of the left side of a substitution statement
(c) /, (,=, and ;
(d) First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT and PCLOS

On the CRT screen, a blank is placed before each of the above characters and symbols.
(Example) Head positions where the cursor is placed

```
N001 X–#100 ;
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X–SQRT[#3/3*[#4+1]] ;
N004 X–#2 Z#1 ;
N005 #5 =1+2–#10 ;
IF[#1NE0] GOTO10 ;
WHILE[#2LE5] DO1 ;
#[200+#2] =#2*10 ;
#2 =#2+1 ;
END1 ;
```

• Abbreviations of custom macro word

When a custom macro word is altered or inserted, the first two characters or more can replace the entire word.
Namely,

WHILE → WH	**GOTO → GO**	**XOR → XO**	**AND → AN**
SIN → SI	**COS → CO**	**TAN → TA**	**ATAN → AT**
SQRT → SQ	**ABS → AB**	**BCD → BC**	**BIN → BI**
FIX → FI	**FUP → FU**	**ROUND → RO**	**END → EN**
POPEN → PO	**BPRNT → BP**	**DPRNT → DP**	**PCLOS→PC**

(Example) Keying in
WH [AB [#2] LE RO [#3]]
has the same effect as
WHILE [ABS [#2] LE ROUND [#3]]
The program is also displayed in this way.

9.8 BACKGROUND EDITING

Editing a program while executing another program is called background editing. The method of editing is the same as for ordinary editing (foreground editing).
A program edited in the background should be registered in foreground program memory by performing the following operation:
During background editing, all programs cannot be deleted at once.

Procedure for background editing

1 Enter **MEMORY** mode.
Memory mode is allowed even while the program is being executed.

2 Press function key PROG .

3 Press the **[(OPRT)]** soft key to display the **[BG–EDT]** soft key.

4 Press the **[BG–EDT]** soft key.
The program list screen for background editing appears ("PROGRAM (BG–EDIT)" is displayed at the top left).

5 Edit a program using the background editing screen, in the same way as for ordinary editing.
The program editing procedure is the same in both the background and foreground. The number of the program being executed is dimmed on the screen and the cursor cannot be positioned to that number.

6 Once you have completed editing, press the **[BG–END]** soft key. The edited program is registered into the program memory for the foreground.

Explanation

- **Alarms during background editing**

Alarms that may occur during background editing do not affect foreground operation. Conversely, alarms that may occur during foreground operation do not affect background editing. In background editing, if an attempt is made to edit a program selected for foreground operation, a BP/S alarm (No. 140) is raised. On the other hand, if an attempt is made to select a program subjected to background editing during foreground operation (by means of subprogram calling or program number search operation using an external signal), a P/S alarm (Nos. 059, 078) is raised in foreground operation. As with foreground program editing, P/S alarms occur in background editing. However, to distinguish these alarms from foreground alarms, BP/S is displayed in the data input line on the background editing screen.

9.9 PASSWORD FUNCTION

The password function (bit 4 (NE9) of parameter No. 3202) can be locked using parameter No. 3210 (PASSWD) and parameter No. 3211 (KEYWD) to protect program Nos. 9000 to 9999. In the locked state, parameter NE9 cannot be set to 0. In this state, program Nos. 9000 to 9999 cannot be modified unless the correct keyword is set.
A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

Procedure for locking and unlocking

Locking

1 Set the MDI mode.

2 Enable parameter writing. At this time, P/S alarm No. 100 is issued on the CNC.

3 Set parameter No. 3210 (PASSWD). At this time, the locked state is set.

4 Disable parameter writing.

5 Press the RESET key to release the alarm state.

Unlocking

1 Set the MDI mode.

2 Enable parameter writing. At this time, P/S alarm No. 100 is issued on the CNC.

3 In parameter No. 3211 (KEYWD), set the same value as set in parameter No. 3210 (PASSWD) for locking. At this time, the locked state is released.

4 Set bit 4 (NE9) of parameter No. 3202 to 0.

5 Disable parameter writing.

6 Press the RESET key to release the alarm state.

7 Subprograms from program Nos. 9000 to 9999 can now be edited.

Explanations

- **Setting parameter PASSWD**

The locked state is set when a value is set in the parameter PASSWD. However, note that parameter PASSWD can be set only when the locked state is not set (when PASSWD = 0, or PASSWD = KEYWD). If an attempt is made to set parameter PASSWD in other cases, a warning is given to indicate that writing is disabled. When the locked state is set (when PASSWD = 0 and PASSWD = KEYWD), parameter NE9 is automatically set to 1. If an attempt is made to set NE9 to 0, a warning is given to indicate that writing is disabled.

- **Changing parameter PASSWD**

Parameter PASSWD can be changed when the locked state is released (when PASSWD = 0, or PASSWD = KEYWD). After step 3 in the procedure for unlocking, a new value can be set in the parameter PASSWD. From that time on, this new value must be set in parameter KEYWD to release the locked state.

- **Setting 0 in parameter PASSWD**

When 0 is set in the parameter PASSWD, the number 0 is displayed, and the password function is disabled. In other words, the password function can be disabled by either not setting parameter PASSWD at all, or by setting 0 in parameter PASSWD after step 3 of the procedure for unlocking. To ensure that the locked state is not entered, care must be taken not to set a value other than 0 in parameter PASSWD.

- **Re–locking**

After the locked state has been released, it can be set again by setting a different value in parameter PASSWD, or by turning the power to the NC off then on again to reset parameter KEYWD.

> **Note**
>
> Once the locked state is set, parameter NE9 cannot be set to 0 and parameter PASSWD cannot be changed until the locked state is released or the memory all–clear operation is performed. Special care must be taken in setting parameter PASSWD.

10 CREATING PROGRAMS

Programs can be created using any of the following methods:

- MDI keyboard
- PROGRAMMING IN TEACH IN MODE
- AUTOMATIC PROGRAM PREPARATION DEVICE (FANUC SYSTEM P)

This chapter describes creating programs using the MDI panel and Teach IN mode. This chapter also describes the automatic insertion of sequence numbers.

10.1 CREATING A PROGRAM WITH GUIDANCE PROGRAMMING

This section describes how to create and register a program by means of the optional guidance programming function.

10.1.1 Program menu screen

Procedure for displaying the program menu screen

Procedure

1 Select **EDIT** mode.

2 Press the PROG function key or the **[PRGRM]** soft key. The program menu screen is displayed.

```
PROGRAM  ( EDIT )                          O9654 N00000
 ** PROGRAM MENU **
PROGRAM NO.  USED/FREE                        5/     58
MEMORY AREA  USED/FREE                      480/   3280

 O1002 (NO. 4568-256-47893)
 O0100 (#23546-21)
 O0110 (TEST/12[7]-54)
 O9654 (SAMPLE-1[ARC])
 O3243 (PN4567891-12/TA)


 PROGRAM NO.   >
INPUT NO. OR SELECT MENU BY CURSOR                 1/2
 EDIT    DELETE   COPY
```

```
PROGRAM  ( EDIT )                          O9654 N00000
 ** PROGRAM MENU **
PROGRAM NO.  USED/FREE                        5/     58
MEMORY AREA  USED/FREE                      480/   3280

 O1002 (NO. 4568-256-47893)
 O0100 (#23546-21)
 O0110 (TEST/12[7]-54)
 O9654 (SAMPLE-1[ARC])
 O3243 (PN4567891-12/TA)


 PROGRAM NO.   >
INPUT NO. OR SELECT MENU BY CURSOR                 1/2
 PUNCH   READ    NC-OUT           COMENT
```

The program menu screen displays seven lines. These lines can be scrolled up and down by pressing the [↑] or [↓] key.

From the program menu screen, the following operations can be selected by pressing the corresponding soft keys. If the rightmost soft key is pressed, the subsequent page is displayed.

Soft key	Page	Function
EDIT	1/2	Registers and edits a program.
DELETE		Deletes a program.
COPY		Copies a program.
PUNCH	2/2	Outputs a program, through the reader/punch interface, to an external memory unit.
READ		Reads a program, through the reader/punch interface, from an external memory unit and registers that program in a part program storage area.
NC OUT	2/2	Converts a program to NC format
COMENT	2/2	Selects whether to display a program name.

Description

- **For an explanation of the use of the [DELETE] and [COPY] soft keys**

  See Chapter 9.

- **For an explanation of the use of the [PUNCH] and [READ] soft keys**

  See Section 8.4.

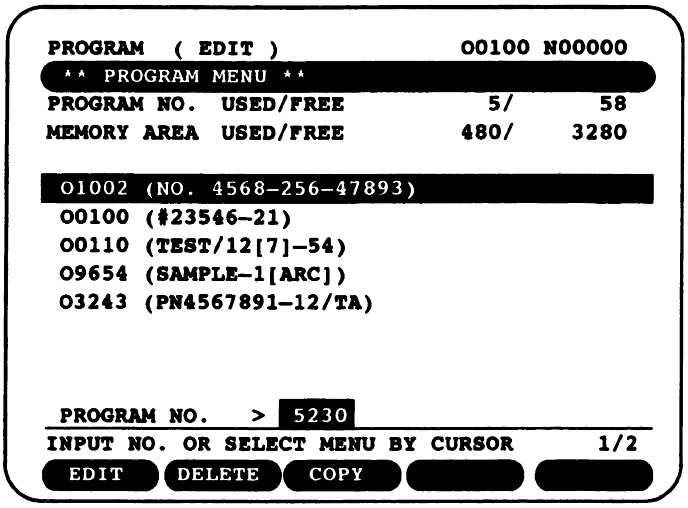
10.1.2 Registering a program

When the program menu screen is displayed, a guidance message prompting the operator to input a program number is displayed.

Procedure for registering a program

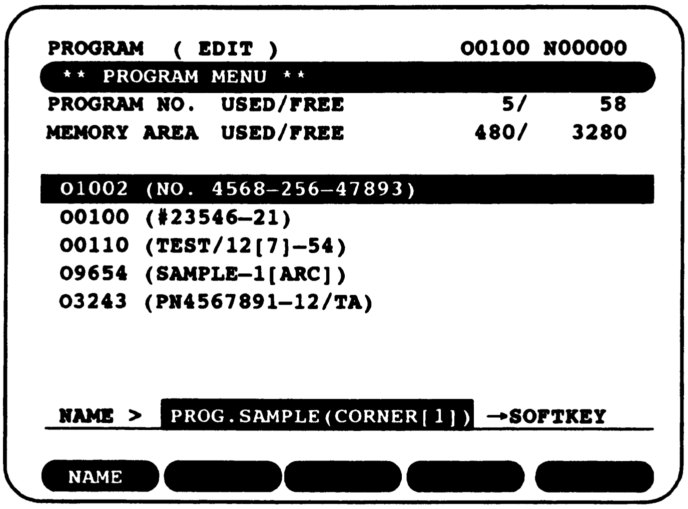
Procedure

1 Using the numeric keypad, enter a program number of up to four digits.
Specify a number which is not yet registered.



2 Press the **[EDIT]** soft key.
Check that the data protect key on the operator's panel is released.

3 The **[NAME]** soft key is displayed and the guidance message for entering a program name is displayed.
Using the address keys and numeric keypad, enter a desired program name of up to 23 characters. Then, press the **[NAME]** soft key.
Do not include the EOB (;) in the program name.



If a name does not have to be registered, press the **[NAME]** soft key directly.

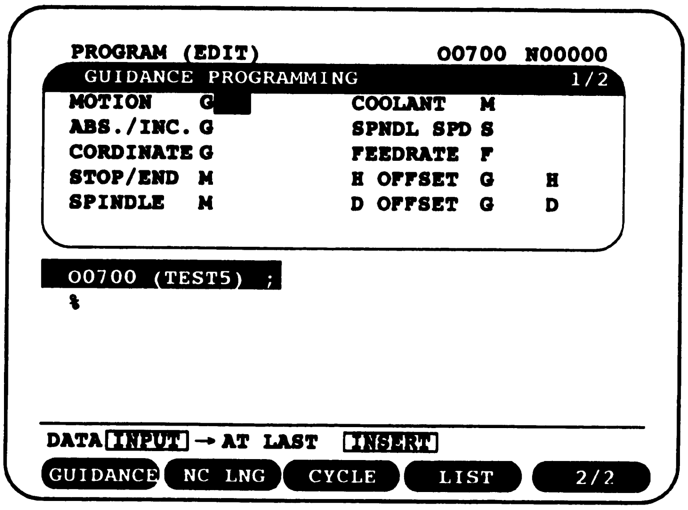
4 The guidance programming screen or CNC language screen is displayed. The screen to be displayed depends on the setting of the following parameter:

Bit 0 of Parameter 9320 (NCE)

0: Displays the guidance programming screen after a program number is registered.

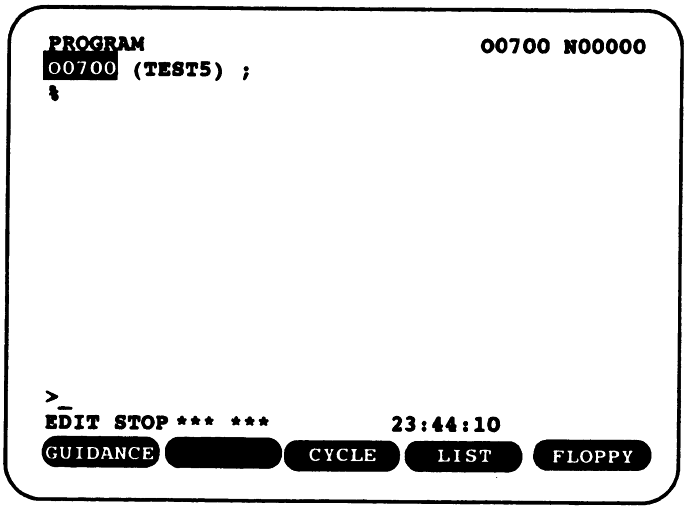
1: Displays the CNC language screen after a program number is registered.

If the NCE bit is set to 0, the guidance programming screen is displayed as shown below:



To display the CNC language screen, press the **[NC LNG]** soft key.

If the NCE bit is set to 1, the CNC language screen is displayed as shown below:



To display the guidance programming screen, press the **[GUIDANCE]** soft key.

To return to the program menu screen, press the **[LIST]** soft key.

5 Creating a program.

For details of creating a program by means of guidance programming, see Section 10.1.4.

For details of creating a program in the CNC language, see Section 9.1.8.

10.1.3 Comment

The operator can select whether to display program names on the program menu screen.

Procedure for switching the comment display

Procedure

1 Press the rightmost soft key. The **[COMMENT]** soft key is displayed. If the **[COMMENT]** soft key is pressed while the program names are displayed on the program menu screen, the program names are hidden and only the program numbers are displayed.

```
PROGRAM   ( EDIT )                    O0100 N00000
 ** PROGRAM MENU **
PROGRAM NO.  USED/FREE                  6/      57
MEMORY AREA  USED/FREE                560/    3200

  O1002 O0100 O0110 O5230 O9654 O3234

PROGRAM NO.    >
INPUT NO. OR SELECT MENU BY CURSOR             2/2
 PUNCH    READ    NC–OUT            COMENT
```

2 If the **[COMMENT]** soft key is pressed while only the program numbers are displayed on the program menu screen, the program names are displayed on the screen.

```
PROGRAM   ( EDIT )                    O0100 N00000
 ** PROGRAM MENU **
PROGRAM NO.  USED/FREE                  6/      57
MEMORY AREA  USED/FREE                560/    3200

  O1002 (NO. 4568–256–47893)
  O0100 (#23546–21)
  O0110 (TEST/12[7]–54)
  O5230 (PROG.SAMPLE(CORNER[1])
  O9654 (SAMPLE–1[ARC])
  O3243 (PN4567891–12/TA)

PROGRAM NO.    >
INPUT NO. OR SELECT MENU BY CURSOR             2/2
 PUNCH    READ    NC–OUT            COMENT
```

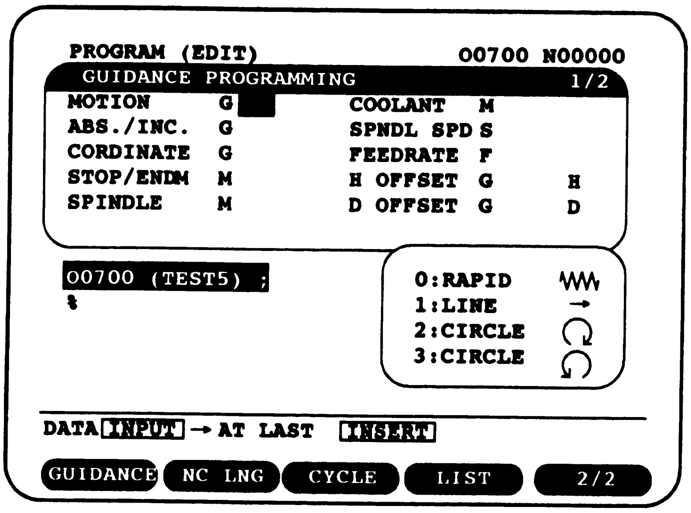
10.1.4 Guidance programming

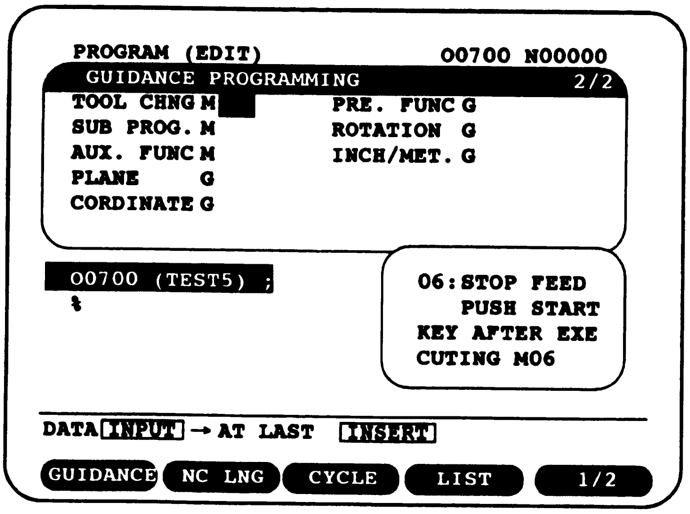
The guidance programming function automatically creates a program in the CNC language using data entered by the operator in response to prompts displayed on the screen. Thus, even a user with little experience of CNC can easily create a program.

Creating a program by means of guidance programming

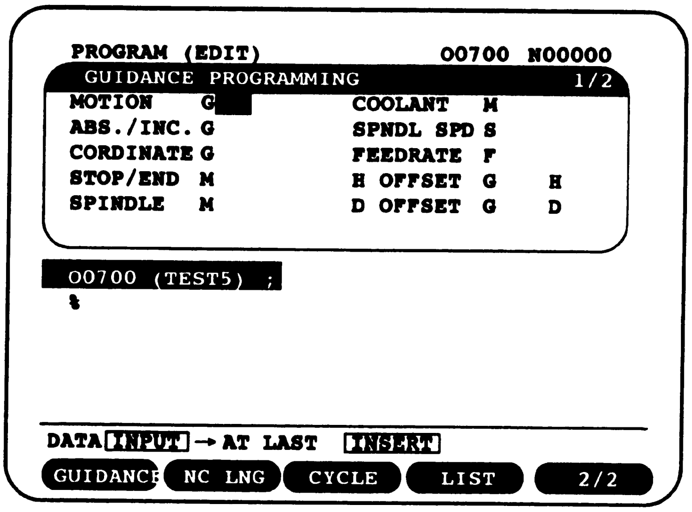
Procedure

1 Register a program as described in Section 10.1.2. The following guidance programming screen is displayed. The input fields, such as MOTION and CORDINATE, are displayed over two pages. The guidance for the field selected by positioning the cursor is also displayed in the bottom right frame.
To display another page, press the [1/2] or [2/2] soft key.





2 If the rightmost soft key is pressed, the secondary soft keys are displayed. To clear the guidance from the screen, press the **[GUIDE]** soft key. To display the guidance, press the same soft key again.



3 Enter data as described below:

3–1 Select a desired input field such as MOTION or CORDINATE by pressing the ← or → key. Select the data entry position for each field, by pressing the ↑ or ↓ key.

3–2 Key in the desired data as explained in the guidance. Then, press the INPUT key.

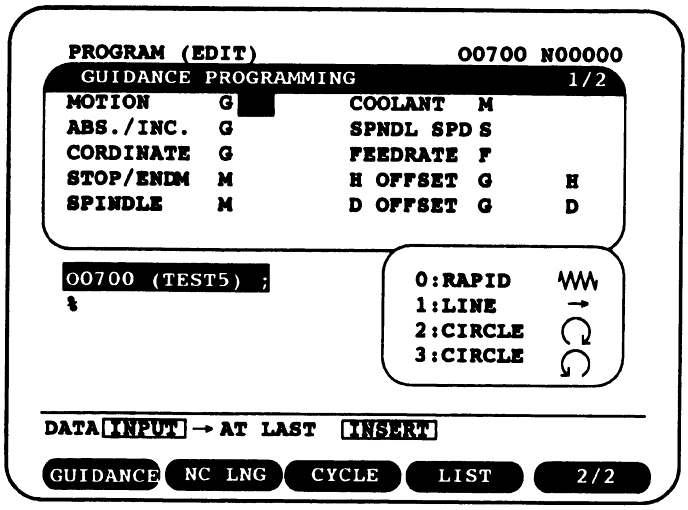
3–3 After entering data into the desired input fields, press the INSERT key. A program consisting of a single block is created.

4 Press the **[NC LNG]** soft key. The CNC language program screen is displayed. Using this screen, a program can be created in the CNC language. For details, see Section 10.2.

Programming a move command

1 Positioning
To specify positioning, enter data as follows:

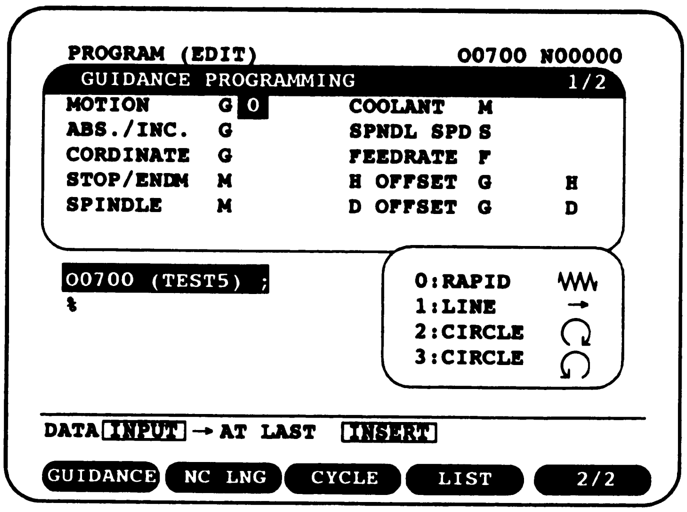
1–1 Enter 0 for the motion command.



1–2 Associated input items are displayed half-way down the screen, at the extreme right. Enter the end position.

Method 1 (Absolute programming)
Enter the X, Y, and Z coordinates of the end point.

Method 2 (Incremental programming)
Enter the X, Y, and Z displacements, relative to the current position.



2 Linear interpolation
To specify linear interpolation, enter data as follows:

2–1 Enter 1 for the motion command.

2–2 Associated input items are displayed half-way down the screen, on the extreme right. Enter the end position.

Method 1 (Absolute programming)
Enter the X, Y, and Z coordinates of the end point.

Method 2 (Incremental programming)
Enter the X, Y, and Z displacements, relative to the current position.

3 Circular interpolation (clockwise rotation)
Circular interpolation (clockwise rotation) is specified as follows:

3–1 Enter 2 for the motion command.

3–2 Associated input items are displayed half-way down the screen, on the extreme right. Enter the end position.

Method 1 (Absolute programming)

Item	Description
X	X coordinate of the end point
Y	Y coordinate of the end point
R	Arc radius
I	Signed value indicating the distance from the initial X coordinate to the final X coordinate
J	Signed value indicating the distance from the initial Y coordinate to the final Y coordinate

Method 2 (Incremental programming)

Item	Description
X	Distance from initial X coordinate to final X coordinate
Y	Distance from initial Y coordinate to final Y coordinate
R	Arc radius
I	Signed value indicating the distance from the initial X coordinate to the final X coordinate
J	Signed value indicating the distance from the initial Y coordinate to the final Y coordinate

4 Circular interpolation (counterclockwise rotation)
To specify circular interpolation (counterclockwise rotation), enter data as follows:

4–1 Enter 3 for the motion command.

4–2 Associated input items are displayed half-way down the screen, on the extreme right. Enter the end position.

Method 1 (Absolute programming)

Item	Description
X	X coordinate of the end point
Y	Y coordinate of the end point
R	Radius of an arc
I	Signed value indicating the distance from the initial X coordinate to the final X coordinate
J	Signed value indicating the distance from the initial Y coordinate to the final Y coordinate

Method 2 (Incremental programming)

Item	Description
X	Distance from initial X coordinate to final X coordinate
Y	Distance from initial Y coordinate to final Y coordinate
R	Arc radius
I	Signed value indicating the distance from the initial X coordinate to the final X coordinate
J	Signed value indicating the distance from the initial Y coordinate to the final Y coordinate

Specifying Absolute and Incremental Programming

The distance the tool moves along an axis can be specified as either an absolute or incremental value. Either of these two programming modes, absolute or incremental, can be specified with the following G codes. (For details, see Section 8.1, "Absolute and Incremental Programming" of Part II "Programming.")

- G90 (Absolute programming)
  Specified when the coordinates of the final position of the tool movement along an axis are input.
- G91 (Incremental programming)
  Specified when the distance the tool moves along an axis is input.

Specifying the coordinate system

Five special workpiece coordinate systems are preset for the machine. One of these five coordinate systems is selected by specifying the corresponding G code, from G55 to G59. (For details, see Section 7.2, "Programming of Workpiece Coordinate System" of Part II "Programming.")

Specifying an end/stop

A program can be automatically stopped or terminated by using the following M codes. (For details, see Section 11, "Miscellaneous Function" of Part II "Programming.")

M02

- Indicates the end of the main program.
- Stops automatic operation, and resets the system.
- Returns control to the top of the program.

M30
The same function as M02

M00
Executes the block containing M00, then stops automatic operation. When the start signal subsequently goes on, automatic operation is resumed.

M01
Executes the block containing M01, then stops automatic operation, as with M00. Note, however, that automatic operation can be stopped only when the setting of the optional stop switch is ON.

Specifying the spindle command

The spindle can be started and stopped with the following M codes:

M03
Rotates the spindle such that a right-hand thread moves forward relative to the workpiece.

M04
Rotates the spindle such that a right-hand thread moves back relative to the workpiece.

M05
Stops the spindle.

Coolant programming

The coolant supply can be turned on and off by using the following M codes:

M08
Turns on the coolant supply.

M09
Turns off the coolant supply.

Spindle speed programming

Enter a spindle speed. (For details, see Section 9, "Spindle Speed Function" of Part II " Programming.")

Feedrate programming

Enter a tool feedrate in operation, such as linear interpolation (G01), and circular interpolation (G02, G03). (For details, see Section 5, "Feed Functions" of Part II "Programming.")

Tool length compensation

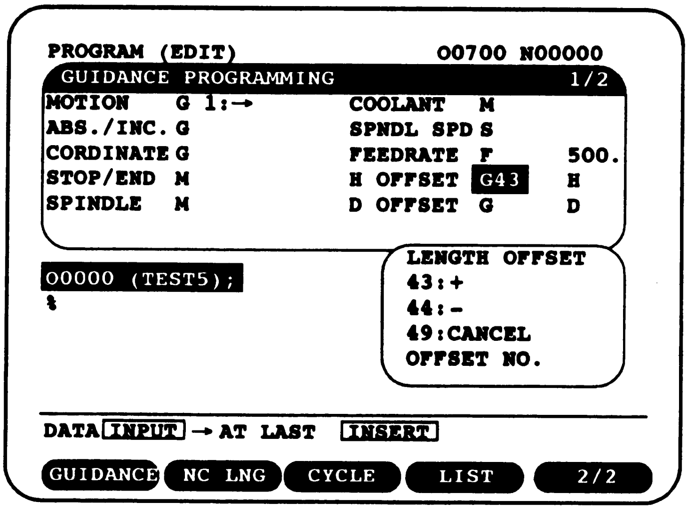
1 Specifying the tool length compensation

1–1 Enter the offset direction for tool length compensation.

G code	Function
G43	Positive offset
G44	Negative offset

(For details, see Section 14.1, "Tool Length Offset" of Part II "Programming.")

1–2 Associated input items are displayed half-way down the screen, on the extreme right. Enter an offset value.



2 Canceling tool length compensation
To cancel the offset, enter 49.

Cutter compensation

Cutter compensation can be specified.

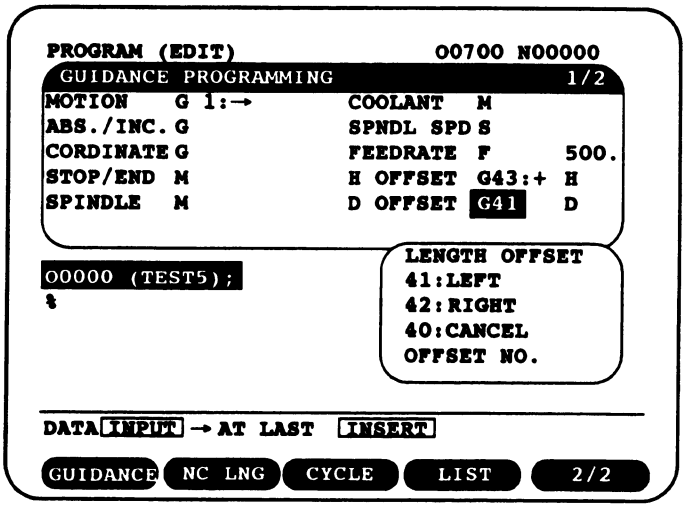
1 Specifying cutter compensation

1–1 Enter the offset direction.

G code	Function
G41	Offset to the left, viewed from behind the tool
G42	Offset to the right, viewed from behind the tool

(For details, see Section 14.2, "Cutter Compensation C" of Part II "Programming.")

1–2 Associated input items are displayed half-way down the screen, on the extreme right. Enter an offset value.



2 Canceling cutter compensation
To cancel the offset, enter 40.

Tool change

Enter M06 to change the tool.

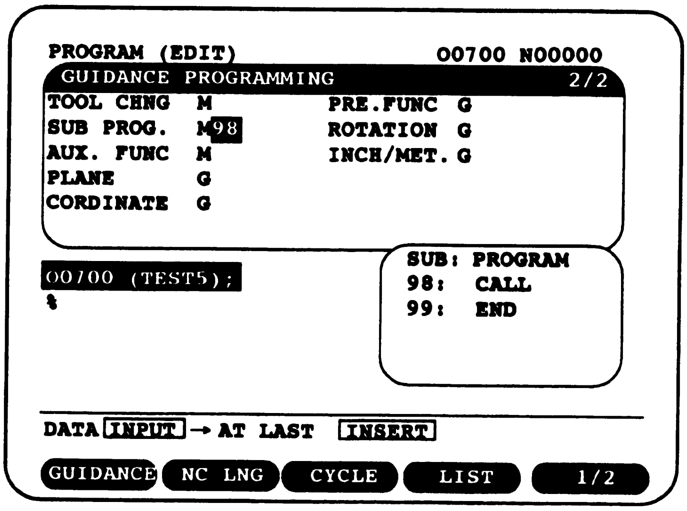
Subprogram call

1 A subprogram can be called by using the following procedure:

1–1 Enter 98.

1–2 Associated input items are displayed half-way down the screen, on the extreme right.

(For details, see Section 12, "Program Configuration" of Part II "Programming.")



1–3 Enter the program number of the subprogram to be called.

1–4 To call a subprogram repeatedly, enter a calling count.

2 Ending a subprogram
To end a subprogram, enter 99.

Auxiliary function M codes

M codes other than those for stop/end, spindle commands, coolant supply, tool change, and subprogram call are entered.

Plane selection

Enter the plane for circular interpolation, cutter compensation, coordinate rotation, or drilling. (For details, see Section 7.4, "Plane Selection" of Part II "Programming.")

G code	Selected plane
G17	XY plane
G18	ZX plane
G19	YZ plane

Coordinate system

Specify whether the machine coordinate system or local coordinate system is to be used for programming. (For details, see Sections 7.1, "Machine Coordinate System" and 7.3, "Local Coordinate System" of Part II "Programming.")

- G53 (Machine coordinate system)
  Specified when rapid traverse is to be performed using the machine coordinate system.
  Note) G53 is valid only in absolute mode; in incremental mode, this code is ignored.
- G52 (Local coordinate system)
  Specified when programming is to use the local coordinate system.

Optional G code programming

G codes other than those for motion commands, absolute/incremental programming, the workpiece coordinate systems, tool length compensation, cutter compensation, plane selection, coordinate systems, coordinate rotation, and input systems can be entered.

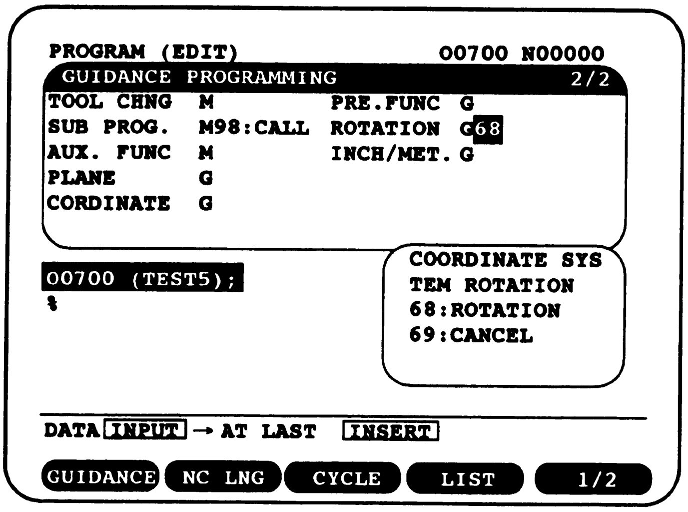
Coordinate system rotation

A programmed figure can be rotated. (For details, see Section 14.6, "Coordinate System Rotation" of Part II "Programming.")

1 Specifying coordinate system rotation

1–1 Enter 68.

1–2 Associated input items are displayed half-way down the screen, on the extreme right.



The items input as the coordinates of the rotation center, X, Y, and Z, are always displayed. Specify these values according to the selected plane.

(a) When G17 is specified
Enter the X and Y coordinates of the rotation center, as well as angular displacement R.

(b) When G18 is specified
Enter the X and Z coordinates of the rotation center, as well as angular displacement R.

(c) When G19 is specified
Enter the Y and Z coordinates of the rotation center, as well as angular displacement R.

2 G69 (canceling coordinate system rotation)
To cancel coordinate system rotation, enter 69.

Specifying input system

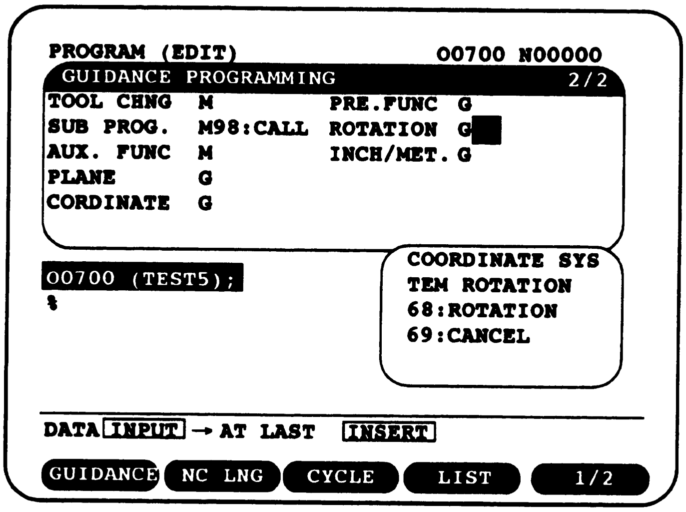
Specify whether input is to be done using inches or metric measurements. (For details, see Section 8.2, "Inch/Metric Conversion" of Part II "Programming.")

Input unit	G code
Inches	G20
Metric	G21

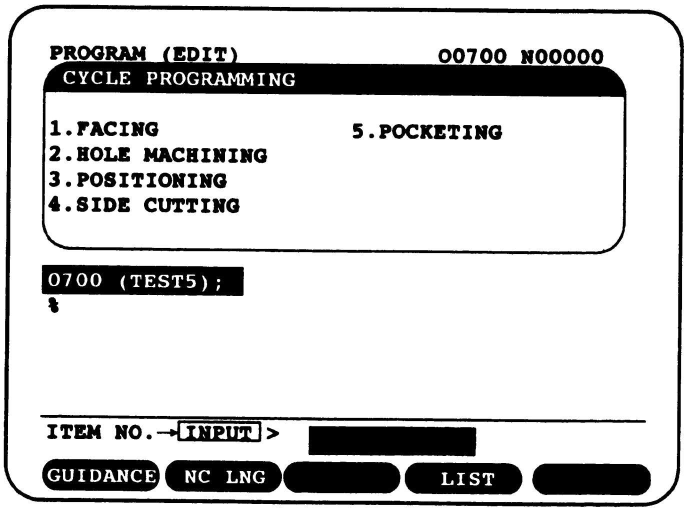
10.2 CREATING A PROGRAM BY CYCLE PROGRAMMING

The optional guidance programming function allows you to easily program facing, drilling, pattern positioning, side face machining, and pocketing by means of cycle programming.

Cycle programming



Press the [CYCLE] soft key on the guidance programming screen. Then, the machining menu screen for cycle programming, shown below, appears.

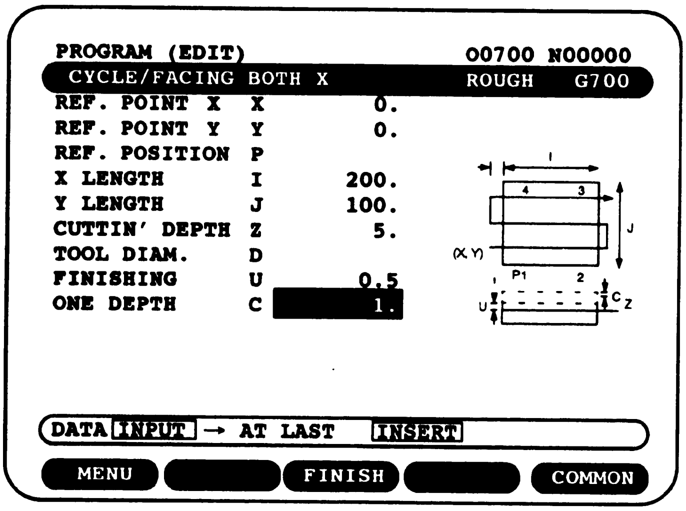


The table below lists the items in the cycle programming menu.

Machining menu	Machining	Machining menu	Machining
1. Facing	1. Both directions along the X-axis	4. Side cutting	1. External circular side cutting
	2. Both directions along the Y-axis		2. External rectangular side cutting
	3. Single direction along the X-axis		3. External track side cutting
	4. Single direction along the Y-axis		4. Internal circular side cutting
2. Hole machining	1. Center drilling	5. Pocketing	1. Circular pocketing
	2. Drilling		2. Rectangular pocketing
	3. Tapping		3. Track pocketing
	4. Boring		
3. Pattern positioning	1. Points on a circumference		
	2. Points on an arc		
	3. Points on a rectangle		
	4. Points on a grid		
	5. Arbitrary points		

10.2.1 Facing

Facing in both diractions along the X–axis (roughing)



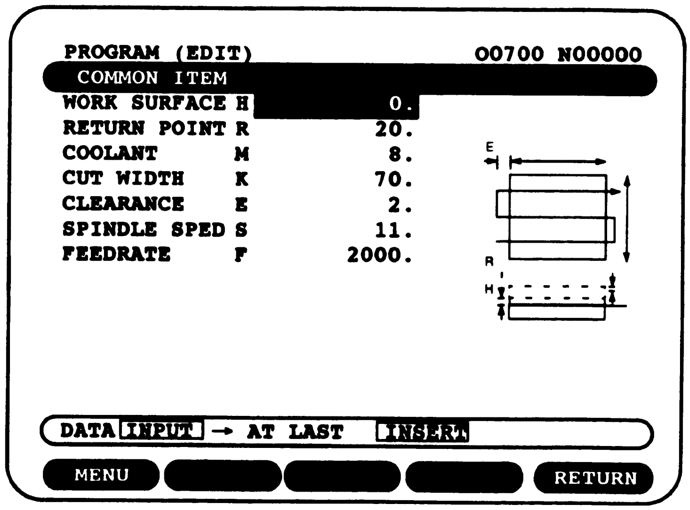
1 Position the cursor to an item to be specified.

2 Enter the desired value, then press the INPUT key.

Table 10.2.1 (a) Input Items for Facing in Both Directions along the X–Axis (Roughing)

Item	Description
Reference position X	X coordinate of the reference position of the work-piece
Reference position Y	Y coordinate of the reference position of the work-piece
Reference position	Position of the reference of the workpiece
X-side length	Length of workpiece along the X-axis
Y-side length	Length of the workpiece along the Y-axis
Cutting depth	Cutting depth relative to the surface of the workpiece, along Z-axis
Tool diameter	Diameter of the tool
Finishing allowance	Allowance for finishing
Depth of cut made by one cut	Depth of cut made by one roughing cut

3 Press the [COMMON] soft key to display the common item screen.



4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

Note) Once a common item has been specified, the entered data remains effective indefinitely.

Table 10.2.1 (b) Common Item screen Input Items

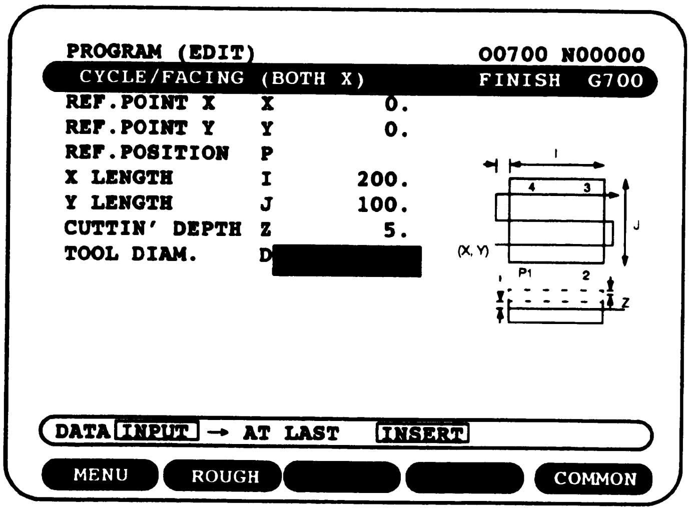
Item	Description
Workpiece surface coordinate	Z coordinate of the surface of the workpiece
Return point coordinate	Z coordinate to which the tool will return after cutting
Coolant	M code for coolant
Clearance	Clearance of the tool used between two machining operations
Cutting width (%)	Effective cutting width of the tool
Spindle speed	Speed of the spindle
Feedrate	Feedrate

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Facing in both directions along the x–axis (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Table 10.2.1 (c) Input Items for Facing in both Directions along the X–Axis (Finishing)

Item	Description
Reference position X	X coordinate of the reference position of the workpiece
Reference position Y	Y coordinate of the reference position of the workpiece
Reference position	Position of the reference of the workpiece
X-side length	Length of the workpiece along the X-axis
Y-side length	Length of the workpiece along the Y-axis
Cutting depth	Cutting depth, relative to the surface of the workpiece along the Z-axis
Tool diameter	Diameter of the tool

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

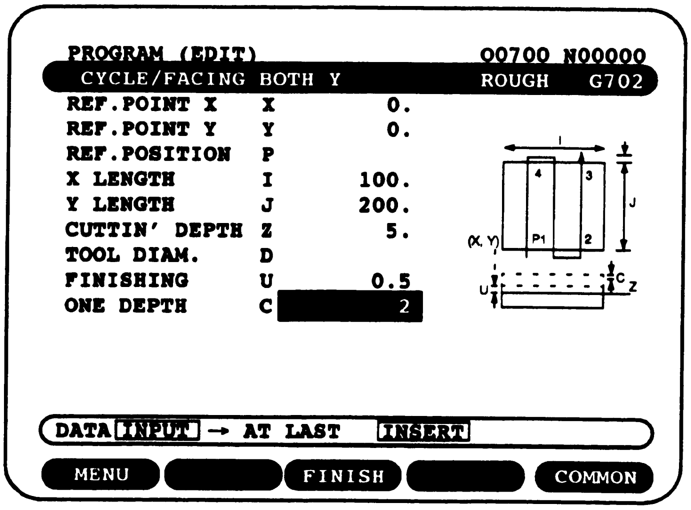
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Facing in both directions along the Y–axis (roughing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

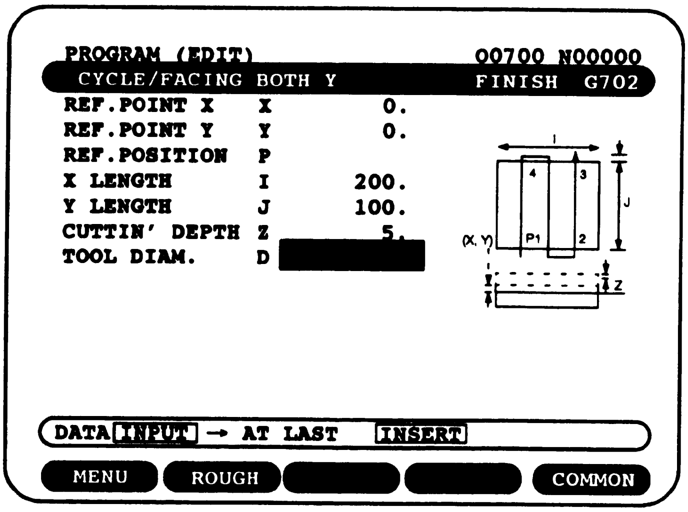
Note) Once a common item has been specified, the entered data remains effective indefinitely.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Facing in both directions along the Y–axis (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

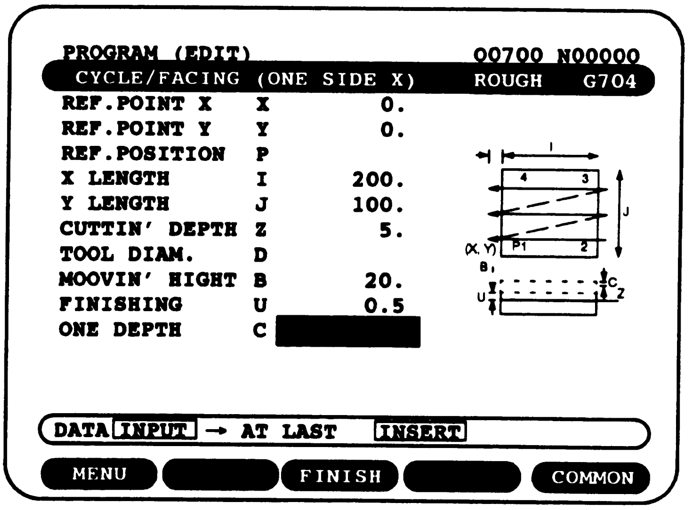
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Facing in one direction along the X–axis (roughing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the [INPUT] key.

Table 10.2.1 (d) Input Items for Facing in One Direction along the X–Axis (Roughing)

Item	Description
Reference position X	X coordinate of the reference position of the workpiece
Reference position Y	Y coordinate of the reference position of the workpiece
Reference position	Position of the reference of the workpiece
X-side length	Length of the workpiece along the X-axis
Y-side length	Length of the workpiece along the Y-axis
Cutting depth	Cutting depth relative to the surface of a workpiece along the Z-axis
Tool diameter	Diameter of the tool
Movement height	Height at which the tool is moved to the start point
Finishing allowance	Allowance for finishing
Depth of cut made by one cut	Depth of cut made by one roughing cut

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the [INPUT] key.

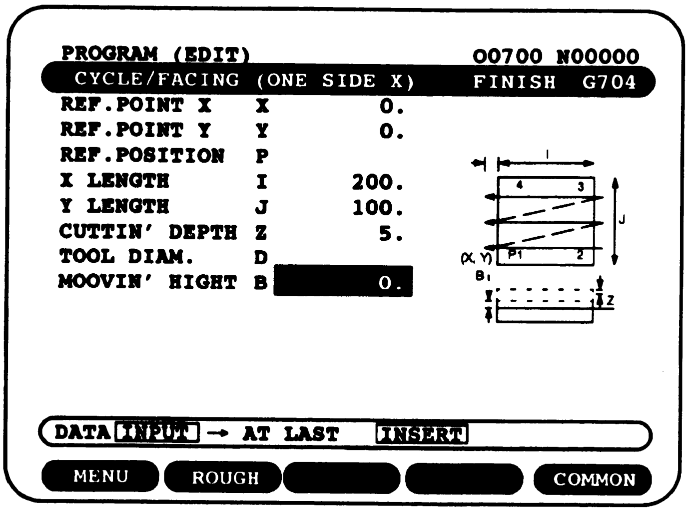
Note) Once a common item has been specified, the entered data remains effective indefinitely.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the [INSERT] key.

Facing in single direction along the X–axis (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Table 10.2.1 (e) Input Items for Facing in One Direction along the X–Axis (Finishing)

Item	Description
Reference position X	X coordinate of the reference position of the workpiece
Reference position Y	Y coordinate of the reference position of the workpiece
Reference position	Position of the reference of the workpiece
X-side length	Length of the workpiece along the X-axis
Y-side length	Length of the workpiece along the Y-axis
Cutting depth	Cutting depth relative to the surface of a workpiece along the Z-axis
Tool diameter	Diameter of the tool
Movement height	Height at which a tool is moved to the start point

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

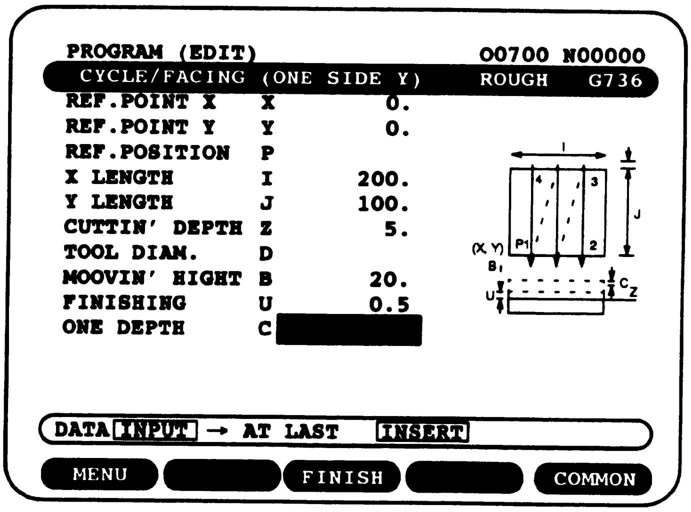
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common item to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Facing in single direction along the Y–axis (roughing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

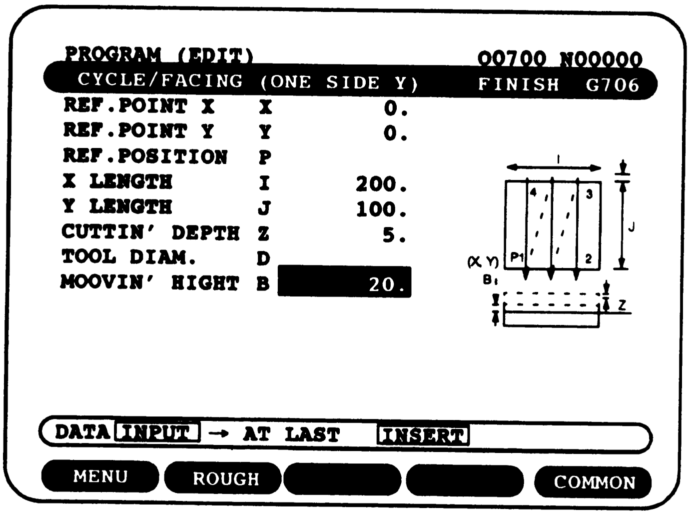
Note) Once a common item has been specified, the entered data remains effective indefinitely.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Facing in one direction along the Y–axis (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

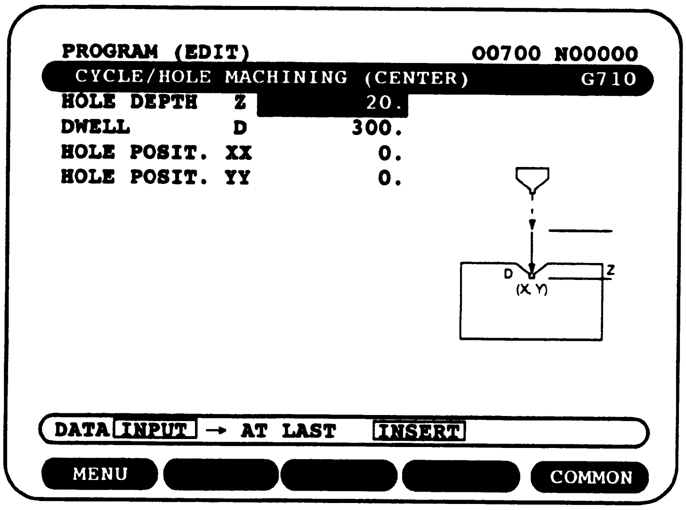
6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

10.2.2 Hole mahcining

Center drilling

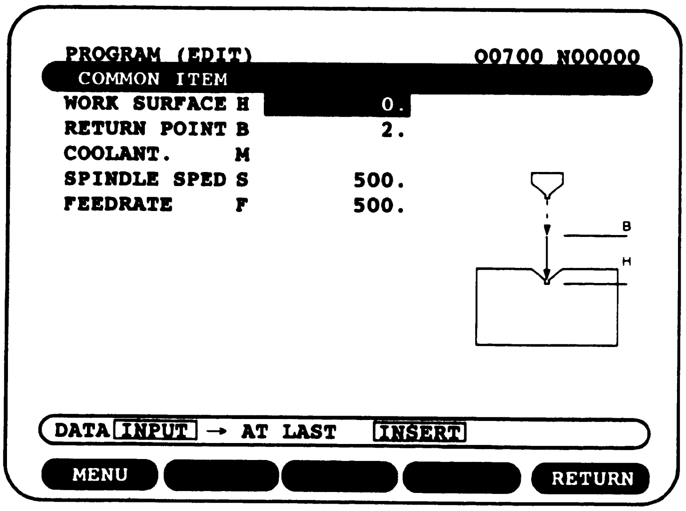


1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Hole depth	Cutting depth relative to the surface of the workpiece along the Z-axis
Dwell time	Dwell time at the bottom of the hole
Hole position X	X coordinate of the hole
Hole position Y	Y coordinate of the hole

3 Press the [COMMON] soft key to display the common item screen.



4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

Note) Once a common item has been specified, the entered data remains effective indefinitely.

Table 10.2.2 (a) Input Items on the Common Item Screen

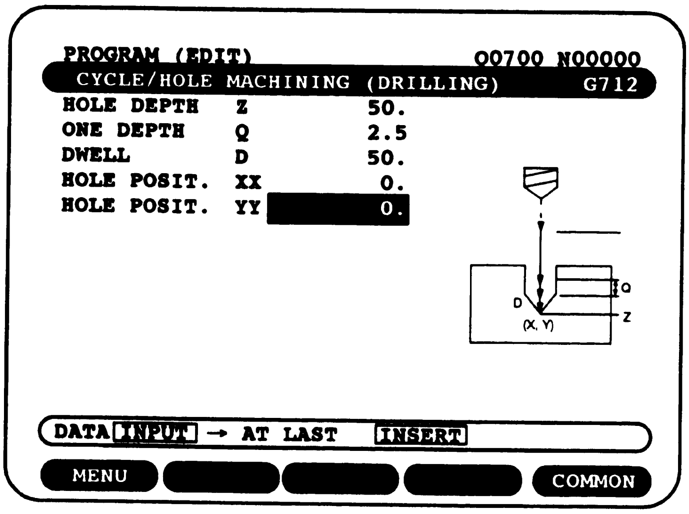
Item	Description
Workpiece surface coordinate	Z coordinate on the surface of the workpiece
Return point coordinate	Z coordinate to which the tool will return after hole machining
Coolant	M code for coolant
Spindle speed	Speed of the spindle
Feedrate	Feedrate

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Drilling



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Hole depth	Cutting depth relative to the surface of the workpiece along the Z-axis
Depth of cut made by one cut	Depth of cut made by one cutting operation
Dwell time	Dwell time at the bottom of the hole
Hole position X	X coordinate of the hole
Hole position Y	Y coordinate of the hole

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

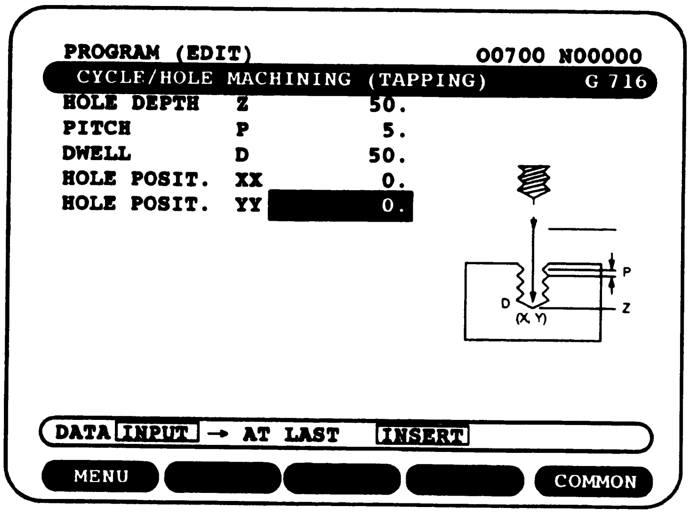
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Tapping



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Hole depth	Cutting depth relative to the surface of the work-piece along the Z-axis
Pitch	Pitch of the tapping tool
Dwell time	Dwell time used when the spindle is rotated counterclockwise at the bottom of the hole and when the spindle is rotated clockwise after machining
Hole position X	X coordinate of the hole
Hole position Y	Y coordinate of the hole

3 Press the [COMMON] soft key to display the common item screen.

4 Move the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

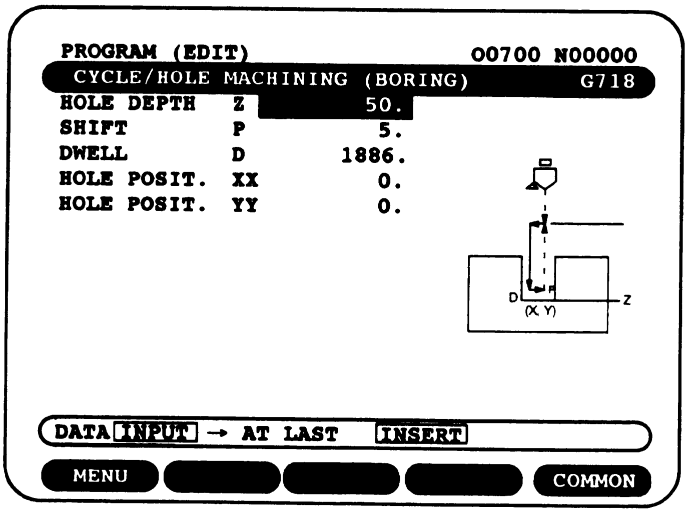
Note) Once a common item has been specified, the entered data remains effective indefinitely.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Boring



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Hole depth	Cutting depth relative to the surface of the workpiece along the Z-axis
Shift amount	Amount of shift in the tool tip direction after an oriented spindle stop
Dwell time	Dwell time at point Z
Hole position X	X coordinate of the hole
Hole position Y	Y coordinate of the hole

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

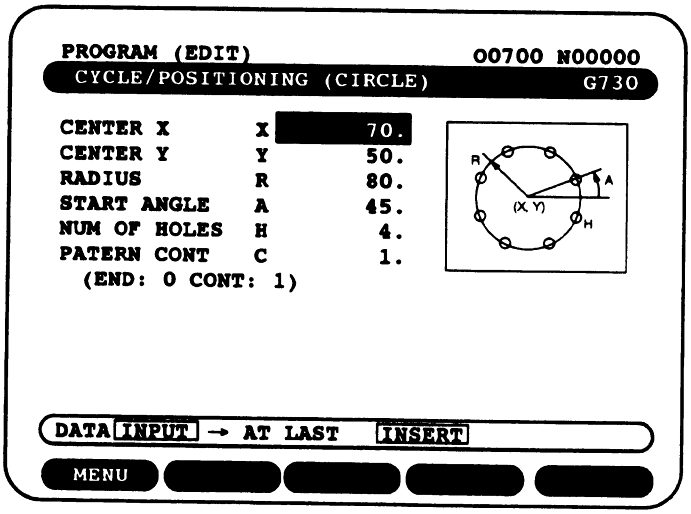
Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

10.2.3 Pattern positioning

Pattern positioning is specified after a hole machining operation has been specified.

Points on a circumference



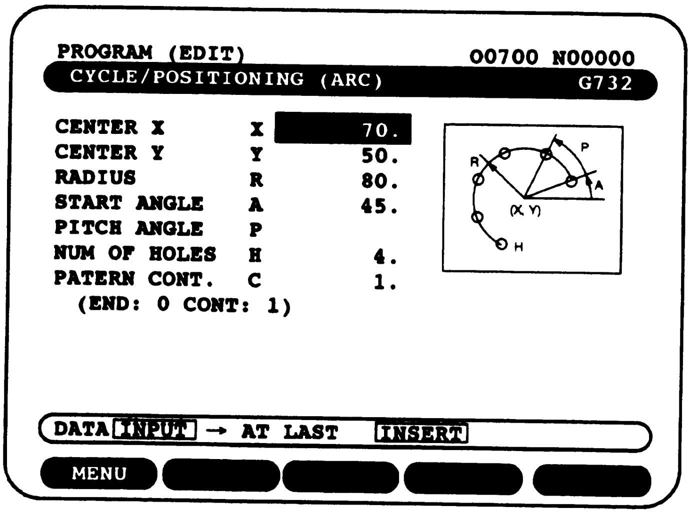
1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Center coordinate X	X coordinate of the center of the circle
Center coordinate Y	Y coordinate of the center of the circle
Radius	Radius of the circle
Start angle	Angle of the position of the first hole on the circumference
Number of holes	Number of holes on the circumference
Pattern continuation	Specifies whether to continue hole machining pattern input. 1: Continues hole machining pattern specification. 0: Ends hole machining pattern specification.

3 Press the INSERT key.

Points on an arc



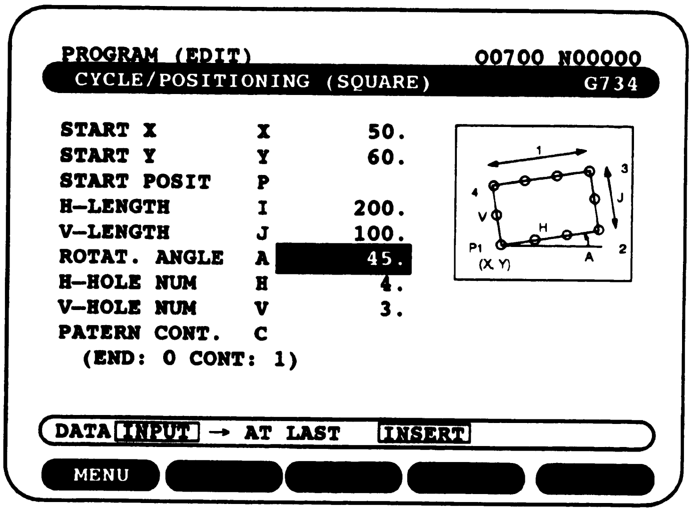
1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Center coordinate X	X coordinate of the center of the arc
Center coordinate Y	Y coordinate of the center of the arc
Radius	Radius of the arc
Start angle	Angle of the position of the first hole on the arc
Pitch angle	Angle subtended by two adjacent holes
Number of holes	Number of holes on the arc
Pattern continuation	Specifies whether to continue hole machining pattern input.

3 Press the INSERT key.

Points on a rectangle



1 Position the cursor to the item to be specified.

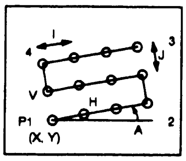
2 Enter the desired value, then press the INPUT key.

Item	Description
Start point coordinate X	X coordinate of the reference position
Start point coordinate Y	Y coordinate of the reference position
Start position	First hole machining position
X-side length	Length of the X side of the rectangle
Y-side length	Length of the Y side of the rectangle
Angle of rotation	Angle of the rectangle
Number of X-side holes	Number of holes on the X side of the rectangle
Number of Y-side holes	Number of holes on the Y side of the rectangle
Pattern continuation	Specifies whether to continue hole machining pattern input.

3 Press the INSERT key.

Points on a grid

```
PROGRAM (EDIT)                          O0700 N00000
 CYCLE/POSITIONING (GRID)                       G736

START X          X       50.
START Y          Y       60.
START POSIT      P
H-PITCH          I      200.
V-PITCH          J      100.
LINE ANGLE       A       45.
H-HOLE NUM       H        4.
V-HOLE NUM       V        3.
PATERN CONT.     C        1.
  (END: 0 CONT: 1)

DATA [INPUT] → AT LAST   [INSERT]
MENU
```



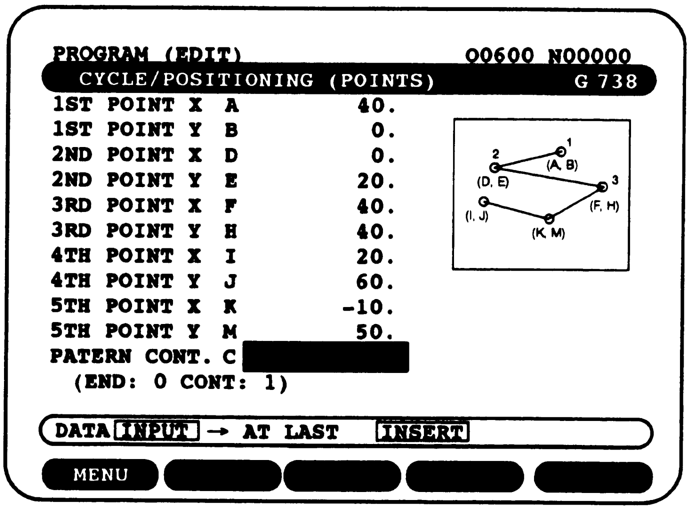
1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Start point coordinate X	X coordinate of the reference position
Start point coordinate Y	Y coordinate of the reference position
Start position	First hole machining position
Line length	Length of lines
Distance between lines	Distance between two adjacent parallel lines
Line angle	Angle of the lines
Number of holes	Number of holes on the lines
Number of lines	Number of lines
Pattern continuation	Specifies whether to continue hole machining pattern input.

3 Press the INSERT key.

Arbitrary points



1 Position the cursor to the item to be specified.

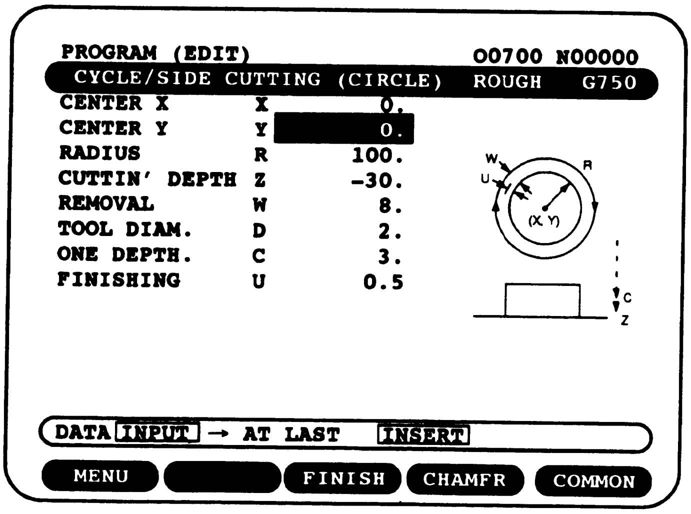
2 Enter the desired value, then press the INPUT key.

Item	Description
Point 1 X	X coordinate of point 1
Point 1 Y	Y coordinate of point 1
Point 2 X	X coordinate of point 2
Point 2 Y	Y coordinate of point 2
Point 3 X	X coordinate of point 3
Point 3 Y	Y coordinate of point 3
Point 4 X	X coordinate of point 4
Point 4 Y	Y coordinate of point 4
Point 5 X	X coordinate of point 5
Point 5 Y	Y coordinate of point 5
Pattern continuation	Specifies whether to continue hole machining pattern input.

3 Press the INSERT key.

10.2.4 Side cutting

External circular side cutting (roughing)

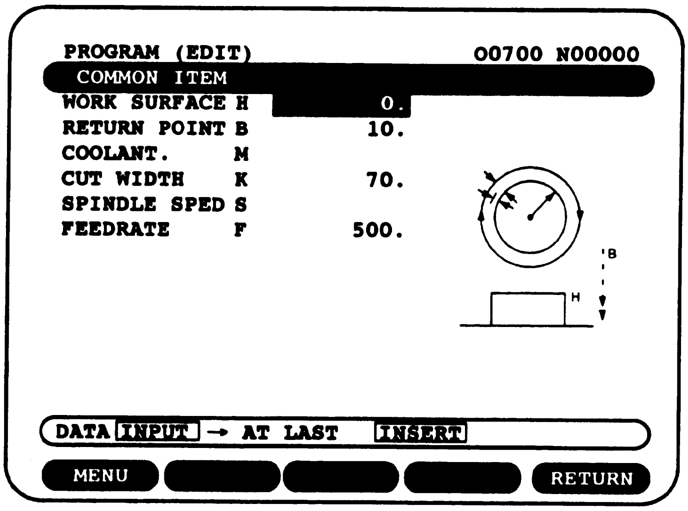


1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Center coordinate X	X coordinate of the center of the circle
Center coordinate Y	Y coordinate of the center of the circle
Radius	Radius of the cylindrical workpiece
Cutting depth	Cutting depth relative to the surface of the workpiece along the Z-axis
Cutting allowance	Allowance for roughing
Tool diameter	Diameter of the tool
Depth of cut made by one cut	Depth of cut made by one roughing cut
Finishing allowance	Allowance for finishing

3 Press the [COMMON] soft key to display the common item screen.



4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

Note) Once a common item has been specified, the entered data remains effective indefinitely.

Table 10.2.4 (a) Common Item Screen Input Items

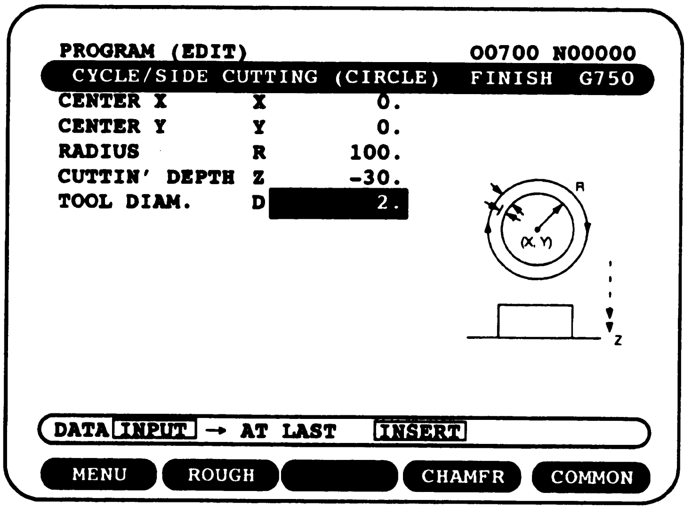
Item	Description
Workpiece surface coordinate	Z coordinate of the surface of the workpiece
Return point coordinate	Z coordinate to which the tool returns after cutting
Coolant	M code for coolant
Cutting width (%)	Effective cutting width of the tool
Spindle speed	Speed of the spindle
Feedrate	Feedrate

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

External circular side cutting (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

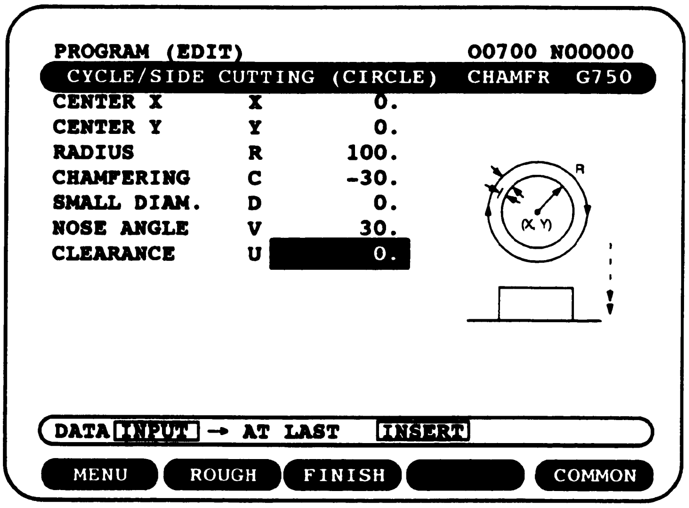
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

External circular side cutting (chamfering)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

Item	Description
Chamfering amount	Amount of chamfering at the top of the workpiece
Small diameter	Small diameter of the chamfering tool
Tool angle	Angle of the chamfering tool
Clearance	Clearance of the chamfering tool

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

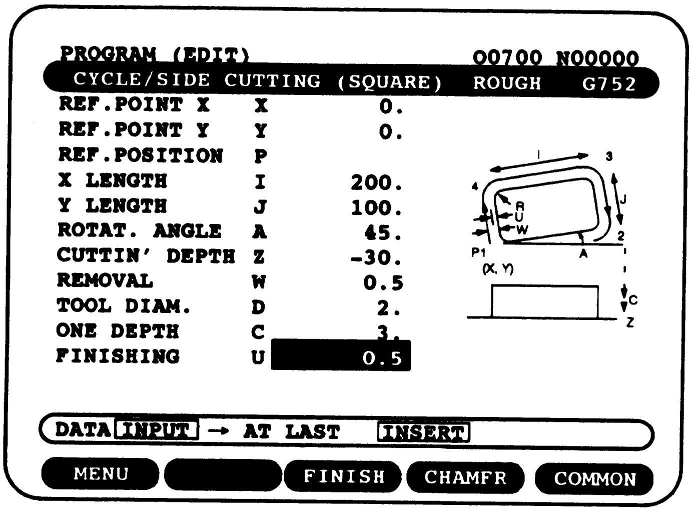
Note) Once a common item has been specified, the entered data remains effective indefinitely.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

External rectangular side cutting (roughing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Reference position X	X coordinate of the reference position of the work-piece
Reference position Y	Y coordinate of the reference position of the work-piece
Reference position	Position of the reference of the workpiece
X-side length	Length of the X side of the workpiece
Y-side length	Length of the Y side of the workpiece

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

Item	Description
Corner R	Radius used when corner R is applied to the four corners of the workpiece
Corner C	Amount of chamfering used when the four corners of the workpiece are chamfered

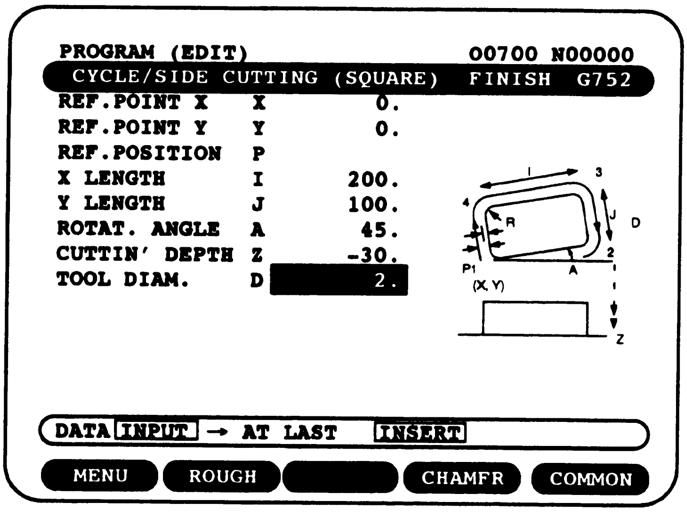
Corner R data and corner C data can not be specified simultaneously.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

External rectangular side cutting (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

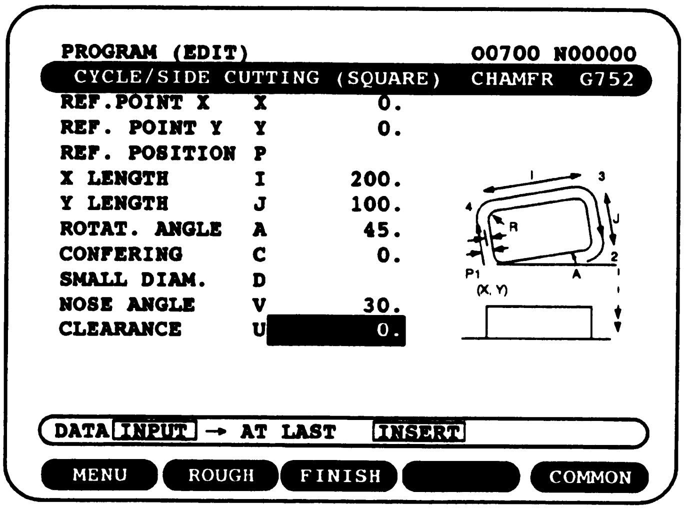
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

External rectangular side cutting (chamfering)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

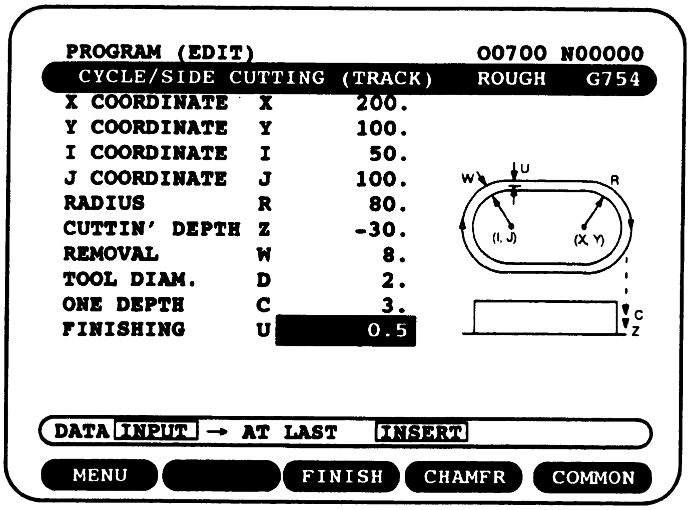
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

External track side cutting (roughing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Right center coordinate X	X coordinate of the center of the right-hand semicircle
Right center coordinate Y	Y coordinate of the center of the right-hand semicircle
Left center coordinate X	X coordinate of the center of the left-hand semi-circle
Left center coordinate Y	Y coordinate of the center of the left-hand semi-circle
Semicircle radius	Radius of the left- and right-hand semicircles forming the track shape

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

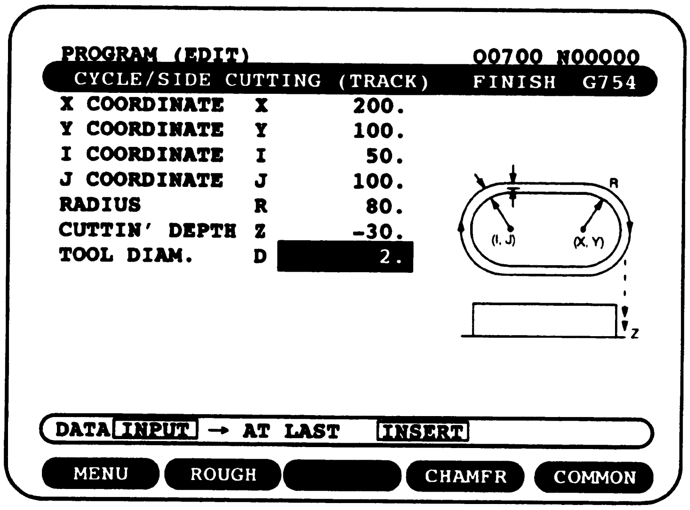
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

External track side cutting (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

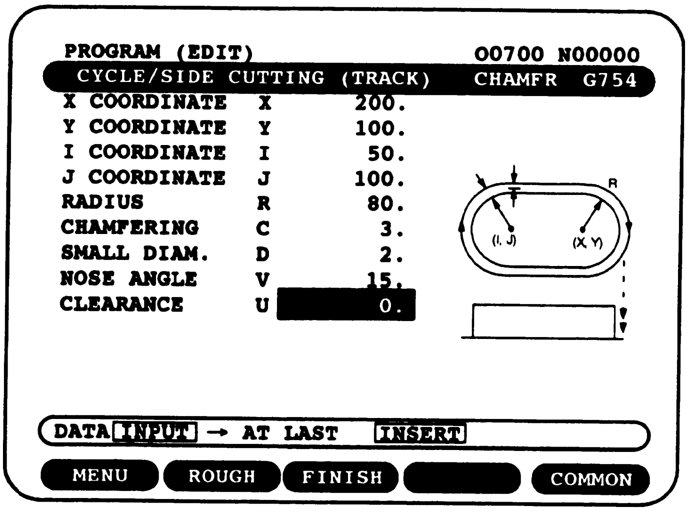
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

External track side cutting (chamfering)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

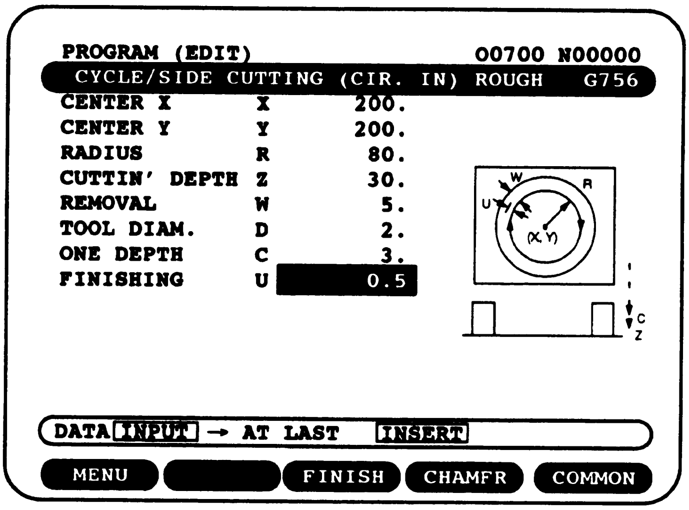
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Internal circular side cutting (roughing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

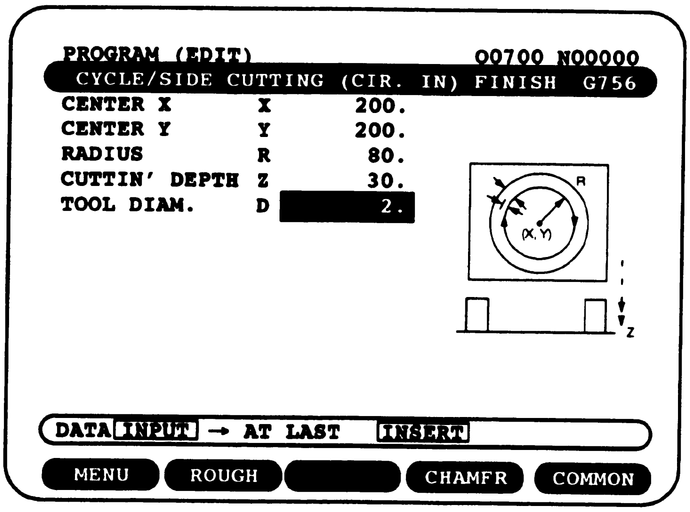
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 need not be performed.

7 Press the INSERT key.

Internal circular side cutting (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

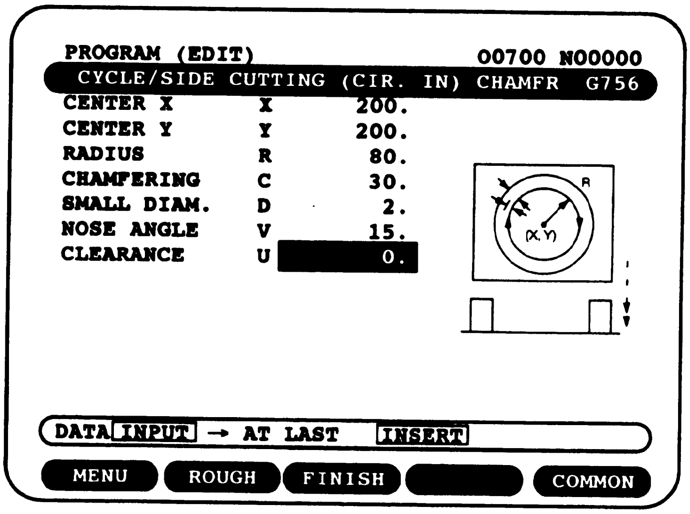
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Internal circular side cutting (chamfering)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

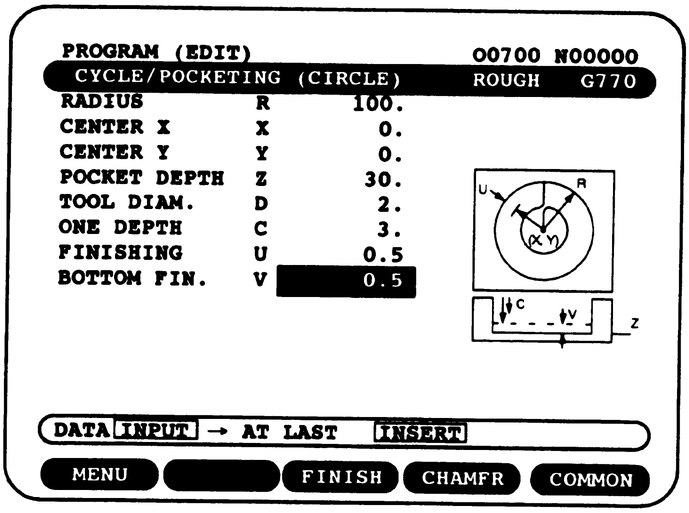
6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

10.2.5 Pocketing

Circular pocketing (roughing)

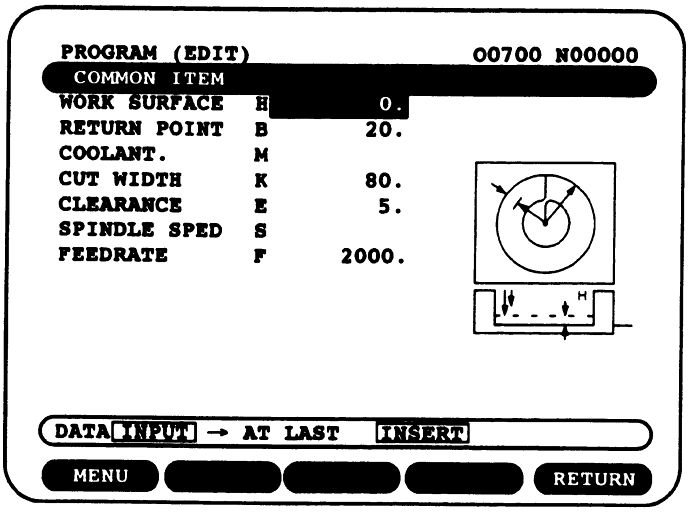


1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Center coordinate X	X coordinate of the center of the circle
Center coordinate Y	Y coordinate of the center of the circle
Radius	Radius of the cylindrical workpiece
Cutting depth	Cutting depth relative to the surface of the workpiece along the Z-axis
Cutting allowance	Allowance for roughing
Tool diameter	Diameter of the tool
Depth of cut made by one cut	Depth of cut made by one roughing cut
Finishing allowance	Allowance for finishing

3 Press the [COMMON] soft key to display the common item screen.



4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

Note) Once a common item has been specified, the entered data remains effective indefinitely.

Table 10.2.5 (a) Common Item Screen Input Items

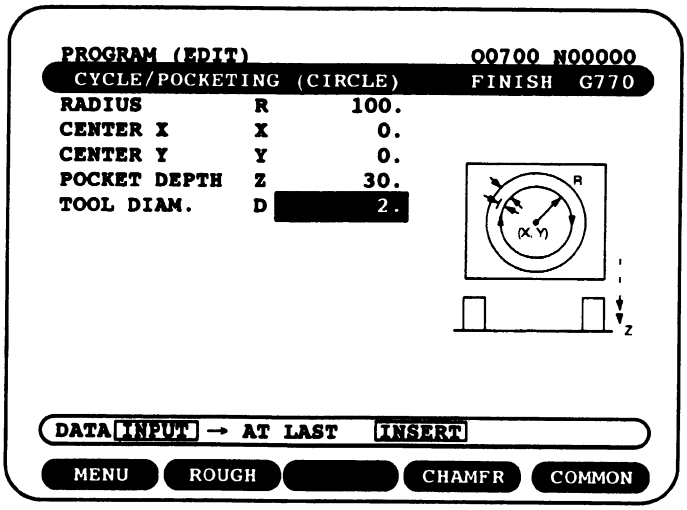
Item	Description
Workpiece surface coordinate	Z coordinate of the surface of the workpiece
Return point coordinate	Z coordinate to which tool returns after cutting
Coolant	M code for coolant
Cutting width (%)	Effective cutting width of the tool
Spindle speed	Speed of the spindle
Feedrate	Feedrate

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Circular pocketing (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

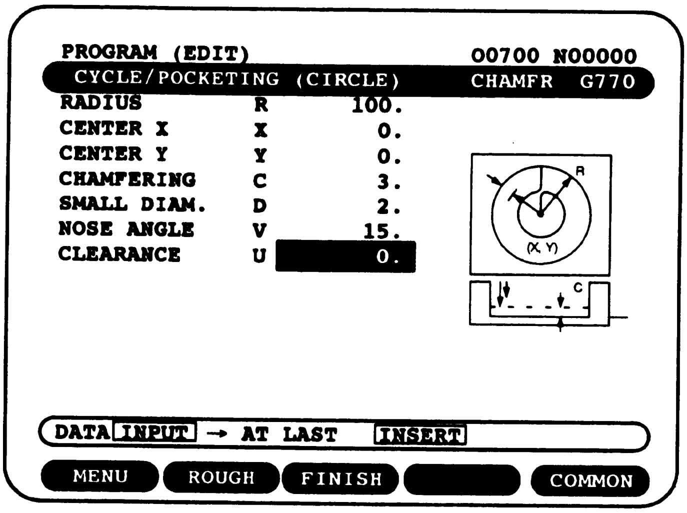
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Circular pocketing (chamfering)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the [INPUT] key.

3 Press the [COMMON] soft key to display the common item screen.

Item	Description
Chamfering amount	Amount of chamfering at the top of a workpiece
Small diameter	Small diameter of the chamfering tool
Tool angle	Angle of the chamfering tool
Clearance	Clearance of the chamfering tool

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the [INPUT] key.

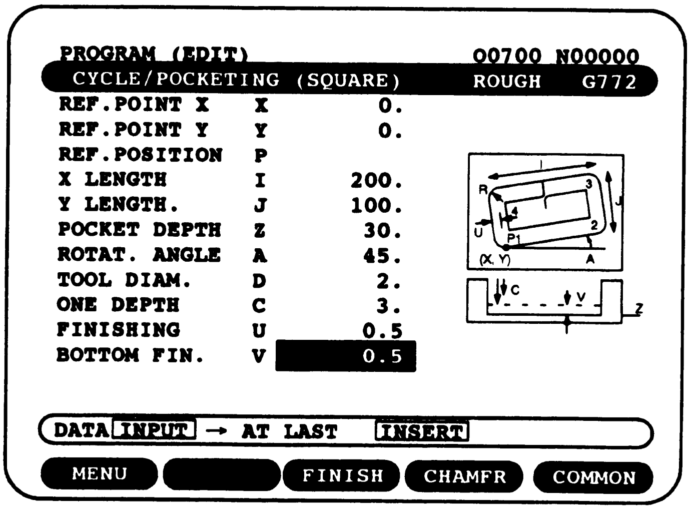
Note) Once a common item has been specified, the entered data remain effective indefinitely.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the [INSERT] key.

Rectangular pocketing (roughing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

Item	Description
Reference position X	X coordinate of the reference position of the workpiece
Reference position Y	Y coordinate of the reference position of the workpiece
Reference position	Position of the reference of the workpiece
X-side length	Length of the X side of the workpiece
Y-side length	Length of the Y side of the workpiece

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

Item	Description
Corner R	Radius used when corner R is applied to the four corners of the workpiece
Corner C	Amount of chamfering used when the four corners of the workpiece are chamfered

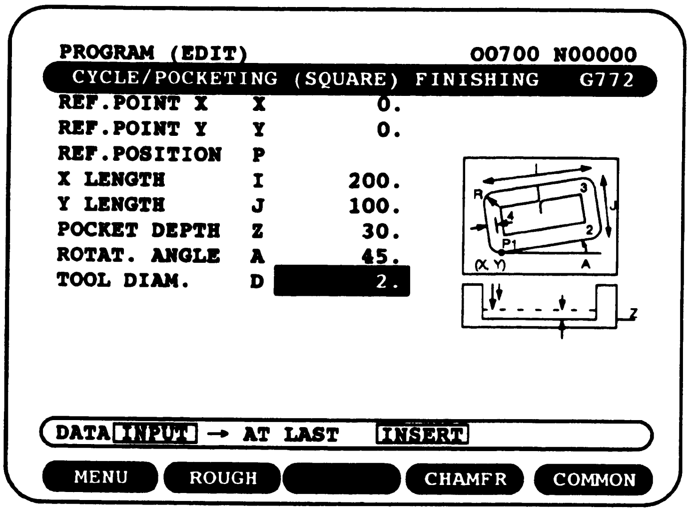
Both corner R data and corner C data can not be specified simultaneously.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Rectangular pocketing (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

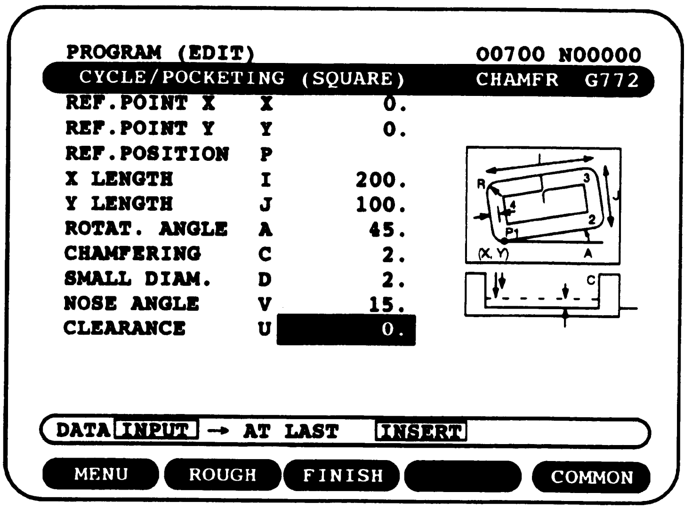
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Rectangular pocketing (chamfering)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

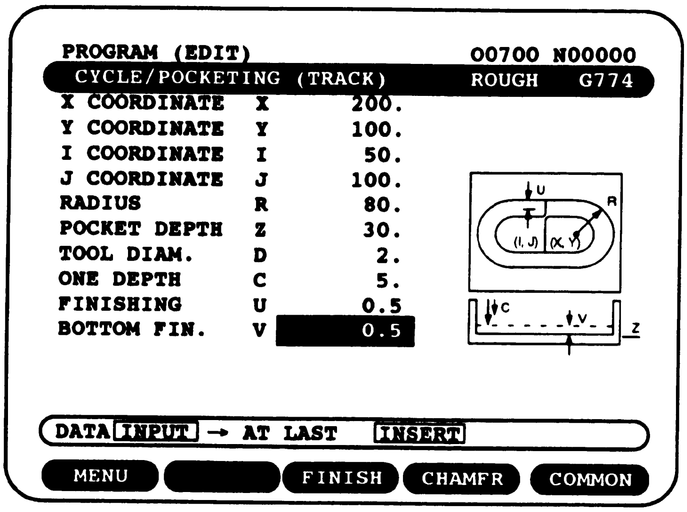
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Track pocketing (roughing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the [INPUT] key.

Item	Description
Right center coordinate X	X coordinate of the center of the right-hand semicircle
Right center coordinate Y	Y coordinate of the center of the right-hand semicircle
Left center coordinate X	X coordinate of the center of the left-hand semicircle
Left center coordinate Y	Y coordinate of the center of the left-hand semicircle
Semicircle radius	Radius of the left- and right-hand semicircles forming the track shape

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

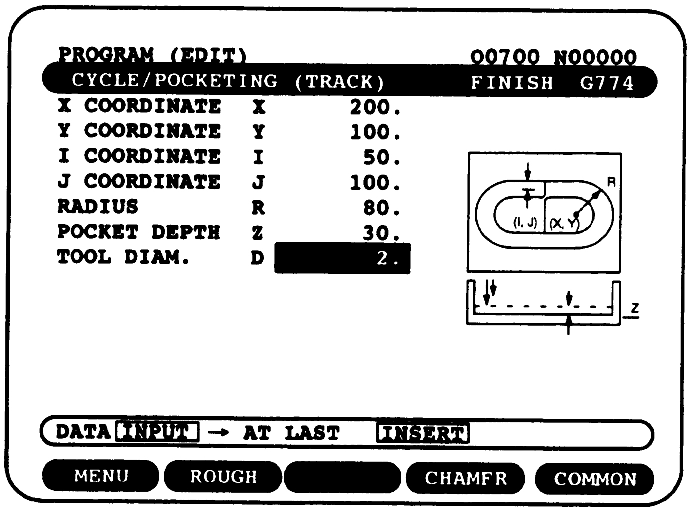
5 Enter the desired value, then press the [INPUT] key.

6 Press the [RETURN] soft key.

Note) When there are no common items needs to be modified, steps 3 through 6 can be skipped.

7 Press the [INSERT] key.

Track pocketing (finishing)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

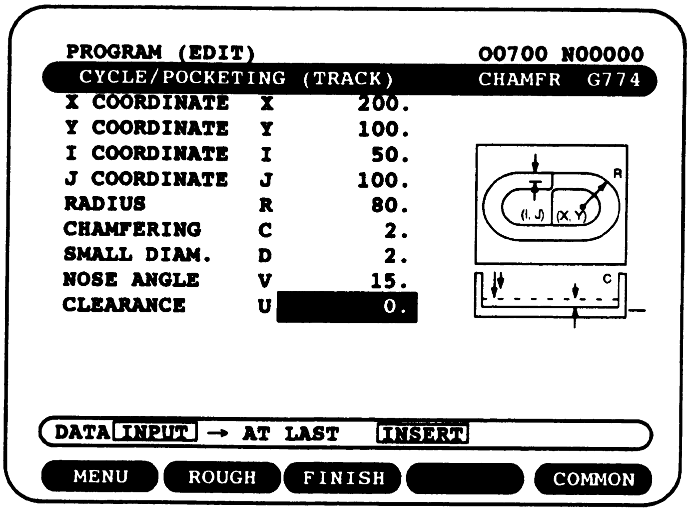
5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

Track pocketing (chamfering)



1 Position the cursor to the item to be specified.

2 Enter the desired value, then press the INPUT key.

3 Press the [COMMON] soft key to display the common item screen.

4 Position the cursor to the item to be specified.

5 Enter the desired value, then press the INPUT key.

6 Press the [RETURN] soft key.

Note) When there are no common items to be modified, steps 3 through 6 can be skipped.

7 Press the INSERT key.

10.3 CREATING PROGRAM A CNC LANGUAGE WITH THE MDI PANEL

Programs can be created in the EDIT mode using the program editing functions described in Chapter 9.

Procedure for Creating a CNC language with the MDI Panel

Procedure

1 Select **EDIT** mode.

2 Press the PROG key or **[PRGRM]** soft key. The program menu screen is displayed.

3 Using the numeric keypad, enter the desired program number.

4 Press the **[EDIT]** soft key.

5 A guidance line for entering a program name is displayed. If required, enter a program name.

6 Press the **[NAME]** soft key.

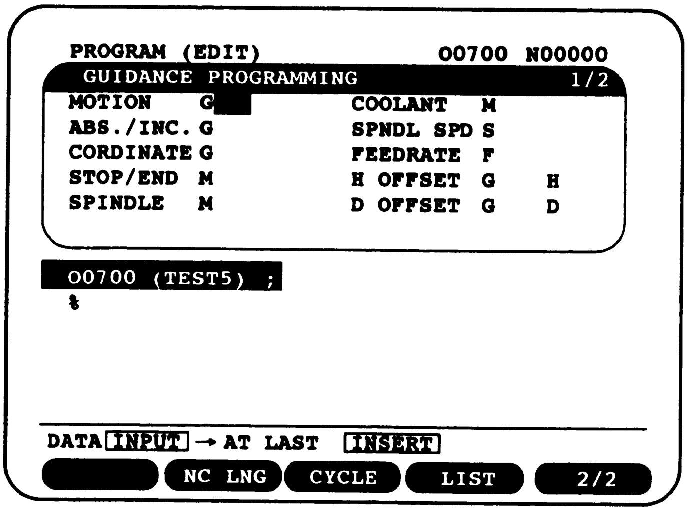
7 The guidance programming screen or CNC language screen is displayed. The screen to be displayed varies with the specification of this parameter:

Bit 0 of parameter 9320 (NCE)

0: Displays the guidance programming screen after a program number is registered.

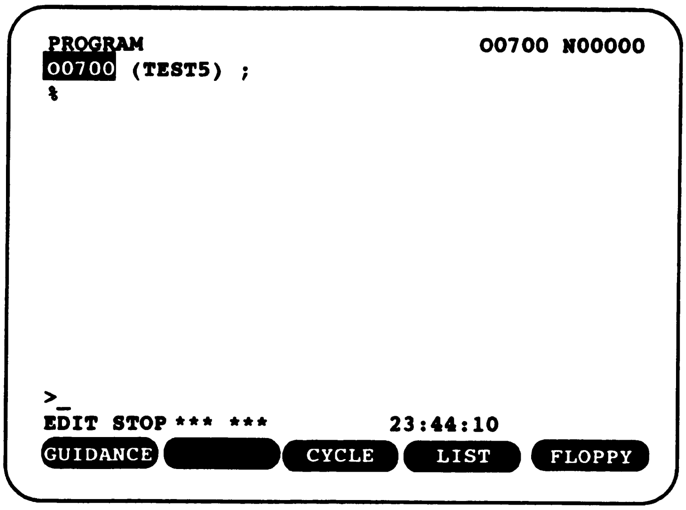
1: Displays the CNC language screen after a program number is registered.

If the NCE bit is set to 0, the guidance programming screen is displayed as shown below:



To display the CNC language screen, press the **[NC LNG]** soft key.

If the NCE bit is set to 1, the CNC language screen is displayed as shown below:



To display the guidance programming screen, press the **[GUIDANCE]** soft key.

8 On the CNC language screen, create a program as described in Chapter 9 (Sections 9.2 to 9.7).

Explanation

• Comments in a program

Comments can be written in a program using the control in/out codes.

Example) O0001 (FANUC SERIES 20) ;
M08 (COOLANT ON) ;

- When the INSERT key is pressed after the control–out code "(", comments, and control–in code ")" have been typed, the typed comments are registered.
- When the INSERT key is pressed midway through comments, to enter the rest of comments later, the data typed before the INSERT key is pressed may not be correctly registered (not entered, modified, or lost) because the data is subject to an entry check which is performed in normal editing.

Note the following to enter a comment:

- Control–in code ")" cannot be registered by itself.
- Comments entered after the INSERT key is pressed must not begin with a number, space, or address O.
- If an abbreviation for a macro is entered, the abbreviation is converted into a macro word and registered (see Section 9.7).
- Address O and subsequent numbers, or a space can be entered but are omitted when registered.

• Detailed procedure for displaying the CNC language screen

For details of the procedure used to display the CNC language screen, see Sections 10.1.1 and 10.1.2.

10.4 AUTOMATIC INSERTION OF SEQUENCE NUMBERS

Sequence numbers can be automatically inserted in each block when a program is created using the MDI keys in the EDIT mode.
Set the increment for sequence numbers in parameter 3216.

Procedure for automatic insertion of sequence numbers

Procedure

1 Set 1 for SEQUENCE NO. (see subjection 11.4.2).

2 Enter the **EDIT** mode.

3 Press PROG to display the program screen.

4 Disply CNC language screen (see section 10.2)

5 Search for or register the number of a program to be edited and move the cursor to the EOB (;) of the block after which automatic insertion of sequence numbers is started.
When a program number is registered and an EOB (;) is entered with the INSERT key, sequence numbers are automatically inserted starting with 0. Change the initial value, if required, according to step 10, then skip to step 7.

6 Press address key N and enter the initial value of N.

7 Press INSERT.

8 Enter each word of a block.

9 Press EOB.

10 Press [INSERT]. The EOB is registered in memory and sequence numbers are automatically inserted. For example, if the initial value of N is 10 and the parameter for the increment is set to 2, N12 inserted and displayed below the line where a new block is specified.

```
PROGRAM                                  O0040 N00012

 O0040 ;
 N10 G92 X0 Y0 Z0 ;
 N12
 %



>_
 EDIT **** *** ***          13 : 18 : 08
[GUIDANCE][        ][ CYCLE ][ LIST ][ FLOPPY]
```

11 • In the example above, if N12 is not necessary in the next block, pressing the [DELETE] key after N12 is displayed deletes N12.

• To insert N100 in the next block instead of N12, enter N100 and press [ALTER] after N12 is displayed. N100 is registered and initial value is changed to 100.

10.5 CREATING PROGRAMS IN TEACH IN MODE

When the playback option is selected, the **TEACH IN JOG** mode and **TEACH IN HANDLE** mode are added. In these modes, a machine position along the X, Y, and Z axes obtained by manual operation is stored in memory as a program position to create a program.
The words other than X, Y, and Z, which include O, N, G, R, F, C, M, S, T, P, Q, and EOB, can be stored in memory in the same way as in **EDIT** mode.

Procedure for Creating Programs in TEACH IN Mode

The procedure described below can be used to store a machine position along the X, Y, and Z axes.

1 Select the **TEACH IN JOG** mode or **TEACH IN HANDLE** mode.

2 Move the tool to the desired position with jog or handle.

3 Press PROG key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the machine position along each axis is to be registered (inserted).

4 Key in address X .

5 Press the INSERT key. Then a machine position along the X axis is stored in memory.

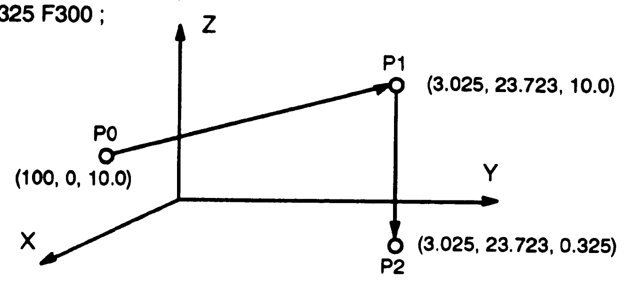
(Example) X10.521 Absolute positon (for mm input)
X10521 Data stored in memory

6 Similarly, key in Y , then press the INSERT key. Then a machine position along the Y axis is stored in memory. Further, key in Z , then press the INSERT key. Then a machine position along the Z axis is stored in memory.

All coordinates stored using this method are absolute coordinates.

Examples

```
O1234 ;
N1 G92 X10000 Y0 Z10000 ;
N2 G00 G90 X3025 Y23723 ;
N3 G01 Z–325 F300 ;
N4 M02 ;
```



1 Set the setting data **SEQUENCE NO.** to 1 (on). (The incremental value parameter (No. 3216) is assumed to be "1".)

2 Select the **TEACH IN HANDLE** mode.

3 Make positioning at position P0 by the manual pulse generator.

4 Select the program screen.

5 Enter program number O1234 as follows:

[O] [1] [2] [3] [4] [INSERT]

This operation registers program number O1234 in memory.
Next, press the following keys:

[EOB] [INSERT]

An EOB (;) is entered after program number O1234. Because no number is specified after N, sequence numbers are automatically inserted for N0 and the first block (N1) is registered in memory.

6 Enter the P0 machine position for data of the first block as follows:

[G] [9] [2] [INSERT] [X] [INSERT] [Y] [INSERT] [Z]
[INSERT] [EOB] [INSERT]

This operation registers G92X10000Y0Z10000; in memory. The automatic sequence number insertion function registers N2 of the second block in memory.

7 Position the tool at P1 with the manual pulse generator.

8 Enter the P1 machine position for data of the second block as follows:

[G] [0] [0] [INSERT] [G] [9] [0] [X] [INSERT] [Y]
[INSERT] [EOB] [INSERT]

This operation registers G00G90X3025Z23723; in memory. The automatic sequence number insertion function registers N3 of the third block in memory.

9 Position the tool at P2 with the manual pulse generator.

10 Enter the P2 machine position for data of the third block as follows:

[G] [0] [1] [INSERT] [Z] [INSERT] [F] [3] [0] [0]
[INSERT] [EOB] [INSERT]

This operation registers G01Z –325F300; in memory.
The automatic sequence number insertion function registers N4 of the fourth block in memory.

11 Register M02; in memory as follows:

[M] [0] [2] [INSERT] [EOB] [INSERT]

N5 indicating the fifth block is stored in memory using the automatic sequence number insertion function. Press the [DELETE] key to delete it.

This completes the registration of the sample program.

Explanations

- **Checking contents of the memory**

The contents of memory can be checked in the **TEACH IN** mode by using the same procedure as in **EDIT** mode.

```
PROGRAM                                     O1234  N00004
   (RELATIVE)                    (ABSOLUTE)
   X    –6.975                   X     3.025
   Y    23.723                   Y    23.723
   Z   –10.325                   Z    –0.325

   O1234 ;
   N1 G92 X10000 Y0 Z10000 ;
   N2 G00 G90 X3025 Y23723 ;
   N3 G01 Z–325 F300 ;
   N4 M02■
   %

 >_
 THND    ****  ***  ***                  14 : 17 : 27
[ PRGRM ][  LIB  ][       ][       ][ (OPRT) ]
```

- **Registering a position with compensation**

When a value is keyed in after keying in address [X], [Y], or [Z], then the [INSERT] key is pressed, the value keyed in for a machine position is added for registration. This operation is useful to correct a machine position by key–in operation.

- **Registering commands other than position commands**

Commands to be entered before and after a machine position must be entered before and after the machine position is registered, by using the same operation as program editing in **EDIT** mode.

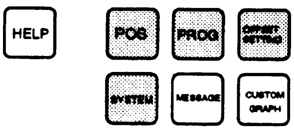
11 SETTING AND DISPLAYING DATA

General

To operate a CNC machine tool, various data must be set on the CRT/MDI panel for the CNC. The operator can monitor the state of operation with data displayed during operation.
This chapter describes how to display and set data for each function.

Explanations

·Screen transition chart



MDI function keys
(Shaded keys ( ) are described in this chapter.)

The screen transition for when each function key on the MDI panel is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.

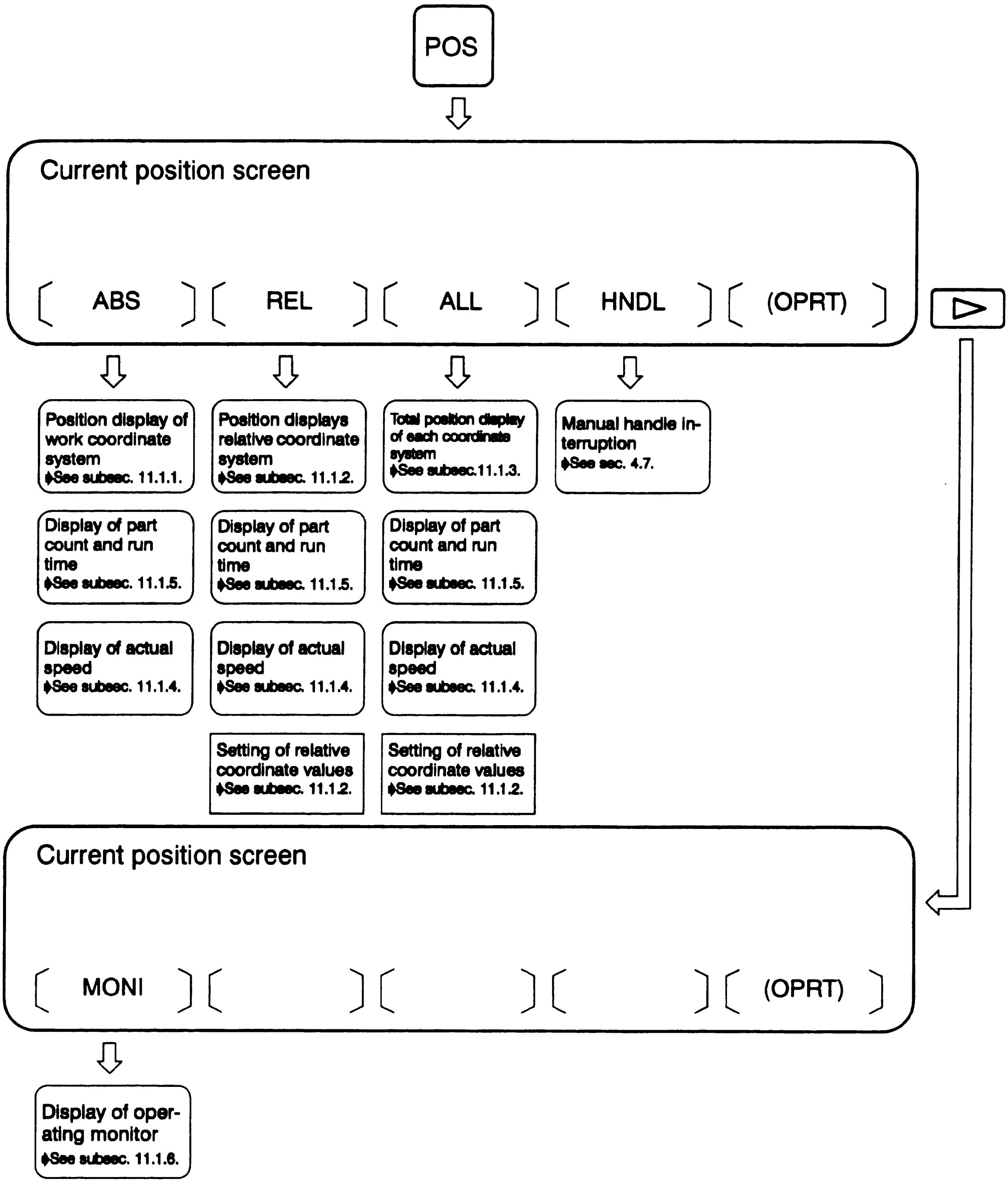
See Chapter 7 for the screen that appears when function key MESSAGE is pressed. See Chapter 12 for the screen that appears when function key HELP is pressed. In general, function key CUSTOM is prepared by the machine tool builder and used for macros. Refer to the manual issued by the machine tool builder for the screen that appears when function key CUSTOM is pressed.

• Data protection key

The machine may have a data protection key to protect part programs, tool compensation values, setting data, and custom macro variables. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

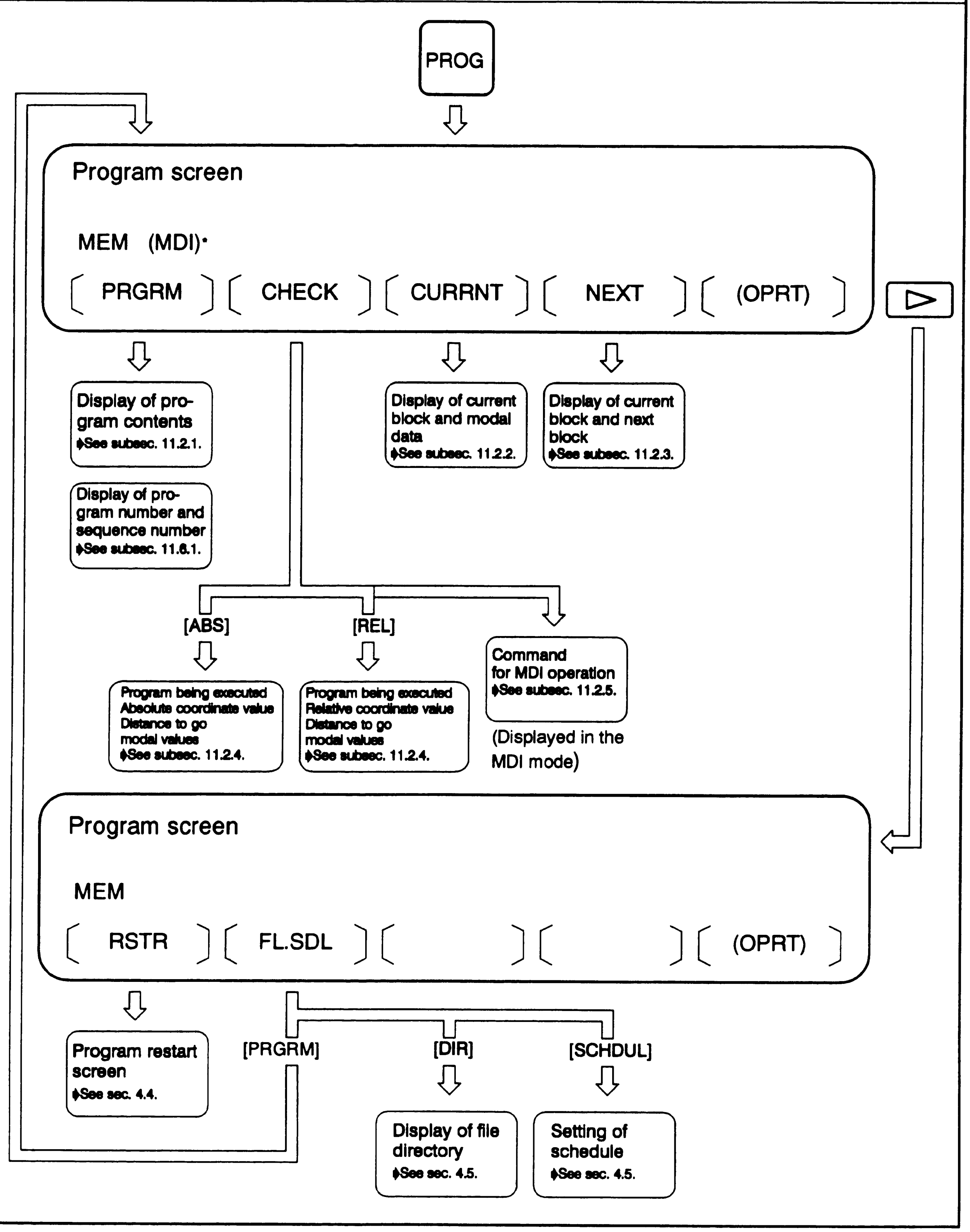
POSITION DISPLAY SCREEN

Screen transition triggered by the function key POS



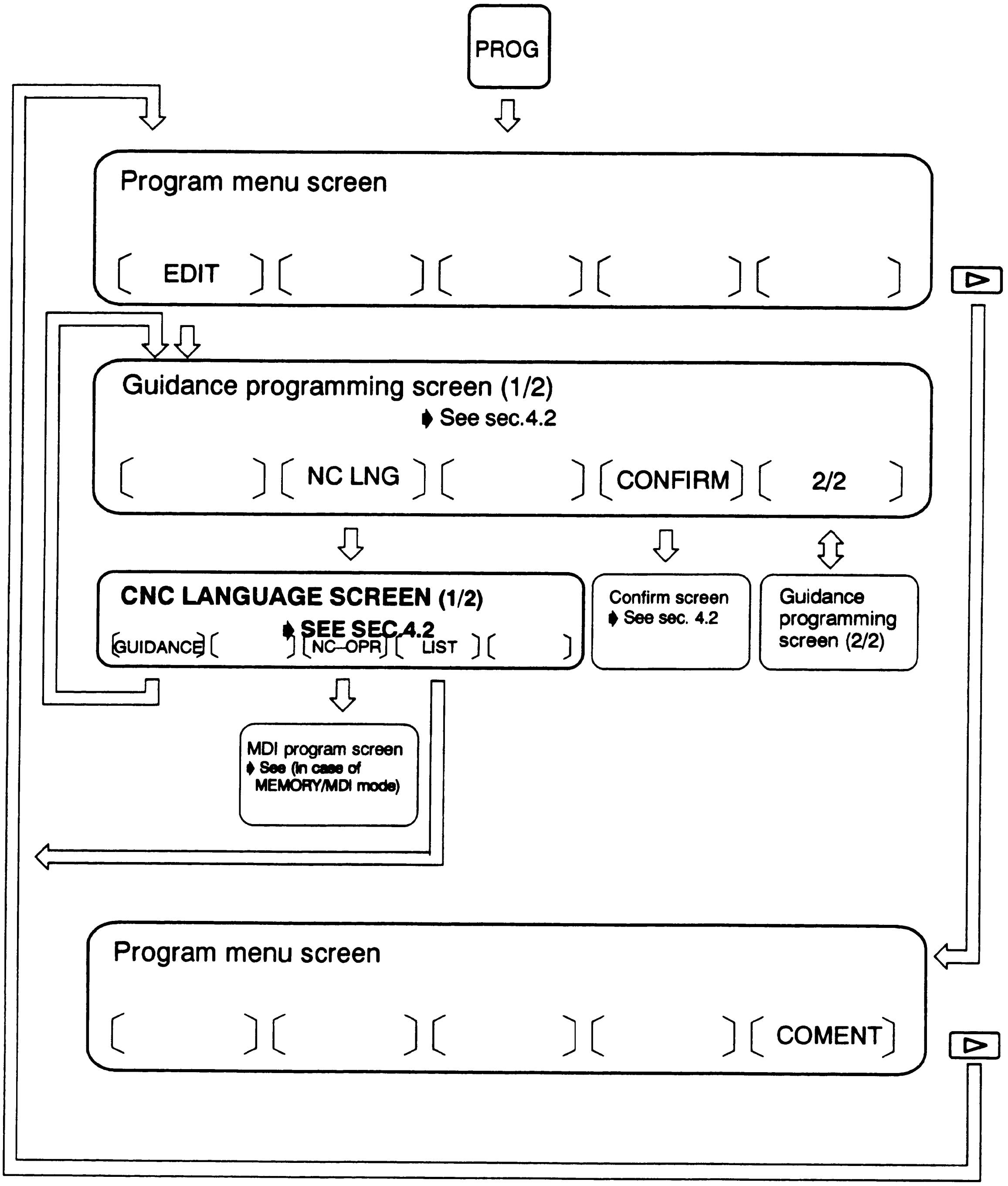
PROGRAM SCREEN

Screen transition triggered by the function key PROG in the MEMORY mode



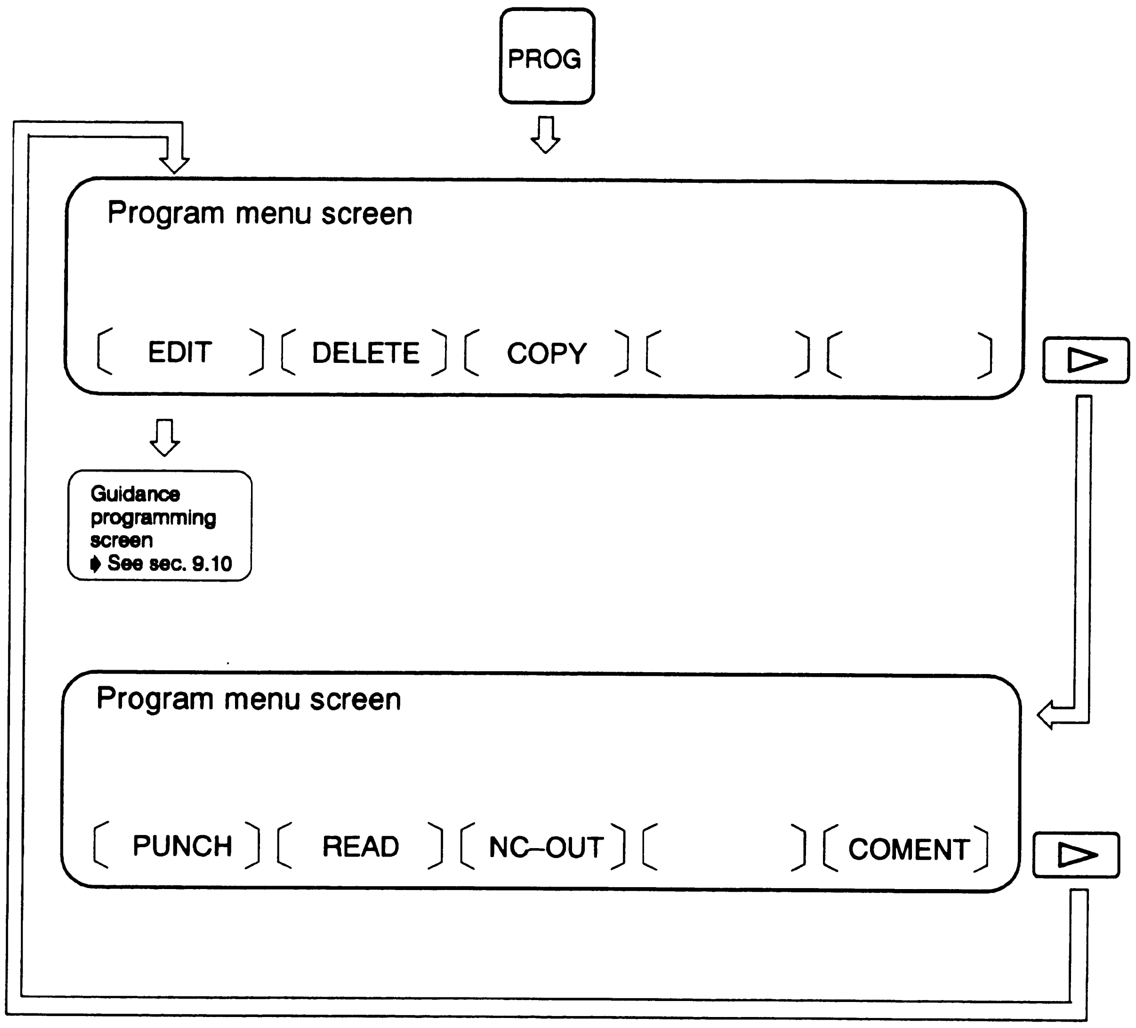
PROGRAM SCREEN

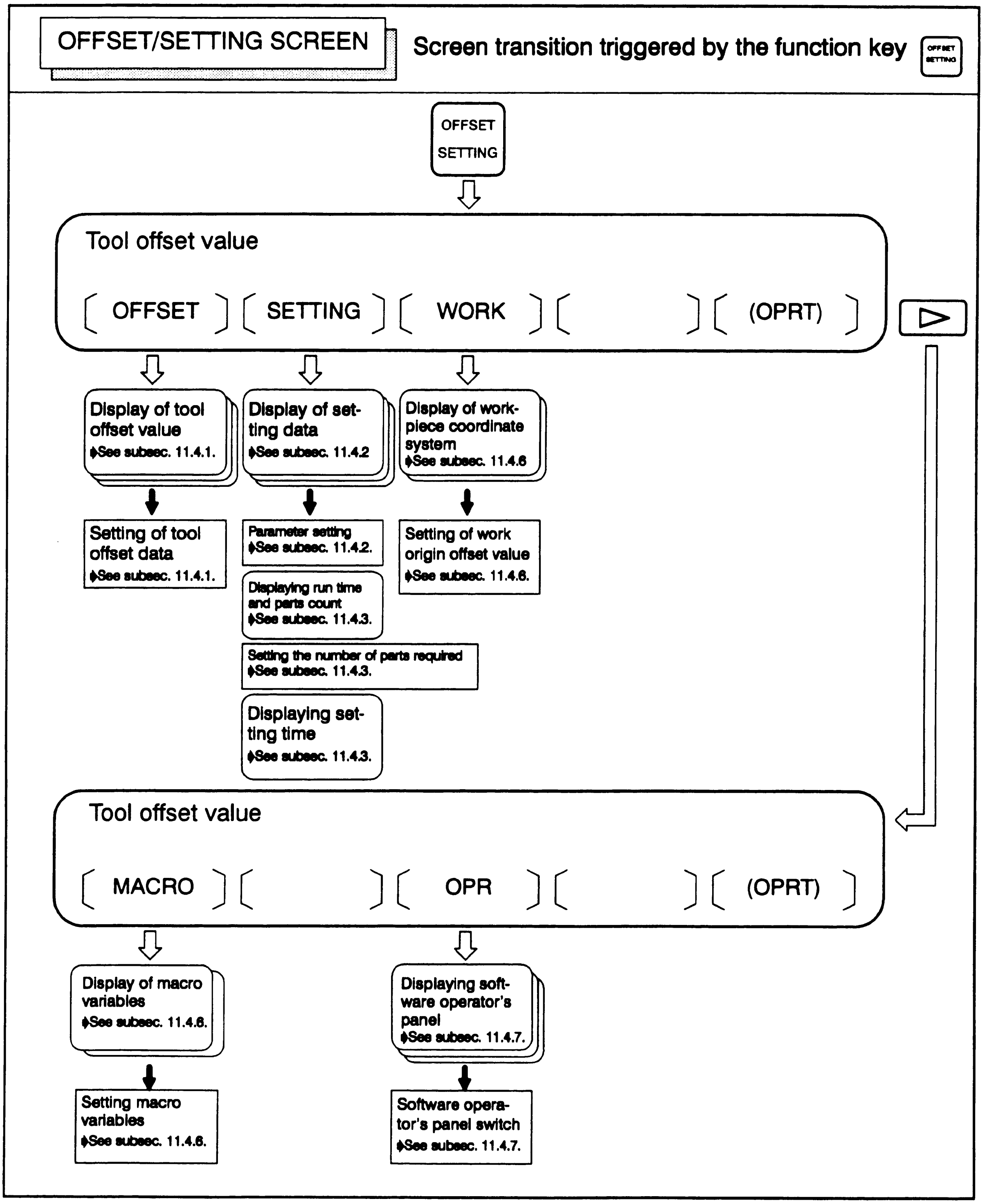
Soft key transition triggered by the function key PROG in the MDI mode



PROGRAM SCREEN

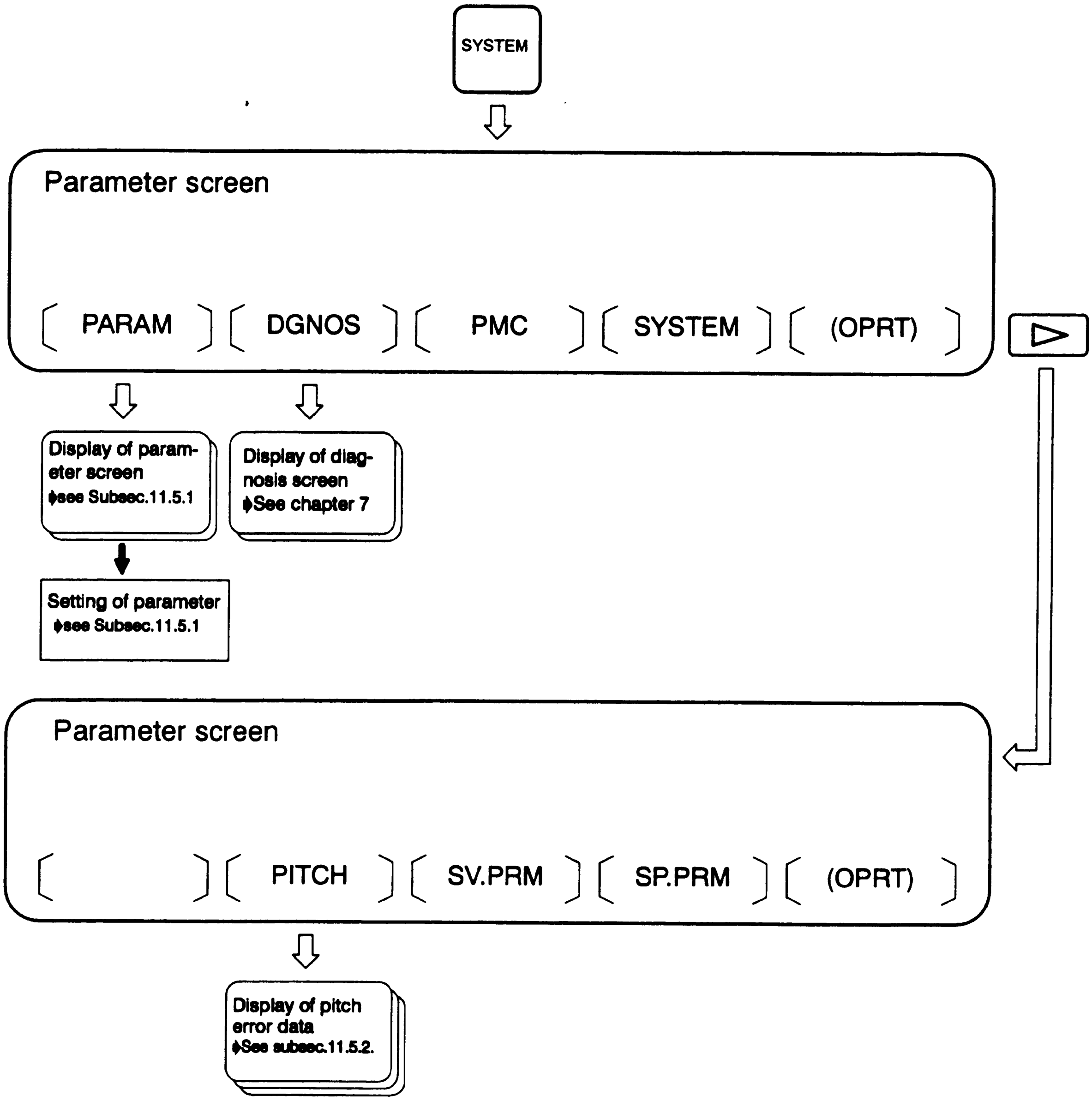
Soft key transition triggered by the function key PROG in the EDIT mode




OFFSET/SETTING SCREEN
Screen transition triggered by the function key
OFFSET SETTING
Tool offset value
OFFSET
SETTING
WORK
(OPRT)
Display of tool offset value
See subsec. 11.4.1.
Display of setting data
See subsec. 11.4.2
Display of workpiece coordinate system
See subsec. 11.4.6
Setting of tool offset data
See subsec. 11.4.1.
Parameter setting
See subsec. 11.4.2.
Setting of work origin offset value
See subsec. 11.4.6.
Displaying run time and parts count
See subsec. 11.4.3.
Setting the number of parts required
See subsec. 11.4.3.
Displaying setting time
See subsec. 11.4.3.
Tool offset value
MACRO
OPR
(OPRT)
Display of macro variables
See subsec. 11.4.6.
Displaying software operator's panel
See subsec. 11.4.7.
Setting macro variables
See subsec. 11.4.6.
Software operator's panel switch
See subsec. 11.4.7.

SYSTEM SCREEN

Screen transition triggered by the function key SYSTEM



- **Setting screens**

The table below lists the data set on each screen.

Table.11. Setting screens and data on them

No.	Setting screen	Contents of setting	Reference item
1	Tool offset value	Tool length offset value Cutter compensation value	Subsec. 11.4.1
2	Setting data(handy)	Parameter write TV check Punch code Input unit (mm/inch) I/O channel Automatic insert of Sequence No.	Subsec. 11.4.2
3	Setting data (mirror image)	Mirror image	Subsec. 11.4.2
4	Setting data (timer)	Parts required	Subsec. 11.4.3
5	Macro variables	Custom macro common variables (#100 to #149) (#500 to #531)	Subsec. 11.4.6
6	Parameter	Parameter	Subsec. 11.5.1
7	Pitch error	Pitch error compensation data	Subsec. 11.5.2
8	Software operator's panel	Mode selection Jog feed axis selection Jog rapid traverse Axis selection for Manual pulse generator Multiplication for manual pulse generator Jog feedrate Feedrate override Rapid traverse override Optional block skip Single block Machine lock Dry run Protect key Feed hold	Subsec. 11.4.7
9	Work coordinate system	Workpiece origin offset value	Subsec. 11.4.4

11.1 SCREENS DISPLAYED BY FUNCTION KEY POS

Press function key POS to display the current position of the tool.

The following three screens are used to display the current position of the tool:

·Position display screen for the work coordinate system.
·Position display screen for the relative coordinate system.
·Overall position display screen.

The above screens can also display the feedrate, run time, and the number of parts.

Function key POS can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key POS can also be used to display the screen for displaying the distance moved by handle interruption. See Section 4.7 for details on this screen.

11.1.1 Position Display in the Work Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate system

1 Press function key POS .

2 Press soft key **[ABS]**.

```
ACTUAL POSITION(ABSOLUTE)          O1000 N00010

   X          123.456
   Y          363.233
   Z            0.000

                          PART COUNT        5
 RUN TIME    0H15M        CYCLE TIME   0H 0M38S
 ACT.F     3000 MM/M              S    0 T0000

MEM STRT MTN ***           09:06:35
[ ABS ] [  REL  ] [  ALL  ] [  HNDL ] [ OPRT ]
```

Explanations

- **Display including compensation values**

Bits 6 and 7 of parameter 3104 can be used to select whether the displayed values include tool length offset and cutter compensation.

11.1.2 Position Display in the Relative Coordinate System

Displays the current position of the tool in a relative coordinate system based on the coordinates set by the operator. The current position changes as the tool moves. The increment system is used as the unit for numeric values. The title at the top of the screen indicates that relative coordinates are used.

Display procedure for the current position screen with the relative coordinate system

1 Press function key POS .

2 Press soft key **[REL]**.

```
ACTUAL POSITION(RELATIVE)          O1000 N00010

   X        123.456
   Y        363.233
   Z          0.000

                              PART COUNT       5
 RUN TIME    0H15M            CYCLE TIME   0H 0M38S
 ACT.F      3000 MM/M                  S   0 T0000

MEM STRT MTN ***              09:06:35
[ ABS  ] [ REL  ] [ ALL  ] [ HNDL ] [ OPRT ]
```

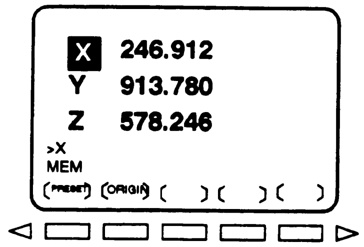
See Explanations for the procedure for setting the coordinates.

Explanations

• Setting the relative coordinates

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows:

Procedure to set the axis coordinate to a specified value

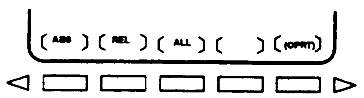


1 Enter an axis address (such as X or Y) on the screen for the relative coordinates. The indication for the specified axis blinks and the soft keys change as shown on the left.

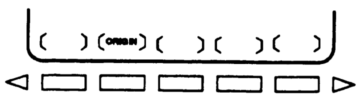
2 · To reset the coordinate to 0, press soft key **[ORGIN]**. The relative coordinate for the blinking axis is reset to 0.

· To preset the coordinate to a specified value, enter the value and press soft key **[PRESET]**. The relative coordinate for the blinking axis is set to the entered value.

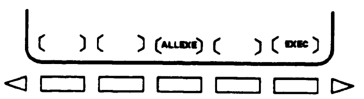
Procedure to reset all axes



1 Press soft key **[(OPRT)]**.



2 Press soft key **[ORIGIN]**.



3 Press soft key **[ALLEXE]**.
The relative coordinates for all axes are reset to 0.

• Display including compensation values

Bits 6 and 7 of parameter 3104 can be used to select whether the displayed values include tool length offset and cutter compensation.

• Presetting by setting a coordinate system

Bit 3 of parameter 3104 is used to specify whether the displayed positions in the relative coordinate system are preset to the same values as in the workpiece coordinate system when a coordinate system is set by a G92 command or when the manual reference position return is made.

11.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance. The relative coordinates can also be set on this screen. See subsection 11.1.2 for the procedure.

Procedure for displaying overall position display screen

1 Press function key [POS] .

2 Press soft key **[ALL]**.

```
ACTUAL POSITION                        O1000 N00010
  (RELATIVE)                    (ABSOLUTE)
 X   246.912                   X   123.456
 Y   913.780                   Y   456.890
 Z  1578.246                   Z   789.123

  (MACHINE)                    (DISTANCE TO GO)
 X      0.000                  X      0.000
 Y      0.000                  Y      0.000
 Z      0.000                  Z      0.000
                                PART COUNT        5
 RUN TIME    0H15M              CYCLE TIME   0H 0M38S
 ACT.F      3000 MM/M                 S    0 T0000

MEM **** *** ***               09:06:35
[  ABS  ] [  REL  ] [  ALL  ] [  HNDL  ] [  OPRT  ]
```

Explanations

• Coordinate display

The current positions of the tool in the following coordinate systems are displayed at the same time:
·Current position in the relative coordinate system (relative coordinate)
·Current position in the work coordinate system (absolute coordinate)
·Current position in the machine coordinate system (machine coordinate)
·Distance to go (distance to go)

• Distance to go

The distance remaining is displayed in the MEMORY or MDI mode. The distance the tool is yet to be moved in the current block is displayed.

• Machine coordinate system

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter 3104.

11.1.4 Actual Feedrate Display

The actual feedrate on the machine (per minute) can be displayed on a current position display screen or program check screen by setting bit 0 (DPF) of parameter 3105.

Display procedure for the actual feedrate on the current position display screen

1 Press function key [POS] to display a current position display screen.

```
ACTUAL POSITION(ABSOLUTE)        O1000 N00010

  X        123.456
  Y        363.233
  Z          0.000

                           PART COUNT       5
 RUN TIME    0H15M         CYCLE TIME   0H 0M38S
 ACT.F      3000 MM/M              S     0 T0000

MEM STRT MTN ***           09:06:35
[ ABS ] [ REL  ] [  ALL  ] [  HNDL ] [ OPRT ]
```

Actual feedrate is displayed after ACT.F.

Explanations

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

• Actual feedrate value

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n}(fi)^2}$$

where

n : Number of axes

fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate

Fact : Actual feedrate displayed

The display unit: mm/min (metric input).
inch/min (Inch input, Two digits below the decimal point are displayed.)

The feedrate along the PMC axis can be omitted by setting bit 1 (PCF) of parameter 3015.

• Actual feedrate display on the other screen

The program check screen also displays the actual feedrate.

11.1.5 Display of Run Time and Parts Count

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

1 Press function key [POS] to display a current position display screen.

```
ACTUAL POSITION(RELATIVE)        O1000 N00010

   X        123.456
   Y        363.233
   Z          0.000

                            PART COUNT        5
RUN TIME     0H15M          CYCLE TIME   0H 0M38S
ACT.F     3000 MM/M                  S    0 T0000

MEM STRT MTN ***             09:06:35
[ ABS ] [ REL ] [ ALL ] [ HNDL ] [ OPRT ]
```

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanations

- **PART COUNT**

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter 6710 is executed.

- **RUN TIME**

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- **CYCLE TIME**

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- **Display on the other screen**

Details of the run time and the number of machined parts are displayed on the setting screen. See subsection 11.4.5.

- **Parameter setting**

The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters No. 6711, 6751, and 6752 or on the setting screen.

- **Incrementing the number of machined parts**

Bit 0 (PCM) of parameter No. 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

11.1.6 Operating Monitor Display

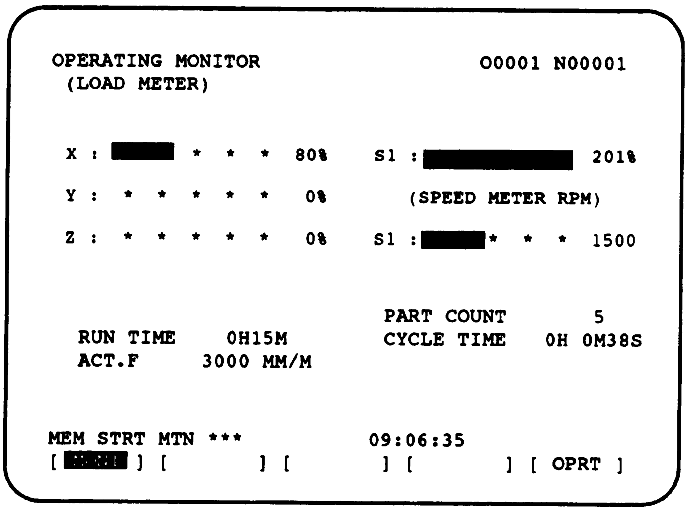
The reading on the load meter can be displayed for each servo axis and the serial spindle by setting bit 5 (OPM) of parameter 3111 to 1. The reading on the speedometer can also be displayed for the serial spindle.

Procedure for displaying the operating monitor

1 Press function key [POS] to display a current position display screen.

2 Press the continuous–menu key [▷].

3 Press soft key **[MONI]**.



Explanations

- **Display of the servo axes**

The reading on the load meter can be displayed for up to three servo axes by setting parameters 3151 to 3153.

- **Display of the spindle axes**

When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- **Unit of graph**

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%). The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- **Load meter**

The reading on the load meter depends on servo parameter 2086 and spindle parameter 4127.

- **Speedometer**

Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter 3111 to 1.

11.2 SCREENS DISPLAYED BY FUNCTION KEY PROG (IN MEMORY MODE OR MDI MODE)

This section describes the screens displayed by pressing function key PROG in MEMORY or MDI mode.The first four of the following screens display the execution state for the program currently being executed in MEMORY or MDI mode and the last screen displays the command values for MDI operation in the MDI mode:

1. Program contents display screen
2. Current block display screen
3. Next block display screen
4. Program check screen
5. Program screen for MDI operation
6. Stamping the machining time

Function key PROG can also be pressed in MEMORY mode to display the program restart screen and scheduling screen.
See Section 4.4 for the program restart screen.
See Section 4.5 for the scheduling screen.

11.2.1 Program Contents Display

Displays the program currently being executed in MEMORY or MDI mode.

Procedure for displaying the program contents

1 Press function key PROG to display the program screen.

2 Press chapter selection soft key **[PRGRM]**.
The cursor is positioned at the block currently being executed.

```
PROGRAM                                   O2000 N00130
O2000 ;
N100 G92 X0 Y0 Z70. ;
N110 G91 G00 Y-70. ;
N120 Z-70. ;
N130 G42 G39 I-17.5
N140 G41 G03 X-17.5 Y17.5 R17.5 ;
N150 G01 X-25. ;
N160 G02 X27.5 Y27.5 R27.5
N170 G01 X20. ;
N180 G02 X45. Y45. R45. ;
> _                                    S    0  T0000
MEM STRT    ***         16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

11.2.2 Current Block Display Screen

Displays the block currently being executed and modal data in the MEMORY or MDI mode.

Procedure for displaying the current block display screen

1 Press function key PROG .

2 Press chapter selection soft key **[CURRNT]**.
The block currently being executed and modal data are displayed.
The screen displays up to 22 modal G codes and up to 11 G codes specified in the current block.

```
PROGRAM                                         O2000 N00130

          (CURRNT)              (MODAL)

   G01  X    17.500  G67   G01   F  2000
   G17  F      2000  G54   G17
   G41  H         2  G64   G91
   G80               G69   G22
                     G15   G94
                     G25  .1G21  H  2  D
                           G41
                           G49   T
                           G80
                           G98   S
                           G50
> _                                      S    0   T0000
MEM STRT     ***          16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

11.2.3 Next Block Display Screen

Displays the block currently being executed and the block to be executed next in the MEMORY or MDI mode.

Procedure for displaying the next block display screen

1 Press function key PROG .

2 Press chapter selection soft key **[NEXT]**.
The block currently being executed and the block to be executed next are displayed.
The screen displays up to 11 G codes specified in the current block and up to 11 G codes specified in the next block.

```
PROGRAM                                   O2000 N00130

         (CURRNT)          (NEXT)
     G01  X    17.500   G39   I     –17.500
     G17  F      2000   G42
     G41  H         2
     G80




>_                                        S   0  T0000
MEM STRT    ***        16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

11.2.4 Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data in the MEMORY mode.

Procedure for displaying the program check screen

1 Press function key PROG .

2 Press chapter selection soft key **[CHECK]**.
The program currently being executed, current position of the tool, and modal data are displayed.

```
PROGRAM                                         O2000 N00130

 O0010
 G92 G90 X100. Y200. Z50. ;
 G00 X0 Y0 Z0 ;
 G01 Z250. F1000 ;
 (ABSOLUTE)(DIST TO GO)  G00  G94  G80
 X    0.000 X    0.000  G17  G21  G98
 Y    0.000 Y    0.000  G90  G40  G50
 Z    0.000 Z    0.000  G22  G49  G67
                                 B
                            H    M
     T                      D
     F              S

> _                                          S    0  T0000
MEM STRT    ***          16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

Explanations

- **Program display**

The screen displays up to four blocks of the current program, starting from the block currently being executed. The block currently being executed is displayed in reverse video. During DNC operation, however, only three blocks can be displayed.

- **Current position display**

The position in the workpiece coordinate system or relative coordinate system and the remaining distance are displayed. The absolute positions and relative positions are switched by soft keys **[ABS]** and **[REL]**.

- **Modal G codes**

Up to 12 modal G codes are displayed.

- **Display during automatic operation**

During automatic operation, the actual speed, SACT is displayed. The key input prompt (>_) is displayed otherwise.

- **T codes**

Then bit 2 (PCT) of parameter No. 3108 is set to 1, the T codes specified with the PMC (HD.T/NX.T) are displayed instead of those specified in the program. Refer to the FANUC PMC Programming Manual (B–61863E) for details of HD.T/NX.T.

11.2.5 Program Screen for MDI Operation

Displays the program input from the MDI and modal data in the MDI mode.

Procedure for displaying the program screen for MDI operation

Procedure

1 Press function key PROG.

2 Press soft key **[EDIT]**.

3 Press soft key **[NC LNG]**. Then CNC language screen is displayed.

4 Press soft key **[NC OPR]**. The program input from the MDI and model data are displayed.

Program

Modal information

```
PROGRAM  (MDI)                              O2000 N00130

O0000 G00 X100.0 Z200.0 ;
M03 ;
G01 Z120.0 F500 ;
M98 P9010 ;
G00 Z0.0 ;
■

G00  G90  G94  G40  G80  G50  G54  G69
G17  G22  G21  G49  G98  G67  G64  G15
                              H    M
     T                        D
     F                   S

> _                                  S    0  T0000
MDI **** *** ***        16:05:59
[ PRGRM ][ MDI ][ CURRNT ][ NEXT ][ (OPRT) ]
```

Explanations

- **MDI operation**

See Section 4.2 for MDI operation.

- **Modal information**

The modal data is displayed when bit 7 (MDL) of parameter 3107 is set to 1. Up to 16 modal G codes are displayed.

- **Displaying during automatic operation**

During automatic operation, the actual speed is displayed. The key input prompt (>_) is displayed otherwise.

11.3 SCREENS DISPLAYED BY FUNCTION KEY PROG (IN THE EDIT MODE)

This section describes the screens displayed by pressing function key PROG in the EDIT mode. Function key PROG in the EDIT mode can display the program editing screen and the program library list screen (displays memory used and a list of programs). Pressing function key PROG in the EDIT mode can also display the floppy file directory screen. See Chapter 9 for the program editing screen. See Chapter 8 for the floppy file directory screen.

11.3.1 Displaying Memory Used and a List of Programs

Displays the number of registered programs, memory used, and a list of registered programs.

Procedure for displaying memory used and a list of programs

1 Select the **EDIT** mode.

2 Press function key PROG .

```
PROGRAM  ( EDIT )                          O0100 N00000
 ** PROGRAM MENU **
PROGRAM NO.  USED/FREE                       67/     57
MEMORY AREA  USED/FREE                      560/   3200

 O1002 (NO. 4568-256-47893)
 O0100 (#23546-21)
 O0110 (TEST/12[7]-54)
 O5230 (PROG.SAMPLE(CORNER[1])
 O9654 (SAMPLE-1[ARC])
 O3243 (PN4567891-12/TA)


 PROGRAM NO.   >  100
INPUT NO. OR SELECT MENU BY CURSOR                  1/2

 EDIT    DELETE    COPY
```

Explanations

- **Details of memory used**

PROGRAM NO. USED

PROGRAM NO. USED : The number of the programs registered (including the subprograms)

FREE : The number of programs which can be registered additionally.

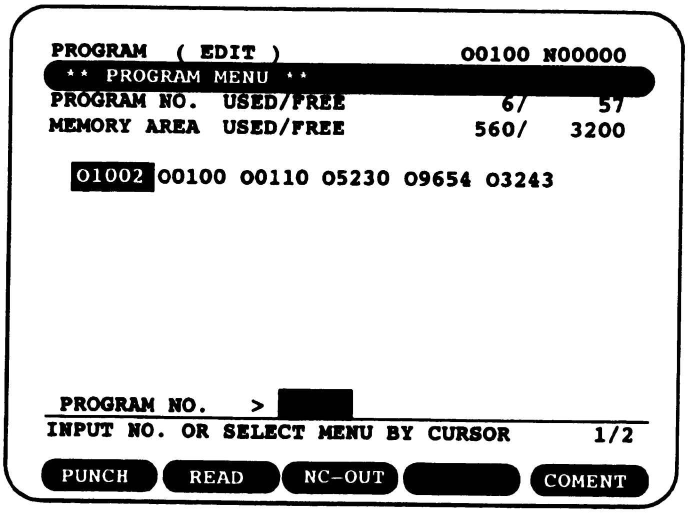
MEMORY AREA USED

MEMORY AREA USED : The capacity of the program memory in which data is registered (indicated by the number of characters).

FREE : The capacity of the program memory which can be used additionally (indicated by the number of characters).

- **Program library list**

Program Nos. registered are indicated.
Pressing the **[COMENT]** soft key on the program list screen hides the program names on the screen.



- **Program name**

Up to 23 characters can be used for naming a program within the parentheses.
Only program number is displayed for the program without any program name.

O □□□□ (ΔΔΔΔ...Δ) ;

Program number Program name (up to 23 characters)

- **Order in which programs are displayed in the program library list**

Programs are displayed in the same order that they are registered in the program library list. However, if bit 4 (SOR) of parameter 3107 is set to 1, programs are displayed in the order of program number starting from the smallest one.

• Order in which programs are registered

If no program is registered, each program is registered after the last program in the list.
If some programs in the list were deleted, then a new program is registered, the new program is inserted in the empty location in the list created by the deleted programs.

Example) When bit 4 (SOR) of parameter 3107 is 0

1. **After clearing all programs, register programs O0001, O0002, O0003, O0004, and O0005 in this order. The program library list displays the programs in the following order:**
   O0001, O0002, O0003, O0004, O0005

2. **Delete O0002 and O0004. The program library list displays the programs in the following order:**
   O0001, O0003, O0005

3. **Register O0009. The program library list displays the programs in the following order:**
   O0001, O0009, O0003, O0005

11.4 SCREENS DISPLAYED BY FUNCTION KEY OFFSET SETTING

Press function key OFFSET SETTING to display or set tool compensation values and other data.
This section describes how to display or set the following data:

1. Tool offset value
2. Settings
3. Run time and part count
4. Workpiece origin offset value
5. Custom macro common variables
6. Software operator's panel

The software operator's panel depends on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

11.4.1 Displaying and Setting the Tool Offset Value

Tool length offset values, and cutter compensation values are specified by D codes or H codes in a program. Compensation values corresponding to D codes or H codes are displayed or set on the screen.

Procedure for setting and displaying the cutter compensation value

1 Press function key [OFFSET SETTING].

2 Press chapter selection soft key **[OFFSET]** or press [OFFSET SETTING] several times until the tool compensation screen is displayed.

```
OFFSET                                   O0001 N00000
   NO.        DATA         NO.       DATA
   001        1.000        009            0.000
   002       -2.000        010           -7.500
   003        0.000        011           12.000
   004        5.000        012          -20.000
   005        0.000        013            0.000
   006        0.000        014            0.000
   007        0.000        015            0.000
   008        0.000        016            0.000
ACTUAL POSITION (RELATIVE)
      X      0.000          Y            0.000
      Z      0.000

> _
MDI **** *** ***          16:05:59
[ OFFSET ][ SETING ][ WORK ][      ][ (OPRT) ]
```

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key **[NO.SRH]**.

4 To set a compensation value, enter a value and press soft key **[INPUT]**. To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key **[+INPUT]**. Or, enter a new value and press soft key **[INPUT]**.

Explanations

• Decimal point input

A decimal point can be used when entering a compensation value.

• Other method

An external input/output device can be used to input or output a cutter compensation value. See Chapter 8. A tool length offset value can be set by measuring the tool length as described in the next subsection.

- **Disabling entry of compensation values**

The entry of compensation values may be disabled by setting bit 0 (WOF) of parameter 3290.
And then, the input of tool compensation values from the MDI can be inhibited for a specified range of offset numbers. The first offset number for which the input of a value is inhibited is set in parameter No. 3294. The number of offset numbers, starting from the specified first number, for which the input of a value is inhibited is set in parameter No. 3295.
Consecutive input values are set as follows:

1) When values are input for offset numbers, starting from one for which input is not inhibited to one for which input is inhibited, a warning is issued and values are set only for those offset numbers for which input is not inhibited.
2) When values are input for offset numbers, starting from one for which input is inhibited to one for which input is not inhibited, a warning is issued and no values are set.

11.4.2 Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing.
See Section 10.2 for automatic insertion of sequence numbers.
This subsection describes how to set data.

Procedure for setting the setting data

Procedure

1 Select the **MDI** mode.

2 Press function key OFFSET SETTING.

3 Press soft key **[SETING]** to display the setting data screen.
This screen consists of several pages.
Press page key PAGE ↑ or PAGE ↓ until the desired screen is displayed.
An example of the setting data screen is shown below.

```
SETTING (HANDY)                              O0001 N00000

  PARAMETER WRITE   =1 (0:DISABLE 1:ENABLE)
  TV CHECK          = 0 (0:OFF    1:ON)
  PUNCH CODE        = 1 (0:EIA    1:ISO)
  INPUT UNIT        = 0 (0:MM     1:INCH)
  I/O CHANNEL       =  0 (0–2:CHANNEL NO.)
  SEQUENCE NO.      = 0 (0:OFF    1:ON)




> _
MDI **** *** ***          16:05:59
[ OFFSET ][ SETING ][ WORK ][      ][ (OPRT) ]
```

```
SETTING (HANDY)                              O0001 N00000

  MIRROR IMAGE X   =0 (0:OFF    1:ON)
  MIRROR IMAGE Y   = 0 (0:OFF    1:ON)
  MIRROR IMAGE Z   = 0 (0:OFF    1:ON)




> _
MDI **** *** ***          16:05:59
[ OFFSET ][ SETING ][ WORK ][      ][ (OPRT) ]
```

4 Move the cursor to the item to be changed by pressing cursor keys [↑], [↓], [←], or [→].

5 Enter a new value and press soft key **[INPUT]**.

Contents of settings

- **PARAMETER WRITE**

Setting whether parameter writing is enabled or disabled.
0 : Disabled
1 : Enabled

- **TV CHECK**

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- **PUNCH CODE**

Setting code when data is output through reader puncher interface.
0 : EIA code output
1 : ISO code output

- **INPUT UNIT**

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- **I/O CHANNEL**

Using channel of reader/puncher interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2

- **SEQUENCE STOP**

Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
0 : Does not perform automatic sequence number insertion.
1 : Perform automatic sequence number insertion.

- **MIRROR IMAGE**

Setting of mirror image ON/OFF for each axes.
0 : Mirror image off
1 : Mirror image on

- **Others**

Page key [↑ PAGE] or [PAGE ↓] can also be pressed to display the SETTING (TIMER) screen. See subsection 11.4.3 for this screen.

11.4.3 Displaying and Setting Run Time,Parts Count, and Time

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed. This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Procedure for Displaying and Setting Run Time, Parts Count and Time

1 Select the MDI mode.

2 Press function key [OFFSET SETTING].

3 Press chapter selection soft key **[SETING]**.

4 Press page key [PAGE ↑] or [PAGE ↓] several times until the following screen is displayed.

```
SETTING (TIMER)                             O0001 N00000

     PARTS TOTAL       =         14
     PARTS REQUIRED    =          0
     PARTS COUNT       =         23

     POWER ON          =      4H 31M
     OPERATING TIME    =      0H  0M   0S
     CUTTING TIME      =      0H 37M   5S
     FREE PURPOSE      =      0H  0M   0S
     CYCLE TIME        =      0H  0M   0S
                  DATE =  1994/07/05
                  TIME =    11:32:52

> _                                         S   0  T0000
MDI **** *** ***          16:05:59
[ OFFSET ][ SETING ][ WORK ][        ][ (OPRT) ]
```

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.

6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press soft key **[INPUT]**.

Display items

• PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter 6710 is executed. This value cannot be set on this screen. Set the value in parameter 6712.

• PARTS REQUIRED

It is used for setting the number of machined parts required.
When the "0" is set to it, there is no limitation to the number of parts. Also, its setting can be made by the parameter (NO. 6713).

• PARTS COUNT

This value is incremented by one when M02, M30, or an M code specified by parameter 6710 is executed. The value can also be set by parameter 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

• POWER ON

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter 6750.

• OPERATING TIME

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter 6751 or 6752.

• CUTTING TIME

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter 6753 or 6754.

• FREE PURPOSE

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

• CYCLE TIME

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

• DATA and TIME

Displays the current date and time. The date and time can be set on this screen.

Limitations

• Usage

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter (NO. 6710) is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (parameter PCM (No. 6700#0) is set to 1). For details, see the manual issued by machine tool builders.

Restrictions

• Run time and part count settings

Negative value cannot be set. Also, the setting of "M" and "S" of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

• Time settings

Neither negative value nor the value exceeding the value in the following table can be set.

Item	Maximum value	Item	Maximum value
Year	2085	Hour	23
Month	12	Minute	59
Day	31	Second	59

11.4.4 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59) and external workpiece origin offset. The workpiece origin offset and external workpiece origin offset can be set on this screen.

Procedure for Displaying and Setting the Workpiece Origin Offset Value

1 Press function key OFFSET SETTING.

2 Press chapter selection soft key **[WORK]**.
The workpiece coordinate system setting screen is displayed.

```
WORK COORDINATES                          O0001 N00000

 (G54)
  NO.        DATA                  NO.        DATA
  00    X       0.000              02    X    152.580
 (EXT)  Y       0.000             (G55)  Y    234.000
        Z       0.000                    Z    112.000

  01    X      20.000              03    X    300.000
 (G54)  Y      50.000             (G56)  Y    200.000
        Z      30.000                    Z    189.000

>_                                       S    0  T0000
MDI **** *** ***          16:05:59
[ OFFSET ][ SETING ][ WORK ][      ][ (OPRT) ]
```

3 The screen for displaying the workpiece origin offset values consists of two or more pages. Display a desired page in either of the following two ways:

Press the page up PAGE ↑ or page down PAGE ↓ key.

Enter the workpiece coordinate system number (0: external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59) and press operation selection soft key **[NO.SRH]**.

4 Turn off the data protection key to enable writing.

5 Move the cursor to the workpiece origin offset to be changed.

6 Enter a desired value by pressing numeric keys, then press soft key **[INPUT]**. The entered value is specified in the the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing soft key **[+INPUT]**, the entered value can be added to the previous offset value.

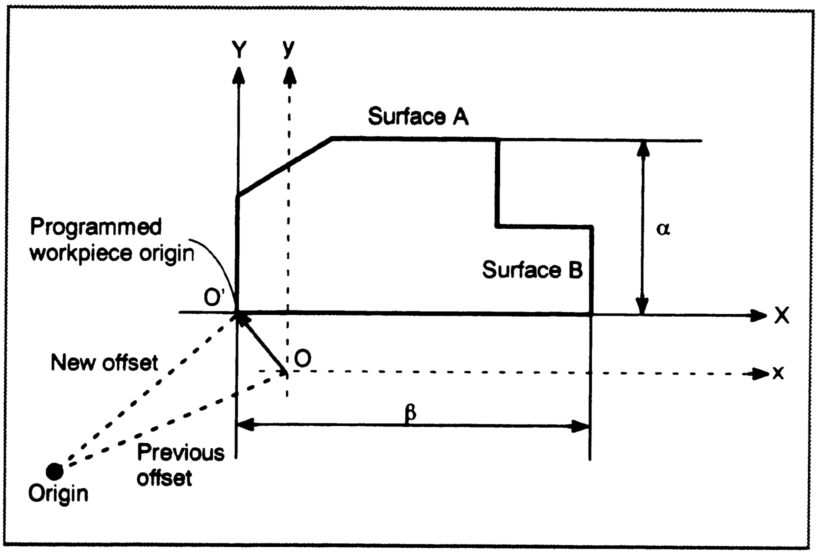
7 Repeat **5** and **6** to change other offset values.

8 Turn on the data protection key to disable writing.

11.4.5 Input of measured workpiece origin offsets

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen such that the command values match the actual dimensions.
Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

Procedure for Inputting of Measured Workpiece Origin Offsets



1 When the workpiece is shaped as shown above, position the reference tool manually until it touches surface A of the workpiece.

2 Retract the tool without changing the Y coordinate.

3 Measure distance α between surface A and the programmed origin of the workpiece coordinate system as shown above.

4 Press function key [OFFSET SETTING] .

5 To display the workpiece origin offset setting screen, press the chapter selection soft key **[WORK]**.

```
WORK COORDINATES                              O1234 N56789
(G54)
      NO.        DATA            NO.        DATA
      00   X     0.000           02   X     0.000
     (EXT) Y     0.000          (G55) Y     0.000
           Z     0.000                Z     0.000

      01   X     0.000           03   X     0.000
     (G54) Y     0.000          (G56) Y     0.000
           Z     0.000                Z     0.000

> Z100.                                   S    0   T0000
MDI **** *** ***          16:05:59
[ NO.SRH ] [ MEASUR ] [        ] [ +INPUT ] [ INPUT ]
```

6 Position the cursor to the workpiece origin offset value to be set.

7 Press the address key for the axis along which the offset is to be set (Y–axis in this example).

8 Enter the measured value (α) then press the **[MEASUR]** soft key.

9 Move the reference tool manually until it touches surface B of the workpiece.

10 Retract the tool without changing the X coordinate.

11 Measure distance β then enter the distance at X on the screen in the same way as in steps 7 and 8.

Limitations

• Consecutive input

Offsets for two or more axes cannot be input at the same time.

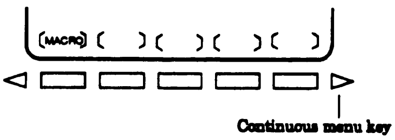
• During program execution

This function cannot be used while a program is being executed.

11.4.6 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #149 and #500 to #531) on the CRT. When the absolute value for a common variable exceeds 99999999, ******** is displayed. The values for variables can be set on this screen. Relative coordinates can also be set to valiables.

Procedure for displaying and setting custom macro common variables



1 Press function key OFFSET SETTING.

2 Press the continuous menu key ▷, then press chapter selection soft key **[MACRO]**.The following screen is displayed:

```
VARIABLE                                 O0001 N00000
  NO.       DATA           NO.        DATA
  100   1000.000          108         0.000
  101      0.000          109     40000.000
  102 -50000.000          110     153020.00
  103      0.000          111      0001.000
  104  1238501.0          112         0.000
  105      0.000          113     20000.000
  106      0.000          114         0.000
  107      0.000          115         0.000
 ACTUAL POSITION (RELATIVE)
    X      0.000            Y         0.000
    Z      0.000

> _                                  S    0  T0000
MDI **** *** ***           16:05:59
[ NO.SRH ] [      ] [ INP.C. ] [      ] [ INPUT ]
```

3 Move the cursor to the variable number to set using either of the following methods:
– Enter the variable number and press soft key **[NO.SRH]**.
– Move the cursor to the variable number to set by pressing page keys ↑ PAGE and/or PAGE ↓ and cursor keys ↑, ↓, ←, and/or →.

4 Enter data with numeric keys and press soft key **[INPUT]**.

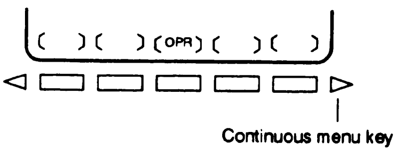
5 To set a blank in a variable, just press soft key **[INPUT]**. The value field for the variable becomes blank.

11.4.7 Displaying and Setting the Software Operator's Panel

With this function, functions of the switches on the machine operator's panel can be controlled from the CRT/MDI panel.
Jog feed can be performed using numeric keys.

Procedure for displaying and setting the software operator's panel

1 Press function key [OFFSET SETTING].



2 Press the continuous menu key [▷], then press chapter selection soft key **[OPR]**.

3 The screen consists of several pages.
Press page key [↑ PAGE] or [PAGE ↓] until the desired screen is displayed.

```
OPERATOR'S PANEL                          O0000 N00000

 MODE : MDI  AUTO   EDIT   HNDL   JOG   ZRN

  STEP MULTI.   :  *1    *10   *100
  RAPID OVRD.   :  100%   50%    25%    F0
  JOG FEED      :    2.0%
                              **************
  FEED OVRD.    :   100%
                                    ***
 ACTUAL POSITION (ABSOLUTE)
     X       0.000          Y      0.000
     Z       0.000

>_
MDI **** *** ***           16:05:59
[ MACRO ] [       ] [  OPR  ] [TOOLLF] [ (OPRT) ]
```

```
OPERATOR'S PANEL                          O0000 N00000

BLOCK SKIP     :        OFF       ON
SINGLE BLOCK   :        OFF       ON
MACHINE LOCK   :        OFF       ON
PROTECT KEY    :        PROTECT     RELEASE
FEED HOLD      :        OFF

ACTUAL POSITION (ABSOLUTE)
     X      0.000           Y      0.000
     Z      0.000

                                         S    0   T0000
MDI **** *** ***           16:05:59
[ MACRO ] [       ] [  OPR  ] [TOOLLF] [ (OPRT) ]
```

4 Move the cursor to the desired switch by pressing cursor key [↑] or [↓].

5 Push the cursor move key [←] or [→] to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform jog feed. Press the [5] key together with an arrow key to perform jog rapid traverse.

[8↑] [9↗]
[4←] [5] [6→]
[1↙] [2↓]

Explanations

• Valid operations

The valid operations on the software operator's panel are shown below. Whether to use the CRT or machine operator's panel for each group of operations can be selected by parameter 7200.

Group1 : Mode selection
Group2 : Selection of jog feed axis, jog rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification x1, x10, x100
Group4 : Jog federate, federate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

• Display

The groups for which the machine operator's panel is selected by parameter 7200 are not displayed on the software operator's panel.

• Screens on which jog feed is valid

When the CRT indicates other than the software operator's panel screen and diagnostic screen, jog feed is not conducted even if the arrow key is pushed.

• Jog feed and arrow keys

The feed axis and direction corresponding to the arrow keys can be set with parameters (Nos. 7210 to 7217).

• General purpose switches

Eight optionally definable switches are added as an extended function of the software operator's panel. The name of these switches can be set by parameters (Nos. 7220 to 7283) as character strings of max. 8 characters. For the meanings of these switches, refer to the manual issued by machine tool builder.

11.5 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM

When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.
This chapter describes how to set parameters on the MDI panel. Parameters can also be set with external input/output devices such as the Handy File (see Chapter 9).
In addition, pitch error compensation data used for improving the precision in positioning with the ball screw on the machine can be set or displayed by the operations under function key SYSTEM .
See Chapter 7 for the diagnostic screens displayed by pressing function key SYSTEM .

11.5.1 Displaying and Setting Parameters

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.
Normally, the user need not change parameter setting.

Procedure for displaying and setting parameters

1 Set 1 for **PARAMETER WRITE** to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key SYSTEM .

3 Press chapter selection soft key **[PARAM]** to display the parameter screen.

```
PARAMETER (SETTING)                          O0010 N00002

 0000         SEQ                     INI  ISO  TVC
          0    0    0    0    0    0    0    0
 0001
          0    0    0    0    0    0    0    0
 0012                                           MIR
  X       0    0    0    0    0    0    0    0
  Y       0    0    0    0    0    0    0    0
  Z       0    0    0    0    0    0    0    0
 0020 I/O CHANNEL                               0
                                                0

 > _
 THND **** *** ***        16:05:59
 [ PARAM ][ DGNOS ][  PMC  ][ SYSTEM ][ (OPRT) ]
```

4 Move the cursor to the parameter number to be set or displayed in either of the following ways:

- Enter the parameter number and press soft key **[NO.SRH]** .
- Move the cursor to the parameter number using the page keys, PAGE ↑ and PAGE ↓ , and cursor keys, ↑ , ↓ , ← , and → .

5 To set the parameter, enter a new value with numeric keys and press soft key **[INPUT]**. The parameter is set to the entered value and the value is displayed.

6 Set 0 for **PARAMETER WRITE** to disable writing.

Procedure for enabling/displaying parameter writing

1 Select the **MDI** mode or enter state emergency stop.

2 Press function key OFFSET SETTING .

3 Press soft key **[SETING]** to display the setting screen.

```
SETTING (HANDY)                            O0001 N00000

  PARAMETER WRITE  =1 (0:DISABLE 1:ENABLE)
  TV CHECK         = 0 (0:OFF    1:ON)
  PUNCH CODE       = 1 (0:EIA    1:ISO)
  INPUT UNIT       = 0 (0:MM     1:INCH)
  I/O CHANNEL      = 0 (0–2:CHANNEL NO.)
  SEQUENCE NO.     = 0 (0:OFF    1:ON)




> _                                        S    0  T0000
MDI **** *** ***         16:05:59
[ OFFSET ][ SETING ][ WORK ][       ][ (OPRT) ]
```

4 Move the cursor to **PARAMETER WRITE** using cursor keys.

5 Press soft key **[(OPRT)]**, then press **[1: ON]** to enable parameter writing.
At this time, the CNC enters the P/S alarm state (No. 100).

6 After setting parameters, return to the setting screen. Move the cursor to **PARAMETER WRITE** and press soft key **[(OPRT)]** , then press **[0: OFF]**.

7 Depress the RESET key to release the alarm condition. If alarm No. 000 has occurred, however, turn off the power supply and then turn it on, otherwise the alarm is not released.

Explanations

• Setting parameters with external input/output devices

See Chapter 8 for setting parameters with external input/output devices such as the Handy File.

• Parameters that require turning off the power

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm 000. In this case, turn off the power, then turn it on again.

• Parameter list

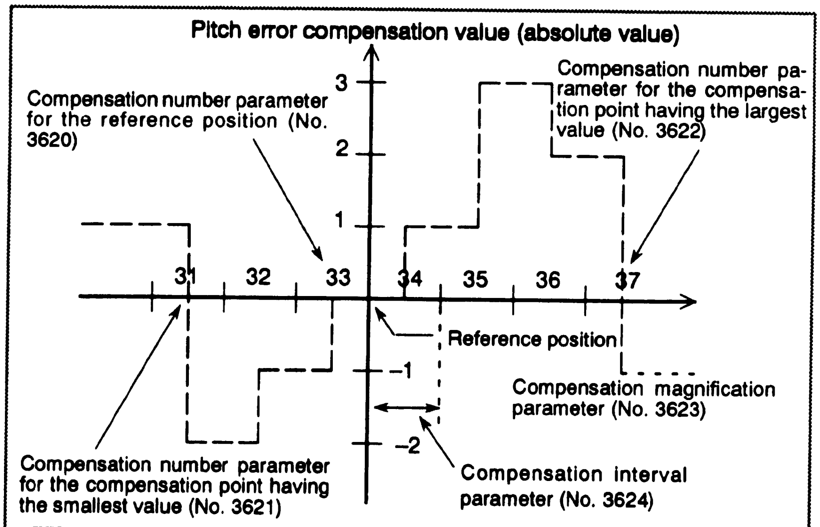
Refer to the FANUC Series 20–FA Parameter Manual (B–62180E) for the parameter list.

• Setting data

Some parameters can be set on the setting screen if the parameter list indicates "Setting entry is acceptable". Setting 1 for **PARAMETER WRITE** is not necessary when three parameters are set on the setting screen.

11.5.2 Displaying and Setting Pitch Error Compensation Data

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.
Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.
The pitch error compensation data is set according to the characteristics of the machine connected to the NC. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced.
In principle, the end user must not alter this data.
Pitch error compensation data can be set with external devices such as the Handy File (see Chapter 8). Compensation data can also be written directly with the MDI panel.
The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation point number set by these parameters.
In the following example, 33 is set for the pitch error compensation point at the reference position.



Compensation position number	31	32	33	34	35	36	37
Compensation value to be set	+3	−1	−1	+1	+2	−1	−3

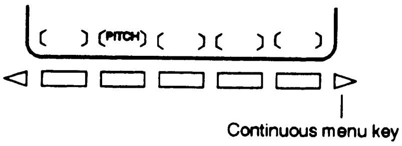
- Number of the pitch error compensation point at the reference position (for each axis) : Parameter 3620
- Number of the pitch error compensation point having the smallest value (for each axis) : Parameter 3621
- Number of the pitch error compensation point having the largest value (for each axis) : Parameter 3622
- Pitch error compensation magnification (for each axis) : Parameter 3623
- Interval of the pitch error compensation points (for each axis) : Parameter 3624

Procedure for displaying and setting the pitch error compensation data

1 Set the following parameters:

- Number of the pitch error compensation point at the reference position (for each axis): Parameter 3620
- Number of the pitch error compensation point having the smallest value (for each axis): Parameter 3621
- Number of the pitch error compensation point having the largest value (for each axis): Parameter 3622
- Pitch error compensation magnification (for each axis): Parameter 3623
- Interval of the pitch error compensation points (for each axis): Parameter 3624

2 Press function key SYSTEM.



3 Press the continuous menu key ▷, then press chapter selection soft key **[PITCH]**.
The following screen is displayed:

```
PIT–ERROR SETTING                              O0000 N00000

      NO.     DATA     NO.     DATA      NO.     DATA
      0000    0        0010    0         0020    0
      0001    0        0011    0         0021    0
      0002    0        0012    0         0022    0
      0003    0        0013    0         0023    0
   (X)0004    0        0014    0         0024    0
      0005    0        0015    0         0025    0
      0006    0        0016    0         0026    0
      0007    0        0017    0         0027    0
      0008    0        0018    0         0028    0
      0009    0        0019    0         0029    0
>_
MEM **** *** ***          16:05:59
[ NO.SRH ][ ON:1 ][ OFF:0 ][ +INPUT ][ –INPUT ]
```

4 Move the cursor to the compensation point number to be set in either of the following ways:

- Enter the compensation point number and press the [NO.SRH] soft key.
- Move the cursor to the compensation point number using the page keys, PAGE ↑ and PAGE ↓, and cursor keys, ↑, ↓, ←, and →.

5 Enter a value with numeric keys and press the [INPUT] soft key.

11.6 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION

The program number, sequence number, and current CNC status are always displayed on the screen except when the power is turned on, a system alarm occurs, or the PMC screen or CUSTOM screen is displayed. If data setting or the input/output operation is incorrect, the CNC does not accept the operation and displays a warning message.
This section describes the display of the program number, sequence number, and status, and warning messages displayed for incorrect data setting or input/output operation.

11.6.1 Displaying the Program Number and Sequence Number

The program number and sequence number are displayed at the top right on the screen as shown below.

```
PROGRAM                                  O2000 N00130
O2000 ;
N100 G92 X0 Y0 Z70. ;
N110 G91 G00 Y–70. ;
N120 Z–70. ;
N130 G42 G39 I–17.5
N140 G41 G03 X–17.5 Y17.5 R17.5 ;
N150 G01 X–25. ;
N160 G02 X27.5 Y27.5 R27.5
N170 G01 X20. ;
N180 G02 X45. Y45. R45. ;
> _
MEM **** *** ***           16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

Sequence No.
Program No.

The program number and sequence number displayed depend on the screen and are given below:

On the program screen in the EDIT mode on Background edit screen :
The program No. being edited and the sequence number just prior to the cursor are indicated.

Other than above screens :

The program No. and the sequence No. executed last are indicated.

Immediately after program number search or sequence number search :

Immediately after the program No. search and sequence No. search, the program No. and the sequence No. searched are indicated.

11.6.2 Displaying the Status and Warning for Data Setting or Input/Output Operation

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the CRT screen allowing the operator to readily understand the operation condition of the system.
If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed on the next to last line of the CRT screen. This prevents invalid data setting and input/output errors.

Explanations

Description of each display

(9) Data is out of range.
(Note) Actually, this is displayed in the area starting from (2).

(5) EMG (Note) Actually, 5 is displayed in the area for (3) and (4).

(1)	(2)	(3)	(4)	(6)	(7)	(8)
EDIT	STOP	MTN	FIN	ALM	hh:mm:ss	INPUT

(Display soft keys)

(1) Current mode

MDI	:	Manual data input
MEM	:	Automatic operation (Memory operation)
RMT	:	Automatic operation (DNC operation, or such like)
EDIT	:	Memory editing
HND	:	Manual handle feed
JOG	:	Jog feed
TJOG	:	TEACH IN JOG
THND	:	TEACH IN HANDLE
REF	:	Manual reference position return

(2) Automatic operation status

****	:	Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP	:	Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD	:	Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT	:	Automatic operation start–up (The state in which the system operates automatically)

(3) Axis moving status/dwell status

MTN	:	Indicates that the axis is moving.
DWL	:	Indicates the dwell state.
***	:	Indicates a state other than the above.

(4) State in which an auxiliary function is being executed

FIN	:	Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)
***	:	Indicates a state other than the above.

(5) Emergency stop or reset status

– EMG –	:	Indicates emergency stop.(Blinks in reversed display.)
—RESET—	:	Indicates that the reset signal is being received.

(6) Alarm status

ALM	:	Indicates that an alarm is issued. (Blinks in reversed display.)
BAT	:	Indicates that the battery is low. (Blinks in reversed display.)
Space	:	Indicates a state other than the above.

(7) Current time

hh:mm:ss – Hours, minutes, and seconds

(8) Program editing status

INPUT : Indicates that data is being input.
OUTPUT : Indicates that data is being output.
SRCH : Indicates that a search is being performed.
EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)
LSK : Indicates that labels are skipped when data is input.
RSTR : Indicates that the program is being restarted
Space : Indicates that no editing operation is being performed.

(9) Warning for data setting or input/output operation

When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. In this case, the CNC does not accept the setting or input/output operation (retry the operation according to the message). The following are examples of warning messages:

Example 1)
When a parameter is entered

```
> 1
EDIT   WRONG MODE
                  (Display sof tkeys)
```

Example 2)
When a parameter is entered

```
> 999999999
MDI   TOO MANY DIGITS
                  (Display soft keys)
```

Example 3)
When a parameter is output to an external input/output device

```
> _
MEM   WRONG MODE
                  (Display soft keys)
```

12 HELP FUNCTION

The help function displays on the screen detailed information about alarms issued in the CNC and about CNC operations. The following information is displayed.

- **Detailed information of alarms**

When the CNC is operated incorrectly or an erroneous machining program is executed, the CNC enters the alarm state. The help screen displays detailed information about the alarm that has been issued and how to reset it. The detailed information is displayed only for a limited number of P/S alarms. These alarms are often misunderstood and are rather difficult to understand.

- **Operation method**

If you are not sure about a CNC operation, refer to the help screen for information about each operation.

- **Parameter table**

When setting or referring to a system parameter, if you are not sure of the number of the parameter, the help screen displays a list of parameter Nos. for each function.

Help Function Procedure

Procedure

1 Press the [HELP] key on the MDI panel. HELP (INITIAL MENU) screen is displayed.

```
HELP (INITIAL MENU)                         O1234  N00001

                  ***** HELP *****
                   1. ALARM DETAIL
                   2. OPERATION METHOD
                   3. PARAMETER TABLE

                                             S    0  T0000
  MEM  ****   ***   ***               10 : 10 : 40
 [ 1 ALAM ][ 2 OPR ][ 3 PARA ][       ][       ]
```

Fig.12 (a) HELP (INITIAL MENU) Screen

The user cannot switch the screen display from the PMC screen or CUSTOM screen to the help screen. The user can return to the normal CNC screen by pressing the [HELP] key or another function key.

ALARM DETAIL screen

2 Press soft key **[1 ALAM]** on the HELP (INITIAL MENU) screen to display detailed information about an alarm currently being raised.

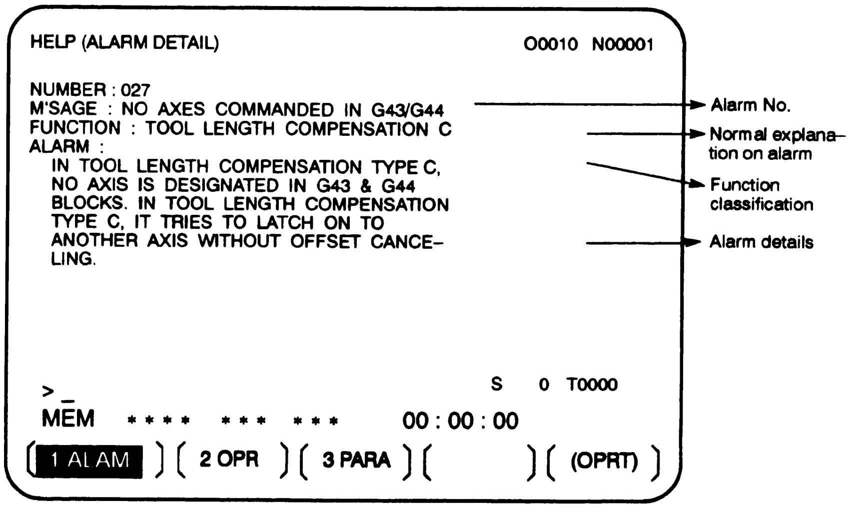


Fig.12 (b) ALARM DETAIL Screen when Alarm (NO. 027) is Issued

Note that only details of the alarm identified at the top of the screen are displayed on the screen.
If the alarms are all reset while the help screen is displayed, the alarm displayed on the ALARM DETAIL screen is deleted, indicating that no alarm is issued.

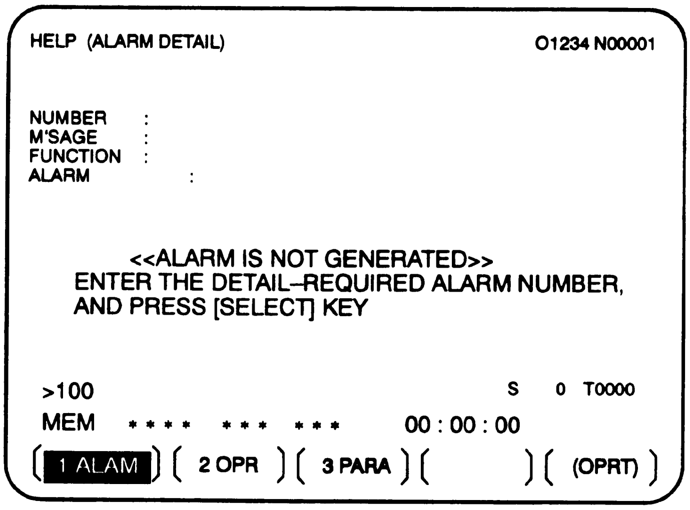


Fig.12 (c) ALARM DETAIL Screen when No Alarm is Issued

3 To get details on another alarm number, first enter the alarm number, then press soft key **[SELECT]**. This operation is useful for investigating alarms not currently being raised.

```
HELP (ALARM DETAIL)                                   O1234 N00001

NUMBER       :  100
M'SAGE       :  PARAMETER WRITE ENABLE
FUNCTION     :
ALARM        :

        <<ALARM IS NOT GENERATED>>
        ENTER THE DETAIL–REQUIRED ALARM NUMBER, AND
        PRESS [SELECT] KEY.

                                                   S    0 T0000
MEM  ****  ***  ***          00 : 00 : 00
(        )(        )(        )(        )( SELECT )
```

Fig.12 (d) ALARM DETAIL Screen when Alarm (NO. 100) is selected

OPERATION METHOD screen

4 To determine an operating procedure for the CNC, press the soft key **[2 OPR]** key on the HELP (INITIAL MENU) screen. The OPERATION METHOD menu screen is then displayed. (See Fig. 12 (e).)

```
HELP (OPERATION METHOD)                     O1234   N00001

 1. PROGRAM EDIT
 2. SEARCH
 3. RESET
 4. DATA INPUT WITH MDI
 5. DATA INPUT WITH TAPE
 6. OUTPUT
 7. INPUT WITH FANUC CASSETTE
 8. OUTPUT WITH FANUC CASSETTE
 9. MEMORY CLEAR

                                             S    0 T0000
MEM  ****  ***  ***          00 : 00 : 00
( 1 ALAM )( 2 OPR )( 3 PARA )(        )( (OPRT) )
```

Fig.12 (e) OPERATION METHOD Menu Screen

To select an operating procedure, enter an item No. from the keyboard then press the **[SELECT]** key.

```
>1                                       S    0   T0000
MEM  ****  ***  ***          00 : 00 : 00
(        )(        )(        )(        )(SELECT)
```

Fig.12 (f) How to select each OPERATION METHOD screen

When “1. PROGRAM EDIT” is selected, for example, the screen in Figure 12 (g) is displayed.

On each OPERATION METHOD screen, it is possible to change the displayed page by pressing the PAGE key. The current page No. is shown at the upper right corner on the screen.

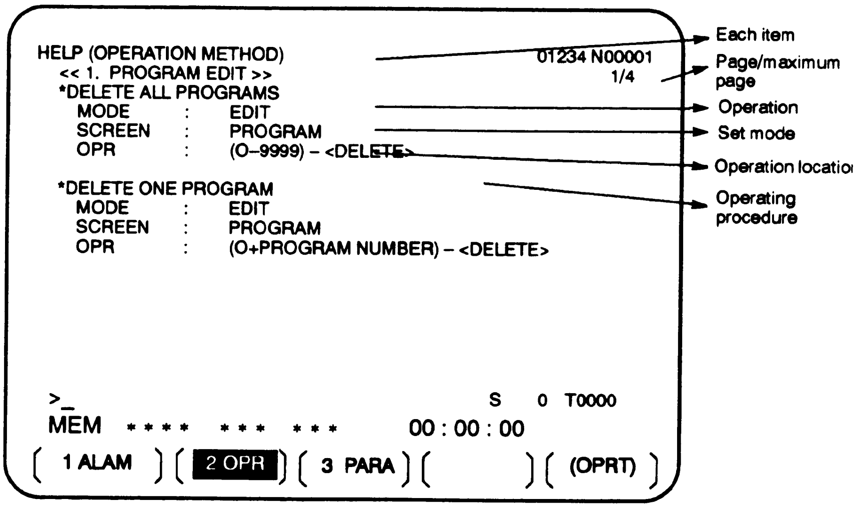
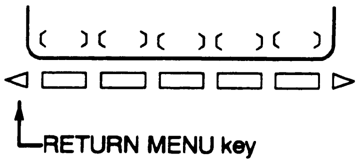


Fig.12 (g) Selected OPERATION METHOD screen



5 To return to the OPERATION METHOD menu screen, press the RETURN MENU key to display “**[2 OPR]**” again, and then press the **[2 OPR]** key again.

To directly select another OPERATION METHOD screen on the screen shown in Figure 12 (g), enter an item No. from the keyboard and press the **[SELECT]** key.

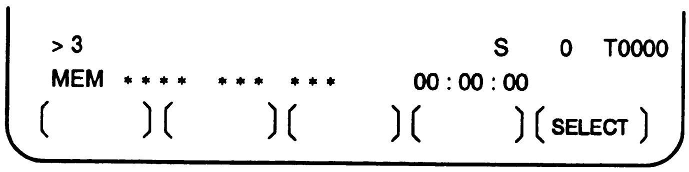


Fig.12 (h) How to select another OPERATION METHOD screen

PARAMETER TABLE screen

6 If you are not sure of the No. of a system parameter to be set, or to refer to a system parameter, press the **[3 PARA]** key on the HELP (INITIAL MENU) screen. A list of parameter Nos. for each function is displayed. (See Figure 12 (i).)

It is possible to change the displayed page on the parameter screen. The current page No. is shown at the upper right corner on the screen.

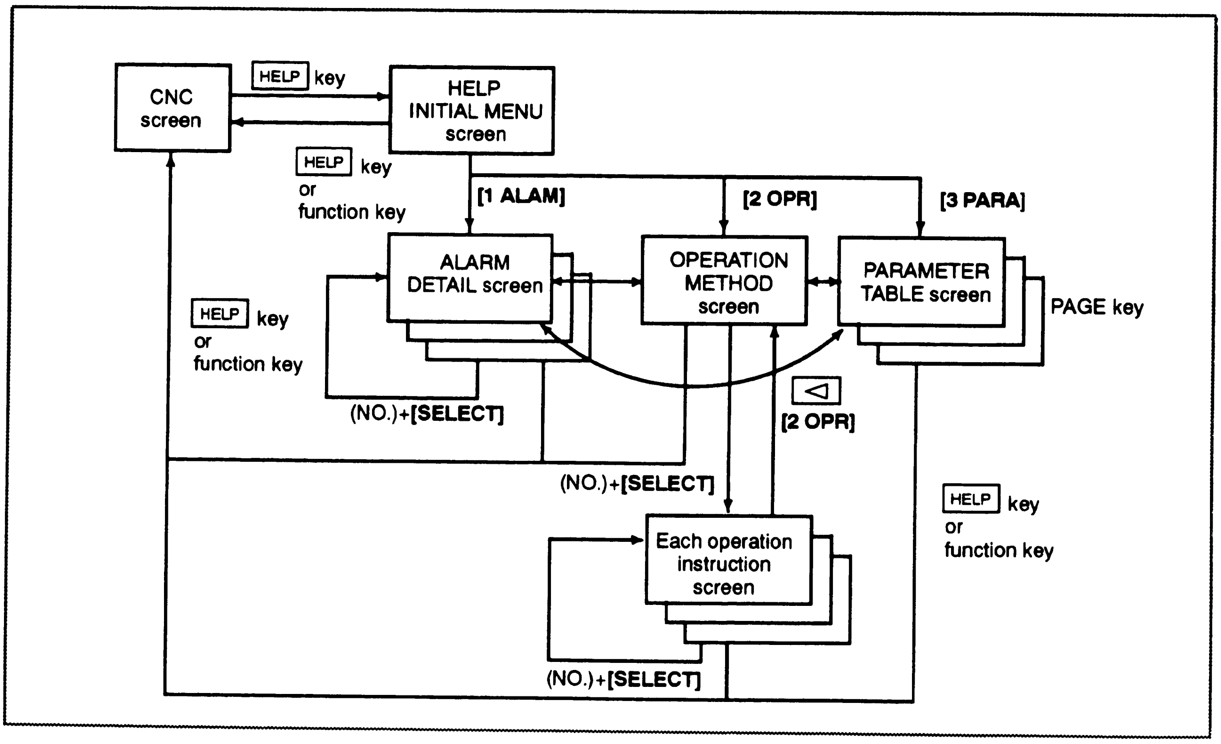
```
HELP (PARAMETER TABLE)                          01234  N00001
                                                          1/4

  • SETTEING         (No. 0000A)
  • READER/PUNCHER INTERFACE                 (No. 0100A)
  • AXIS CONTROL/SETTING UNIT                (No. 1000A)
  • COORDINATE SYSTEM                        (No. 1200A)
  • STROKE LIMIT  (No. 1300A)
  • FEED RATE        (No. 1400A)
  • ACCEL/DECELERATION CTRL                  (No. 1600A)
  • SERVORELATED                             (No. 1800A)
  • DI/DO                                    (No. 3000A)

                                              S     0   T0000
 MEM  ****  ***  ***                   00 : 00 : 00
[ 1 ALAM ][ 2 OPR ][ 3 PARA ][        ][ (OPRT) ]
```

Fig. 12 (i) PARAMETER TABLE screen

7 To exit from the help screen, press the HELP key or another function key.

Explanation

• Configuration of the Help Screen



IV MAINTENANCE

1 METHOD OF REPLACING BATTERY

This chapter describes the method of replacing batteries as follows.

1.1 REPLACING CNC BATTERY FOR MEMORY BACK–UP
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER
1.3 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER (Servo amplifier converter unit only for Series 20 or α series Servo Amplifier Module)

- **Battery for Memory Backup (DC3V)**

Part programs, offset data, and system parameters are stored in CMOS memory in the control unit. The power to the CMOS memory is backed up by a lithium battery mounted on the front panel of the control unit. The above data is not lost even when the main battery goes dead. The backup battery is mounted on the control unit at shipping. This battery can maintain the contents of memory for about a year.

When the voltage of the battery becomes low, alarm message "BAL" blinks on the CRT display and the battery alarm signal is output to the PMC. When this alarm is displayed, replace the battery as soon as possible. In general, the battery can be replaced within two or three weeks, however, this depends on the system configuration.

If the voltage of the battery becomes any lower, memory can no longer be backed up and its contents are lost. Turning on the power to the control unit in this state causes system alarm 910 (SRAM parity alarm) to occur because the contents of memory are lost. Clear the entire memory and reenter data after replacing the battery.

- **Battery for Absolute pulse coder (DC6V)**

When the machine is equipped with absolute encoder system such as an absolute pulse coder or absolute linear scale, there is a battery for them separately from the battery for memory backup.
One battery unit can maintain absolute position data for six absolute pulse coders for a year.

When the voltage of the battery becomes low, APC alarms 3n6 to 3n8 (n: axis number) are displayed on the CRT display. When APC alarm 3n7 or 3n8 is displayed, replace the battery as soon as possible. In general, the battery should be replaced within two or three weeks, however, this depends on the number of pulse coders used.
If the voltage of the battery becomes any lower, the absolute positions for the pulse coders can no longer be maintained. Turning on the power to the control unit in this state causes APC alarm 3n0 (reference position return request alarm) to occur. Return the tool to the reference position after replacing the battery.

1.1 REPLACING CNC BATTERY FOR MEMORY BACK–UP

Replace CNC battery (lithium battery) for memory back–up by the following procedure.
Prepare lithium battery (A02B–0177–K106) in advance.
Observe the following precautions for lithium batteries:

Notes
If an unspecified battery is used, it may explode.
Replace the battery only with the specified battery (A02B–0177–K106).

Dispose of used lithium batteries as follows:
(1) Small quantities
Discharge the batteries and dispose of them as ordinary nonflammable garbage.
(2) Large quantities
Consult FANUC.

Procedure for replacing battery for memory back–up

1 Turn on machine (CNC) power.

Note
Replace the battery under the emergency stop state for safety, to escape the machine from moving during the replacement work.

2 Open the cover on the front panel of the power supply unit.

3 Remove the connector of CP8.

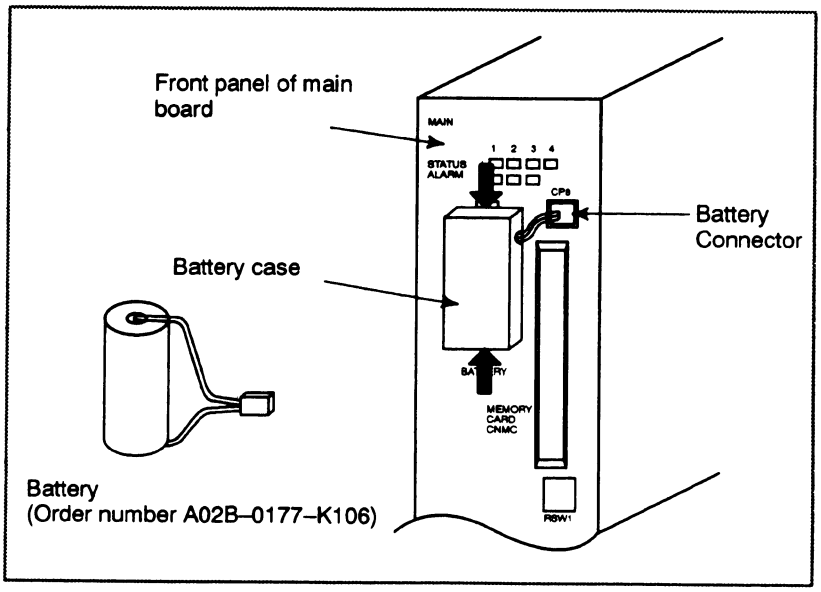


Fig. 1.1 (a) Replacing the battery (1)

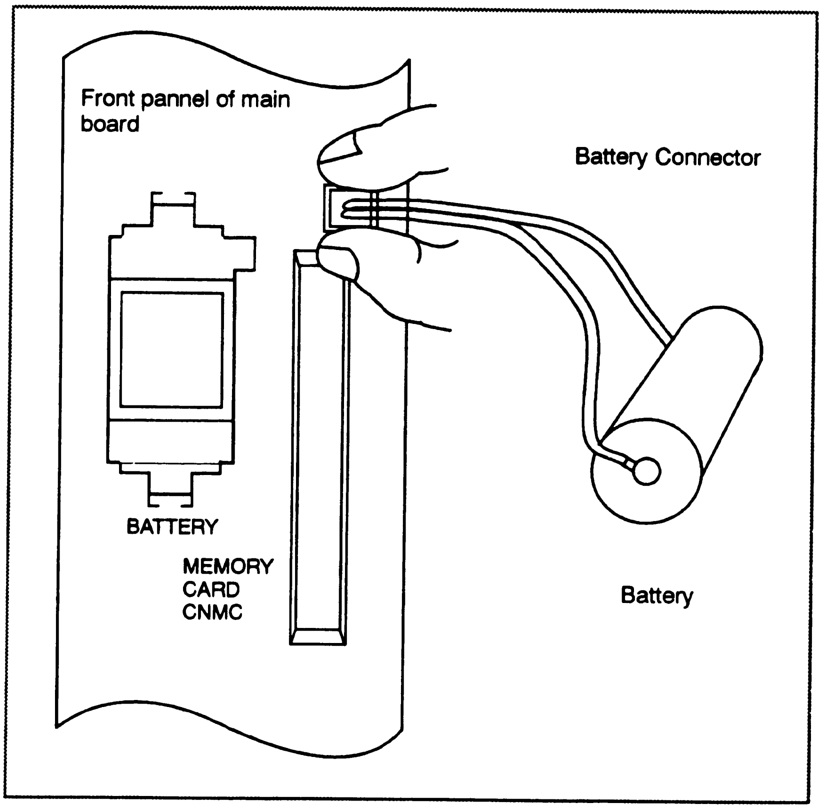


Fig. 1.1 (b) Replacing the battery (2)

4 Replace the battery and reconnect the connector.

5 Close the cover on the front panel of the power supply unit.

6 Turn off machine (CNC) power.

Notes

The batteries can be replaced whether NC power is ON or OFF. However, the replacement should be done within 30 minutes when the power is OFF. Memory contents might be erased if the NC power is OFF without battery for more than 30 minutes.

When the contents of memory are lost, turning on the power to the CNC may cause a RAM PARITY system alarm to occur. In this case, there is need to clear all memory contents and the power ON.

1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER

Replace batteries (alkaline batteries) for absolute pulse coder by the following procedure.
Prepare 4 alkaline batteries (UM–1type) commercially available in advance.
If use the α series servo amplifier module, see section 1.3.

Procedure for replacing batteries for absolute pulse coder

Procedure

1 Turn machine (NC) power ON.

> **Note**
> Replace the battery under the emergency stop state for safety, to escape the machine from moving during the replacement work.

2 Loosen screws on the battery case to remove the cover. For placement of the battery case, refer to the machine tool builder's manual.

3 Replace the batteries in the case. Insert 2 batteries each in the opposite direction as illustrated below. Be sure to take notice of polarity of the battery.

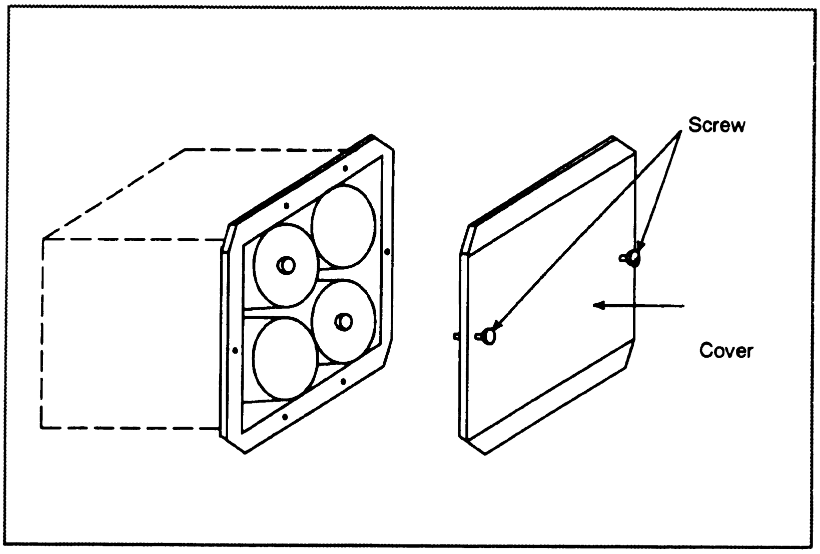


Fig. 1.2

4 After replacement, install the cover.

5 Turn off machine (NC) power OFF.

> **Notes**
> Replace the batteries for absolute pulse coder when NC power is ON.
> Replacing the batteries with power OFF causes the absolute position stored in memory to be lost.

1.3 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER (SERVO AMPLIFIER CONVERTER UNIT ONLY FOR SERIES 20 OR α SERIES SERVO AMPLIFIER MODULE)

In case that the α series servo drive is used, the battery for absolute pulse coder could be provided on the servo amplifier converter unit only for Series 20 or α series servo amplifier module instead of the battery case as shown in 1.3. In this case the battery is not an alkaline battery but a lithium battery, A06B–6073–K001. Prepare the battery in advance and replace it by the following procedure.
Observe the following precautions for lithium batteries:

> **Notes**
> If an unspecified battery is used, it may explode.
> Replace the battery only with the specified battery (A06B–6073–K001).

Dispose of used lithium batteries as follows:

(1) Small quantities
Discharge the batteries and dispose of them as ordinary nonflammable garbage.

(2) Large quantities
Consult FANUC.

Procedure for replacing batteries for absolute pulse coder

Procedure

1 Turn on machine (CNC) power.

> **Note**
> Replace the battery under the emergency stop state for safety, to escape the machine from moving during the replacement work.

2 Remove the battery case on the front panel of servo amplifier converter unit only for Series 20 or α series servo amplifier module (SVM).
The battery case can be removed by holding the top of the case and pulling the case towards you.

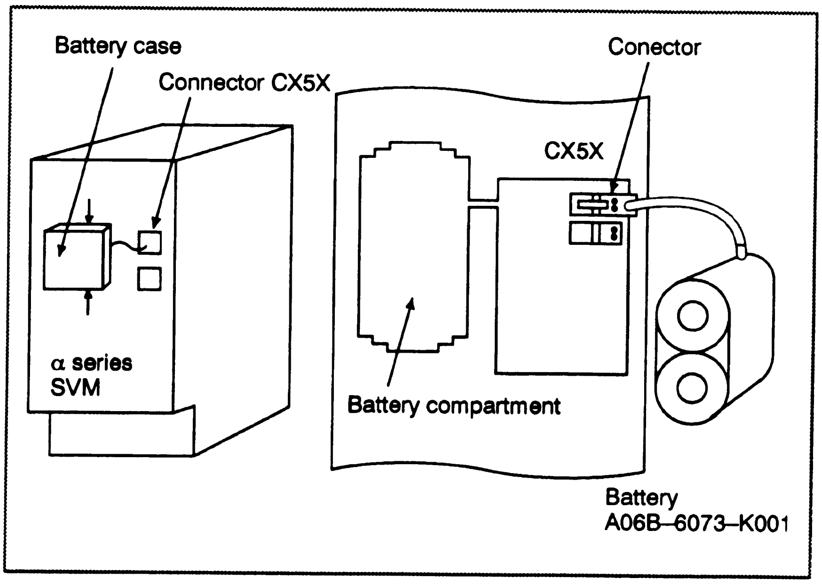


Fig. 1.3

3 Remove the connector of CX5X.

4 Replace the battery, and connect the connector.

5 Attach the battery case.

6 Turn off machine (CNC) power.

Notes

1. Replace the batteries for absolute pulse coder when NC power is ON.
   Replacing the batteries with power OFF causes the absolute position stored in memory to be lost.
2. If your machine is equipped with a separate battery case follow the instructions in 1.2.

APPENDIX

A TAPE CODE LIST

ISO code										EIA code										Meaning		
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Without CUSTOM MACURO B	Without CUSTOM MACRO B
0			O	O		o				0			O			o				Number 0		
1	O		O	O		o			O	1						o			O	Number 1		
2	O		O	O		o		O		2						o		O		Number 2		
3			O	O		o		O	O	3				O		o		O	O	Number 3		
4	O		O	O		o	O			4						o	O			Number 4		
5			O	O		o	O		O	5				O		o	O		O	Number 5		
6			O	O		o	O	O		6				O		o	O	O		Number 6		
7	O		O	O		o	O	O	O	7						o	O	O	O	Number 7		
8	O		O	O	O	o				8					O	o				Number 8		
9			O	O	O	o			O	9				O	O	o			O	Number 9		
A		O				o			O	a		O	O			o			O	Address A		
B		O				o		O		b		O	O			o		O		Address B		
C	O	O				o		O	O	c		O	O	O		o		O	O	Address C		
D		O				o	O			d		O	O			o	O			Address D		
E	O	O				o	O		O	e		O	O	O		o	O		O	Address E		
F	O	O				o	O	O		f		O	O	O		o	O	O		Address F		
G		O				o	O	O	O	g		O	O			o	O	O	O	Address G		
H		O			O	o				h		O	O		O	o				Address H		
I	O	O			O	o			O	i		O	O	O	O	o			O	Address I		
J	O	O			O	o		O		j		O		O		o		O	O	Address J		
K		O			O	o		O	O	k		O		O		o		O		Address K		
L	O	O			O	o	O			l		O				o		O	O	Address L		
M		O			O	o	O		O	m		O		O		o	O			Address M		
N		O			O	o	O	O		n		O				o	O		O	Address N		
O	O	O			O	o	O	O	O	o		O				o	O	O		Address O		
P		O		O		o				p		O		O		o	O	O	O	Address P		
Q	O	O		O		o			O	q		O		O	O	o				Address Q		
R	O	O		O		o		O		r		O			O	o			O	Address R		
S		O		O		o		O	O	s			O	O		o		O		Address S		
T	O	O		O		o	O			t			O			o		O	O	Address T		
U		O		O		o	O		O	u			O	O		o	O			Address U		
V		O		O		o	O	O		v			O			o	O		O	Address V		
W	O	O		O		o	O	O	O	w			O			o	O	O		Address W		
X	O	O		O	O	o				x			O	O		o	O	O	O	Address X		
Y		O		O	O	o			O	y			O	O	O	o				Address Y		
Z		O		O	O	o		O		z			O		O	o			O	Address Z		

ISO code										EIA code										Meaning		
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Without custom macro B	Without custom macro B
DEL	○	○	○	○	○	o	○	○	○	Del		○	○	○	○	o	○	○	○		×	×
NUL						o				Blank						o					×	×
BS	○				○	o				BS			○		○	o		○			×	×
HT					○	o			○	Tab			○	○	○	o	○	○			×	×
LF or NL					○	o		○		CR or EOB	○					o						
CR	○				○	o	○		○	___											×	×
SP	○		○			o				SP				○		o					□	□
%	○		○			o	○		○	ER					○	o		○	○			
(			○		○	o				(2–4–5)				○	○	o		○				
)	○		○		○	o			○	(2–4–7)		○			○	o		○				
+			○		○	o		○	○	+		○	○	○		o					Δ	
–			○		○	o	○		○	–		○				o						
:			○	○	○	o		○		___												
/	○		○		○	o	○	○	○	/			○	○		o			○			
.			○		○	o	○	○		.		○	○		○	o		○	○			
#	○		○			o		○	○	Parameter (No. 6012)												
$			○			o	○			___											Δ	○
&	○		○			o	○	○		&					○	o	○	○			Δ	○
∇			○			o	○	○	○	___											Δ	○
*	○		○		○	o		○		Parameter (No. 6010)											Δ	
,	○		○		○	o	○			,			○	○	○	o		○	○			
;	○		○	○	○	o		○	○	___											Δ	Δ
<			○	○	○	o	○			___											Δ	Δ
=	○		○	○	○	o	○		○	Parameter (No. 6011)											Δ	
>	○		○	○	○	o	○	○		___											Δ	Δ
?			○	○	○	o	○	○	○	___											Δ	○
@	○	○				o				___											Δ	○
"			○					○		___											Δ	Δ
[	○	○		○	○	o		○	○	Parameter (No. 6013)											Δ	
]	○	○		○	○	o	○		○	Parameter (No. 6014)											Δ	

Notes

1. The symbols used in the remark column have the following meanings.

   (Space): The character will be registered in memory and has a specific meaning.
   It it is used incorrectly in a statement other than a comment, an alarm occurs.

   × : The character will not be registered in memory and will be ignored.

   Δ : The character will be registered in memory, but will be ignored during program execution.

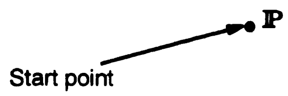
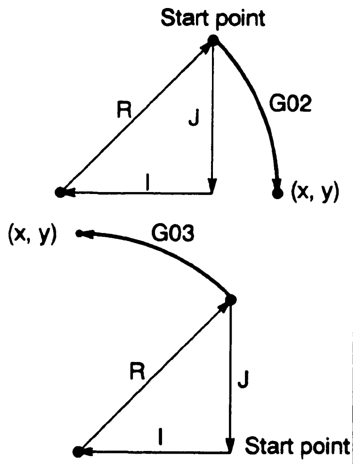
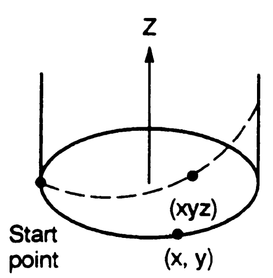
   ○ : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.

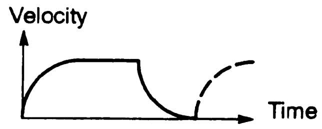
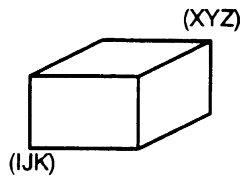
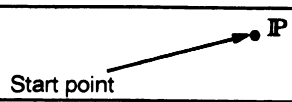
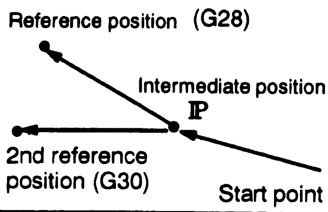
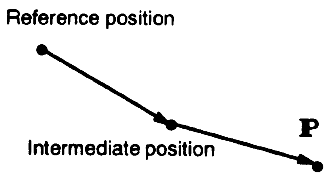
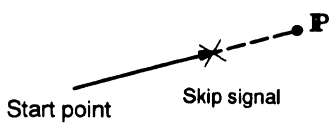
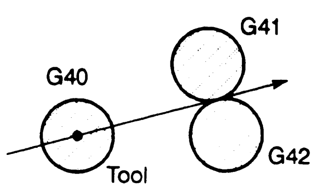
   □ : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.

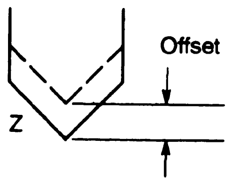
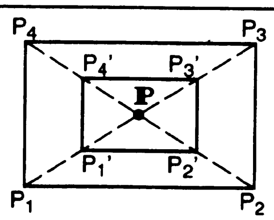
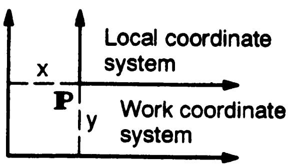
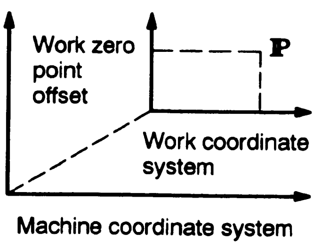
2. Codes not in this table are ignored if their parity is correct.
3. Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.
4. A character with all eight holes punched is ignored and does not generate TH alarm in EIA code.

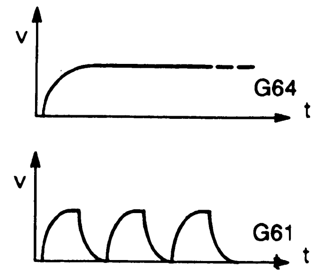
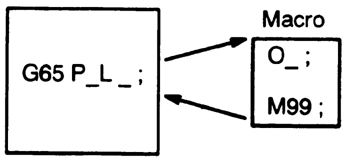
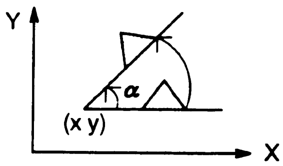
B LIST OF FUNCTIONS AND TAPE FORMAT

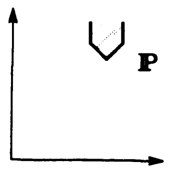
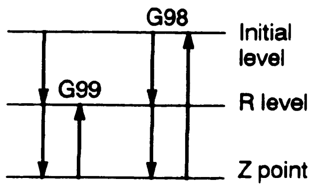
In the tables below, **IP_** :presents a combination of arbitrary axis addresses using X,Y,Z,A,B and C (such as X_Y_Z_A_).
x = 1st basic axis (X usually)
y = 2nd basic axis (Y usually)
z = 3rd basic axis (Z usually)

Functions	Illustration	Tape format
Positioning (G00)		G00 **IP**_ ;
Linear interpolation (G01)		G01 **IP**_ F_;
Circular interpolation (G02, G03)		G17 { G02 / G03 } X_ Y_ { R_ / I_ J_ } F_ ; G18 { G02 / G03 } X_ Z_ { R_ / I_ K_ } F_ ; G19 { G02 / G03 } Y_ Z_ { R_ / J_ K_ } F_ ;
Helical interpolation (G02, G03)	 (In case of X–Y plane)	G17 { G02 / G03 } X_ Y_ { R_ / I_ J_ } α_ F_ ; G18 { G02 / G03 } X_ Z_ { R_ / I_ K_ } α_ F_ ; G19 { G02 / G03 } Y_ Z_ { R_ / J_ K_ } α_ F_ ; α : Any axis other than circular interpolation axes.
Dwell (G04) (In case of X–Y plane)		G04 { X_ / P_ } ;

Functions	Illustration	Tape format
Exact stop (G09)		G09 { G01 / G02 / G03 } IP_;
Change of offset value by program (G10)		G10 L11 P_R_; G10 L1 P_R_;
Plane section (G17, G18, G19)		G17 ; G18 ; G19 ;
Inch/millimeter conversion (G20, G21)		G20 ; Inch input G21 ; Millimeter input
Stored stroke check (G22, G23)		G22 X_Y_Z_I_J_K_; G23 Cancel;
Reference position return check (G27)		G27 IP_ ;
Reference position return (G28) 2nd, reference position return (G30)		G28 IP_ ; G30 IP_ ;
Return from reference position to start point (G29)		G29 IP_ ;
Skip function (G31)		G31 IP_F_;
Cutter compensation C (G40 – G42)		{ G17 / G18 / G19 } { G41 / G42 } D_ ; D : Tool offset G40 : Cancel

Functions	Illustration	Tape format
Tool length offset A (G43, G44, G49)		$\left\{\begin{matrix}\text{G43}\\ \text{G44}\end{matrix}\right\}$ Z_ H_ ; $\left\{\begin{matrix}\text{G43}\\ \text{G44}\end{matrix}\right\}$ H_ ; H : Tool offset G49 : Cancel
Tool length offset B (G43, G44, G49)		$\left\{\begin{matrix}\text{G17}\\ \text{G18}\\ \text{G19}\end{matrix}\right\}\left\{\begin{matrix}\text{G43}\\ \text{G44}\end{matrix}\right\}\left\{\begin{matrix}\text{Z\_}\\ \text{Y\_}\\ \text{X\_}\end{matrix}\right\}$ H_ ; $\left\{\begin{matrix}\text{G17}\\ \text{G18}\\ \text{G19}\end{matrix}\right\}\left\{\begin{matrix}\text{G43}\\ \text{G44}\end{matrix}\right\}$ H_ ; H : Tool offset G49 : Cancel
Scaling (G50, G51)		G51 **P**_ P_; P : Scaling magnification G50 ; Cancel
Setting of local coordinate system (G52)		G52 **P**_ ;
Command in machine coordinate system (G53)		G53 **P**_ ;
Selection of work coordinate system (G54 – G59)		$\left\{\begin{matrix}\text{G54}\\ \vdots\\ \text{G59}\end{matrix}\right\}$ **P**_ ;
Single direction positioning (G60)		G60 **P**_ ;

Functions	Illustration	Tape format
Cutting mode/Exact stop mode, Tapping mode		G64_ ; **Cutting mode** G61_ ; **Exact stop mode** G63_ ; **Tapping mode**
Custom macro (G65, G66, G67)		One–shot call G65 P_ L_ <Argument assignment> ; P : Program No. L : Number of repeatition Modal call {G66 / G66.1} P_L_ <Argument assignment>; G67 ; Cancel
Coordinate system rotation (G68, G69)	(In case of X–Y plane)	G68 {G17 X_ Y_ / G18 Z_ X_ / G19 Y_ Z_} R α ; G69 ; Cancel
Canned cycles (G73, G74, G80 – G89)	Refer to II.14. FUNCTIONS TO SIMPLIFY PROGRAMMING	G80 ; Cancel {G73 / G74 / G76 / G81 / : / G89} X_ Y_ Z_ P_ Q_ R_ F_ K_ ;
Absolute/incremental programming (G90/G91)		G90_ ; Absolute command G91_ ; Incremental command G90_ G91_ ; Combined use

Functions	Illustration	Tape format
Change of workpiece coordinate system (G92)		G92 IP_ ;
Initial point return / R point return (G98, G99)		G98_ ; G99_ ;

C RANGE OF COMMAND VALUE

Linear axis

- **In case of millimeter input, feed screw is millimeter**

	Increment system
	IS–B
Least input increment	0.001 mm
Least command increment	0.001 mm
Max. programmable dimension	±99999.999 mm
Max. rapid traverse **Notes**	240000 mm/min
Feedrate range **Notes**	1 to 240000 mm/min
Incremental feed	0.001, 0.01, 0.1, 1 mm/step
Tool compensation	0 to ±999.999 mm
Dwell time	0 to 99999.999 sec

- **In case of inch input, feed screw is millimeter**

	Increment system
	IS–B
Least input increment	0.0001 inch
Least command increment	0.001 mm
Max. programmable dimension	±9999.9999 inch
Max. rapid traverse **Notes**	240000 mm/min
Feedrate range **Notes**	0.01 to 9600 inch/min
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step
Tool compensation	0 to ±99.9999 inch
Dwell time	0 to 99999.999 sec

- **In case of inch input, feed screw is inch**

	Increment system
	IS–B
Least input increment	0.0001 inch
Least command increment	0.0001 inch
Max. programmable dimension	±9999.9999 inch
Max. rapid traverse **Notes**	9600 inch/min
Feedrate range **Notes**	0.01 to 9600 inch/min
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step
Tool compensation	0 to ±99.9999 inch
Dwell time	0 to 99999.999 sec

- **In case of millimeter input, feed screw is inch**

	Increment system
	IS–B
Least input increment	0.001 mm
Least command increment	0.0001 inch
Max. programmable dimension	±99999.999 mm
Max. rapid traverse **Notes**	9600 inch/min
Feedrate range **Notes**	1 to 240000 mm/min
Incremental feed	0.001, 0.01, 0.1, 1 mm/step
Tool compensation	0 to ±999.999 mm
Dwell time	0 to 99999.999 sec

Notes
The feedrate range shown above are limitations depending on CNC interpolation capacity. As a whole system, limitations depending on servo system must also be considered.

D NOMOGRAPHS

D.1 INCORRECT THREADED LENGTH

The leads of a thread are generally incorrect in δ_1 and δ_2, as shown in Fig. D.1 (a), due to automatic acceleration and deceleration.
Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.

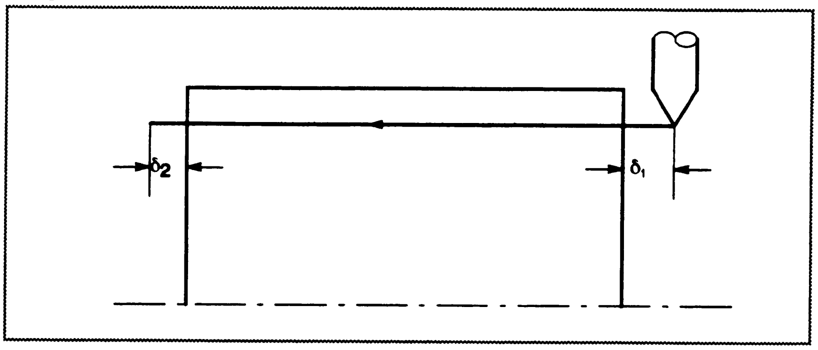


Fig.D.1(a) Incorrect thread position

Explanations

- **How to determine δ_2**

$$\delta_2 = T_1 V \text{ (mm)} \;\cdots\cdots\; (1)$$

$$V = \frac{1}{60} RL$$

T_1 : Time constant of servo system (sec)
V : Cutting speed (mm/sec)
R : Spindle speed (rpm)
L : Thread feed (mm)

Time constant T_1 (sec) of the servo system: Usually 0.033 s.

- **How to determine δ_1**

$$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\} V \;\cdots\cdots\; (2)$$

$$a = \exp(-\frac{t}{T_1}) \;\cdots\cdots\; (3)$$

T_1 : Time constant of servo system (sec)
V : Cutting speed (mm/sec)

Time constant T_1 (sec) of the servo system: Usually 0.033 s.

The lead at the beginning of thread cutting is shorter than the specified lead L, and the allowable lead error is ΔL. Then as follows.

$$a = \frac{\Delta L}{L}$$

When the value of H*a*I is determined, the time lapse until the thread accuracy is attained. The time HtI is substituted in (2) to determine δ_1: Constants V and T_1 are determined in the same way as for δ_2. Since the calculation of δ_1 is rather complex, a nomography is provided on the following pages.

• How to use nomograph

First specify the class and the lead of a thread. The thread accuracy, *a*, will be obtained at (1), and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm / s will be obtained at (2). Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/ s can be obtained at (3).

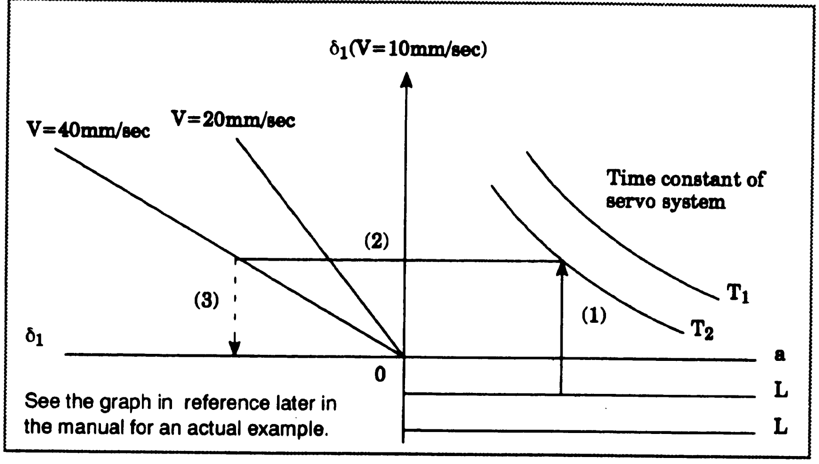


Fig.D.1(b) Nomograph

> **Note**
> The equations for δ_1, and δ_2 are for when the acceleration/ deceleration time constant for cutting feed is 0.

D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

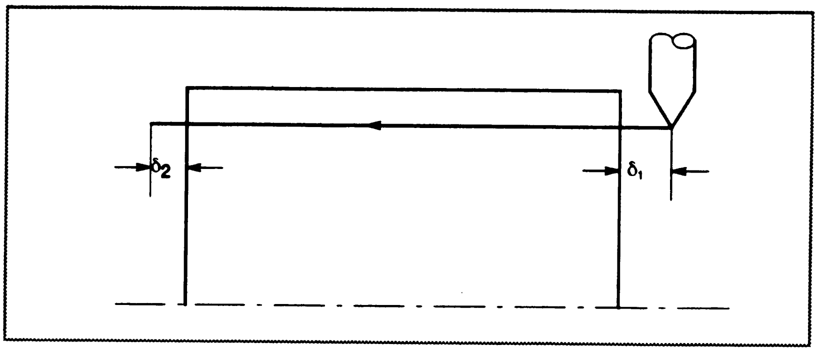


Fig. D.2 (a) Incorrect threaded portion

Explanations

- **How to determine δ_2**

$$\delta_2 = \frac{LR}{1800*} \text{ (mm)}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

- **How to determine δ_1**

$$\delta_1 = \frac{LR}{1800*}(-1-lna) \text{ (mm)}$$
$$= \delta_2(-1-lna) \text{ (mm)}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

Following a is a permited value of thread.

a	−1−lna
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

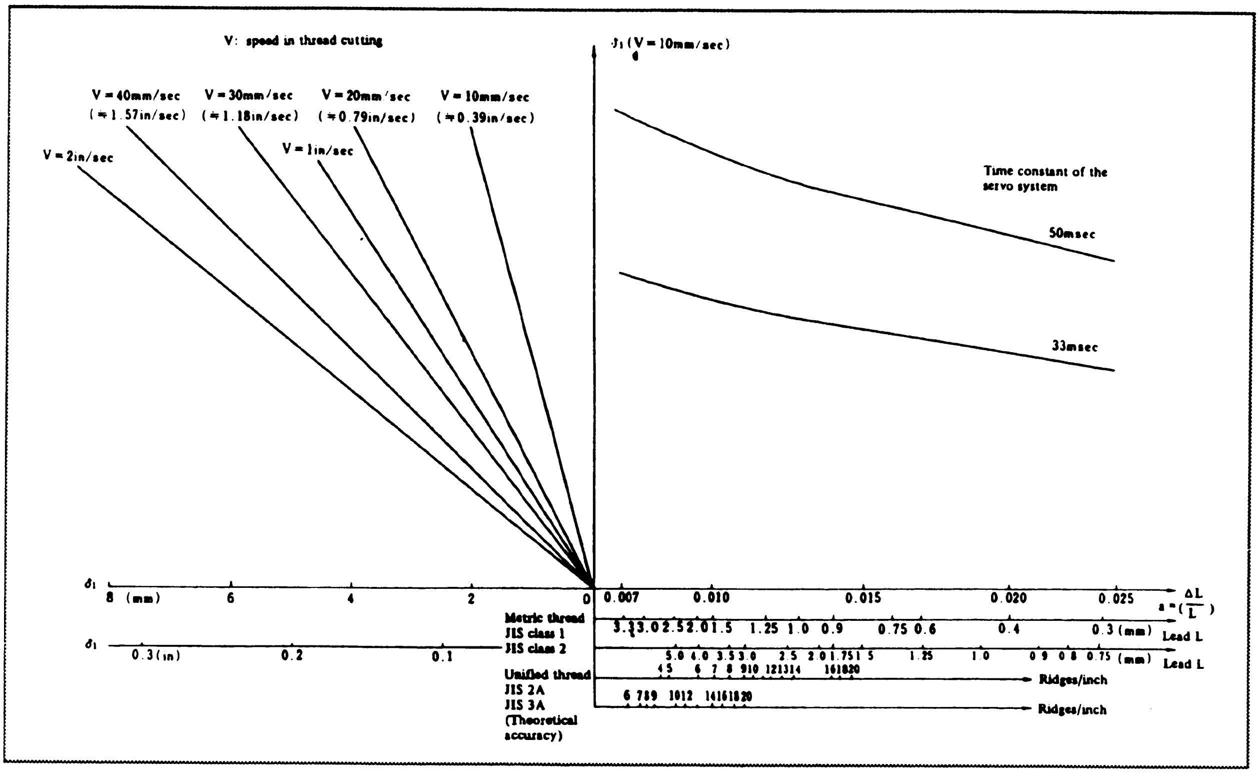
Examples

R=350rpm
L=1mm
a=0.01 then

$$\delta_2 = \frac{350 \times 1}{1800} = 0.194 \text{ (mm)}$$

$$\delta_1 = \delta_2 \times 3.605 = 0.701 \text{ (mm)}$$

- **Reference**



Nomograph for obtaining approach distance δ_1

D.3 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. D.3 (a).

Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

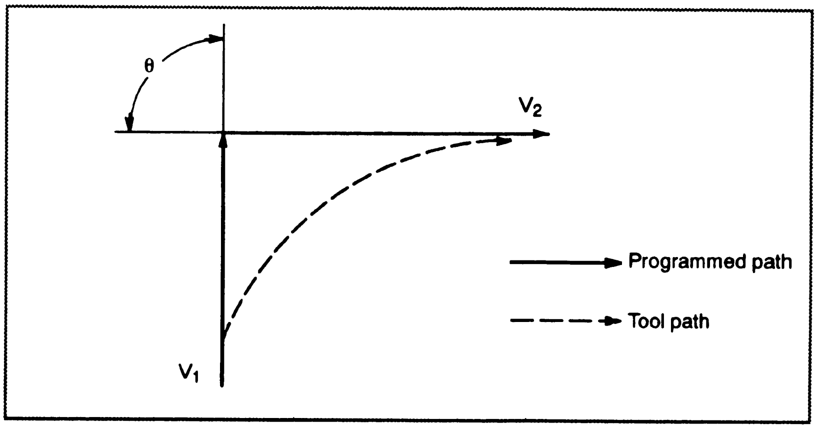


Fig. D.3 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:

- **Feedrate (V_1, V_2)**
- **Corner angle (θ)**
- **Exponential acceleration / deceleration time constant (T_1) at cutting (T_1 = 0)**
- **Presence or absence of buffer register.**

The above parameters are used to theoretically analyze the tool path and above tool path is drawn with the parameter which is set as an example. When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.

In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

Analysis

The tool path shown in Fig. D.3 (b) is analyzed based on the following conditions:
Feedrate is constant at both blocks before and after cornering.
The controller has a buffer register. (The error differs with the reading speed of the tape reader, number of characters of the next block, etc.)

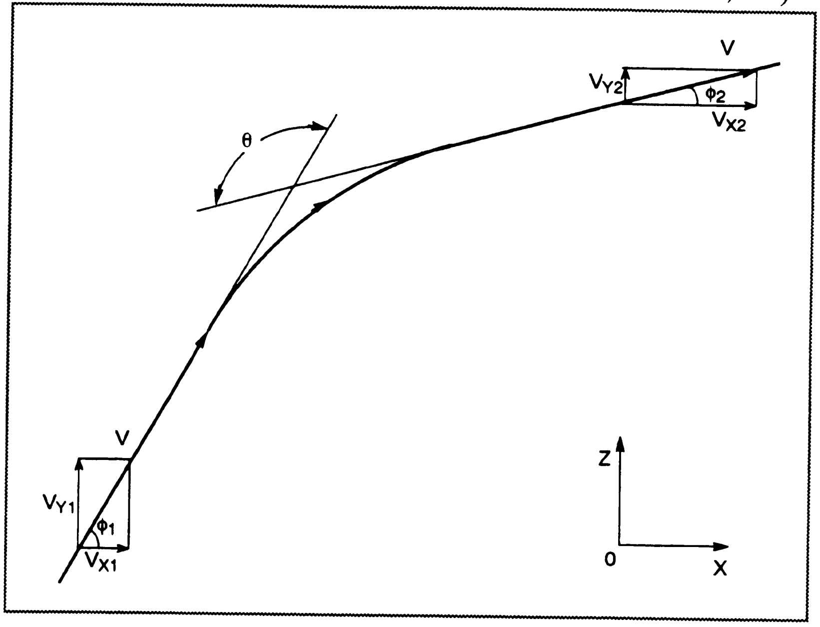


Fig. D.3(b) Example of tool path

- **Description of conditions and symbols**

$$V_{X1} = V\cos\phi_1$$
$$V_{Y1} = V\sin\phi_1$$
$$V_{X2} = V\cos\phi_2$$
$$V_{Y2} = V\sin\phi_2$$

V : Feedrate at both blocks before and after cornering
V_{X1} : X–axis component of feedrate of preceding block
V_{Y1} : Y–axis component of feedrate of preceding block
V_{X2} : X–axis component of feedrate of following block
V_{Y2} : Y–axis component of feedrate of following block
θ : Corner angle
ϕ_1 : Angle formed by specified path direction of preceding block and X–axis
ϕ_2 : Angle formed by specified path direction of following block and X–axis

• Initial value calculation

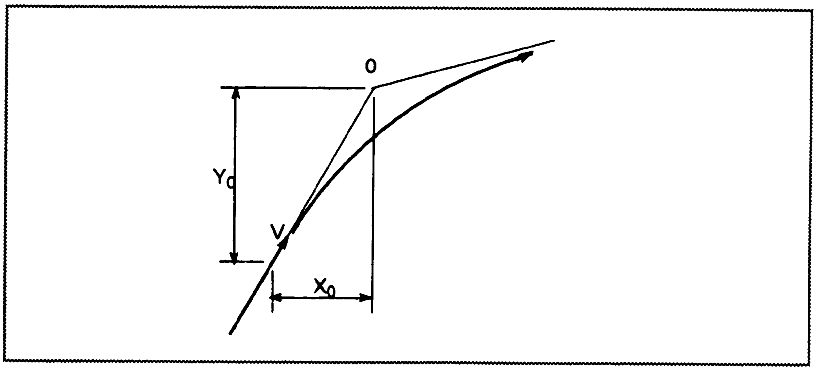


Fig. D.3(c) Initial value

The initial value when cornering begins, that is, the X and Y coordinates at the end of command distribution by the controller, is determined by the feedrate and the positioning system time constant of the servo motor.

$$X_0 = V_{X1}(T_1 + T_2)$$

$$Y_0 = V_{Y1}(T_1 + T_2)$$

T_1:Exponential acceleration / deceleration time constant. (T=0)
T_2:Time constant of positioning system (Inverse of position loop gain)

• Analysis of corner tool path

The equations below represent the feedrate for the corner section in X–axis direction and Y–axis direction.

$$V_X(t) = (V_{X2}-V_{X1})[1-\frac{V_{X1}}{T_1-T_2}\{T_1\exp(-\frac{t}{T_1})-T_2\exp(-\frac{t}{T_2})\} + V_{X1}]$$

$$= V_{X2}[1-\frac{V_{X1}}{T_1-T_2}\{T_1\exp(-\frac{t}{T_1})-T_2\exp(-\frac{t}{T_2})\}]$$

$$V_Y(t) = \frac{V_{Y1}-V_{Y2}}{T_1-T_2}\{T_1\exp(-\frac{t}{T_1})-T_2\exp(-\frac{t}{T_2})\} + V_{Y2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t) = \int_0^t V_X(t)dt-X_0$$

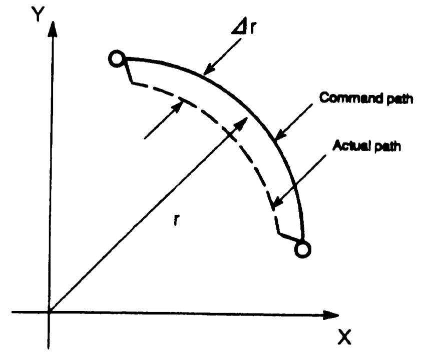
$$= \frac{V_{X2}-V_{X1}}{T_1-T_2}\{T_1^2\exp(-\frac{t}{T_1})-T_2^2\exp(-\frac{t}{T_2})\}-V_{X2}(T_1 + T_2-t)$$

$$Y(t) = \int_0^t V_Y(t)dt-Y_0$$

$$= \frac{V_{Y2}-V_{Y1}}{T_1-T_2}\{T_1^2\exp(-\frac{t}{T_1})-T_2^2\exp(-\frac{t}{T_2})\}-V_{Y2}(T_1 + T_2-t)$$

D.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING

When a servo motor is used, the positioning system causes an error between input commands and output results. Since the tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, sepecially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}(T_1^{\,2} + T_2^{\,2})\frac{V^2}{r} \quad \cdots\cdots\cdot \ (1)$$

Δr : Maximum radius error (mm)
v : Feedrate (mm/s)
r : Circle radius (mm)
T_1 : Exponential acceleration/deceleration time constant (sec) at cutting (T=0)
T_2 : Time constant of positoning system (sec). (Inverse of positon loop gain)

In the case of linear acceleration/deceleration after cutting feed interpolation, an approximation of this radius error can be obtained with the following expression:

$$\Delta r = (\frac{1}{24}T_1^{\,2} + \frac{1}{2}T_2^{\,2})\frac{V^2}{r}$$

Thus, the radius error in the case of linear acceleration/deceleration after interpolation is smaller than in case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo loop time constant.

Since the machining radius r (mm) and allowable error Δr (mm) of the workpiece is given in actual machining, the allowable limit feedrate v (mm /sec) is determined by equation (1).
Since the acceleration/deceleration time constant at cutting which is set by this equipment varies with the machine tool, refer to the manual issued by the machine tool builder.

E 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 :
O: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	O	O	O
	Data set by the MDI setting operation	O	O	O
	Parameter	O	O	O
Various data	Programs in memory	O	O	O
	Contents in the buffer storage	×	×	O : MDI mode × : Other mode
	Display of sequence number	O	O (Note 1)	O (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.)	O
	F	Zero	Zero	O
	S, T, M	×	O	O
	K (Number of repeats)	×	×	×
Work coordinate value		Zero	O	O
Action in operation	Movement	×	×	×
	Dwell	×	×	×
	Issuance of M, S and T codes	×	×	×
	Tool length compensation	×	Depending on parameter LVK(No.5003#6)	O : MDI mode Other modes depend on parameter LVK(No.5003#6).
	Cutter compensation	×	×	O : MDI mode × : Other modes
	Storing called subprogram number	×	× (Note 2)	O : MDI mode × : Other modes (Note 2)

Item		When turning power on	Cleared	Reset
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	×	O (× : Emergency stop)	O (× : Emergency stop)
	S, T codes	×	O	O
	M code	×	×	×
	M, S and T strobe signals	×	×	×
	Spindle revolution signal (S analog signal)	×	O	O
	CNC ready signal MA	ON	O	O
	Servo 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)	×	×	×

Notes

1. When heading is performed, the main program number is displayed.
2. When a reset is performed during execution of a subprogram, control returns the main program.
   Execution cannot be started from the middle of the subprogram.

F CHARACTER-TO-CODES CORRESPONDENCE TABLE

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		"	034	Quotation mark
H	072		#	035	Hash sign
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		'	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		–	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	HAtI mark
1	049		[	091	Left square bracket
2	050		^	092	
3	051		]	094	Right square bracket
4	052		_	095	Underscore
5	053				

G ALARM LIST

1) Program errors (P/S alarm)

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign " – " was input without an address at the beginning of a block. Modify the program .
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign " ." input error (Sign " – " was input after an address with which it cannot be used. Or two or more " – " signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point " – " input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G–CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
014	CAN NOT COMMAND G95	G95 (feed per rotation) cannot be specified. Modify the program.
015	TOO MANY AXES COMMANDED	The number of the commanded axes exceeded that of simultaneously controlled axes.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.
021	ILLEGAL PLANE AXIS COMMANDED	An axis not included in the selected plane (by using G17, G18, G19) was commanded in circular interpolation. Modify the program.
027	NO AXES COMMANDED IN G43/G44	No axis is specified in G43 and G44 blocks for the tool length offset type C. Offset is not canceled but another axis is offset for the tool length offset type C. Modify the program.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
029	ILLEGAL OFFSET VALUE	The offset values specified by H code is too large. Modify the program.

Number	Message	Contents
030	ILLEGAL OFFSET NUMBER	The offset number specified by D/H code for tool length offset or cutter compensation is too large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for cutter compensation C. Modify the program.
034	NO CIRC ALLOWED IN ST–UP /EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in cutter compensation C. Modify the program.
036	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in cutter compensation mode. Modify the program.
037	CAN NOT CHANGE PLANE IN CRC	The plane selected by using G17, G18 or G19 is changed in cutter compensation C mode. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in cutter compensation C because the arc start point or end point coincides with the arc center. Modify the program.
041	INTERFERENCE IN CRC	Overcutting will occur in cutter compensation C. Two or more blocks are consecutively specified in which functions such as the auxiliary function and dwell functions are performed without movement in the cutter compensation mode. Modify the program.
044	G27–G30 NOT ALLOWED IN FIXED CYC	One of G27 to G30 is commanded in canned cycle mode. Modify the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2 are commanded for 2nd reference position return command.
059	PROGRAM NUMBER NOT FOUND	In an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background eiting.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63. Delete unnecessary programs and execute program registeration again.
073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registeration again.
074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
075	PROTECT	An attempt was made to register a program whose number was protected.
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.

Number	Message	Contents
079	PROGRAM VERIFY ERROR	In memory or program collation,a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
090	REFERENCE RETURN INCOMPLETE	The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return.
092	AXES NOT ON THE REFERENCE POINT	The commanded axis by G27 (Reference position return check) did not return to the reference position.
094	P TYPE NOT ALLOWED (COORD CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the coordinate system setting operation was performed.) Perform the correct operation according to th operator's manual.
095	P TYPE NOT ALLOWED (EXT OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the external workpiece offset amount changed.)
096	P TYPE NOT ALLOWED (WRK OFS CHG)	P type cannot be specified when the program is restarted. (After the automatic operation was interrupted, the workpiece offset amount changed.)
097	P TYPE NOT ALLOWED (AUTO EXEC)	P type cannot be directed when the program is restarted. (After power ON, after emergency stop or P/S alarm 94 to 97 were reset, no automatic operation is performed.) Perform automatic operation.
098	G28 FOUND IN SEQUENCE RETURN	A command of the program restart was specified without the reference position return operation after power ON or emergency stop, and G28 was found during search. Perform the reference position return.
099	MDI EXEC NOT ALLOWED AFT. SEARCH	After completion of search in program restart, a move command is given with MDI.
100	PARAMETER WRITE ENABLE	On the PARAMETER(SETTING) screen, PWE(parameter writing enabled) is set to 1. Set it to 0, then reset the system.
101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. If this alarm has occurred, press <RESET> while pressing <PROG>, and only the program being edited will be deleted. Register the deleted program.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.
111	CALCULATED DATA OVERFLOW	The result of calculation is out of the allowable range (-10^{47} to -10^{-29}, 0, and 10^{-29} to 10^{47}).
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°)
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.

Number	Message	Contents
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high–speed cycle cutting.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative, BCD argument is negative, or other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE MACRO MODAL–CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, 1≦ n ≦3 is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation > is used. Modify the program.
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
141	CAN NOT COMMAND G51 IN CRC	G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.
142	ILLEGAL SCALE RATE	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting (G51 Pp . . or parameter 5411 or 5421).
143	SCALED MOTION DATA OVERFLOW	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling mangification.
144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
200	ILLEGAL S CODE COMMAND	In the rigid tap, an S value is out of the range or is not specified. The maximum value for S which can be specified in rigid tapping is set in parameter (No.5241 to 5243). Change the setting in the parameter or modify the program.
201	FEEDRATE NOT FOUND IN RIGID TAP	In the rigid tapping, no F value is specified. Correct the program.
202	POSITION LSI OVERFLOW	In the rigid tapping, spindle distribution value is too large.

Number	Message	Contents
203	PROGRAM MISS AT RIGID TAP-PING	In the rigid tapping, position for a rigid M code (M29) or an S command is incorrect. Modify the program.
204	ILLEGAL AXIS OPERATION	In the rigid tapping, an axis movement is specified between the rigid M code (M29) block and G84 (G74) block. Modify the program.
205	RIGID MODE DI SIGNAL OFF	Rigid tapping signal (DGNG 061#1) is not 1 when G84 (G74) is executed though the rigid M code (M29) is specified.Consult the PMC ladder diagram to find the reason the DI signal is not turned on. Modify the program.
206	CAN NOT CHANGE PLANE (RIGID TAP)	Plane changeover was instructed in the rigid mode. Correct the program.
207	RIGID DATA MISMATCH	The specified distance was too short or too long in rigid tapping.
210	CAN NOT COMAND M198/M99	M198 and M99 are executed in the schedule operation. Or M198 is executed in the DNC operation.
224	RETURN TO REFERENCE POINT	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when bit 0 of parameter 1005 ZRN_X is 0.
231	ILLEGAL FORMAT IN G10 OR L50	Any of the following errors occurred in the specified format at the programmable–parameter input. 1) Address N or R was not entered. 2) A number not specified for a parameter was entered. 3) The axis number was too large. 4) An axis number was not specified in the axis–type parameter. 5) An axis number was specified in the parameter which is not an axis type. 6) An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PSSWD) when they are protected by a password. Correct the program.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.
239	BP/S ALARM	While punching was being performed with the function for controlling external I/O units ,background editing was performed.
240	BP/S ALARM	Background editing was performed during MDI operation.
5010	END OF RECORD	The end of record (%) was specified.

2) Background edit alarm

Number	Message	Contents
070 to 074	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit.
085 to 087	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. **(Note)** Use background editing correctly.

Note
Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.

3) Absolute pulse coder (APC) alarm

Number	Message	Contents
300	nth–axis origin return	Manual reference position return is required for the nth–axis (n=1 – 8).
301	APC alarm: nth–axis communication	nth–axis (n=1 – 3) APC communication error. Failure in data transmission Possible causes include a faulty APC, cable, or servo interface module.
302	APC alarm: nth–axis over time	nth–axis (n=1 – 3) APC overtime error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
303	APC alarm: nth–axis framing	nth–axis (n=1 – 3) APC framing error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
304	APC alarm: nth–axis parity	nth–axis (n=1 – 3) APC parity error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
305	APC alarm: nth–axis pulse error	nth–axis (n=1 – 3) APC pulse error alarm. APC alarm.APC or cable may be faulty.
306	APC alarm: nth–axis battery voltage 0	nth–axis (n=1 – 3) APC battery voltage has decreased to a low level so that the data cannot be held. APC alarm. Battery or cable may be faulty.
307	APC alarm: nth–axis battery low 1	nth–axis (n=1 – 3) axis APC battery voltage reaches a level where the battery must be renewed. APC alarm. Replace the battery.
308	APC alarm: nth–axis battery low 2	nth–axis (n=1 – 3) APC battery voltage has reached a level where the battery must be renewed (including when power is OFF). APC alarm .Replace battery.
309	APC alarm: nth–axis ZRN IMPOSSIBL	nth–axis (n=1 – 4) A reference position return can not be mode because the serial α pulse coder does not rotate more than one turn.

4) Serial pulse coder (SPC) alarms

When either of the following alarms is issued, a possible cause is a faulty serial pulse coder or cable.

Number	Message	Contents
350	SPC ALARM: n AXIS PULSE CODER	The n axis (axis 1–3) pulse coder has a fault. Refer to diagnosis display No. 202 or No. 204 for details.
351	SPC ALARM: n AXIS COMMUNICATION	n axis (axis 1–3) serial pulse coder communication error (data transmission fault) Refer to diagnosis display No. 203 for details.

- **The details of serial α pulse coder alarm No.350**

The explanation of serial pulse coder alarm No. 350 (pulse coder alarm) appears in the diagnosis display (Nos. 202 and 204). If alarm No. 351 is output together with alarm No. 350, see the explanations of alarm No. 351.

How to determine the cause of an alarm is explained in the FANUC Series 20–FA Maintenance Manual (B–62175E) and the Servo Amplifier Maintenance Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
0202					RCA			SPH

#3 (RCA): Count error. The serial α pulse coder may be defective.

#0 (SPH): Abnormal software phase data. The serial α pulse coder or feedback cable may be defective.

	#7	#6	#5	#4	#3	#2	#1	#0
0204				LDA	PMS			

#4 (LDA): Abnormality of the serial α pulse coder LED. The serial α pulse coder may be defective.

#3 (PMS): Abnormal feedback pulse. The serial α pulse coder or feedback cable (between the pulse coder and servo amplifier module, or between the servo amplifier module and main board) may be defective.

- **The details of serial α pulse coder alarm No.351**

The explanation of serial pulse coder alarm No. 351 (communication alarm) appears in the diagnosis display (No. 203).

How to determine the cause of an alarm is explained in the FANUC Series 20–FA Maintenance Manual (B–62175E) and the Servo Amplifier Maintenance Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
0203	DTE	CRC	STB					

#7 (DTE): No response is returned from the serial α pulse coder. Possible causes are the malfunctioning of the serial pulse coder, breakage of the signal cable, or a +5 V drop in the voltage being supplied to the pulse coder.

#6 (CRC): Serial communication failure.

#5 (STB): One of the following may be defective: signal cable (between the pulse coder and servo amplifier module, or between the servo amplifier module or main board), serial α pulse coder, servo interface module, servo control module, or main board.

5) Servo alarms

Number	Message	Contents
400	SERVO ALARM: n–TH AXIS OVERLOAD	The n–th axis (axis 1–3) overload signal is on. Refer to diagnosis display No. 200 or No. 201 for details.
401	SERVO ALARM: n–TH AXIS VRDY OFF	The n–th axis (axis 1–3) servo amplifier READY signal (DRDY) went off.
404	SERVO ALARM: n–TH AXIS VRDY ON	Even though the n–th axis (axis 1–8) READY signal (MCON) went off, the servo amplifier READY signal (DRDY) is still on. Or, when the power was turned on, DRDY went on even though MCON was off. Check that the servo interface module and servo amp are connected.
405	SERVO ALARM: (ZERO POINT RETURN FAULT)	Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.
410	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–3) stops is larger than the set value. Note) Limit value must be set to parameter No.1829 for each axis.
411	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–3) moves is larger than the set value. Note) Limit value must be set to parameter No.1828 for each axis.
413	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis (axis 1–3) are beyond the range of -2^{31} to 2^{31}. This error usually occurs as the result of an improperly set parameters.
414	SERVO ALARM: n–TH AXIS – DETECTION RELATED ERROR	N–th axis (axis 1–3) digital servo system fault. Refer to diagnosis display No. 200, No. 201, and No.204 for details.
415	SERVO ALARM: n–TH AXIS – EXCESS SHIFT	A speed higher than 511875 units/s was attempted to be set in the n–th axis (axis 1–3). This error occurs as the result of improperly set CMR.
416	SERVO ALARM: n–TH AXIS – DISCONNECTION	Position detection system fault in the n–th axis (axis 1–3) pulse coder (disconnection alarm). Refer to diagnosis display No. 200 and No. 201 for details.
417	SERVO ALARM: n–TH AXIS – PARAMETER INCORRECT	This alarm occurs when the n–th axis (axis 1–3) is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 2020 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No.2022 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 2023 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 2024 (number of position feedback pulses per motor revolution). 5) Parameters No. 2084 and No. 2085 (flexible field gear rate) have not been set. 6) A value outside the limit of {1 to the number of control axes} or a non–continuous value (Parameter 1023 (servo axis number) con tains a value out of the range from 1 to the number of axes, or an isolated value (for example, 3 not prceded by 2).was set in parame ter No. 1023 (servo axisnumber).

• Details of servo alarm No. 400, No.414 and No. 416

The explanations of serial pulse coder alarm Nos. 400, 414, and 416 (pulse coder alarms) appear in the diagnosis display (Nos. 201, 202, and 204).
How to determine the cause of an alarm is explained in the FANUC Series 20–FA Maintenance Manual (B–62175E) and the Servo Amplifier Maintenance Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
0200	OVL	LV	OVC	HCA	HVA	DCA	FBA	OFA

	#7	#6	#5	#4	#3	#2	#1	#0
0201	ALD			EXP				

	#7	#6	#5	#4	#3	#2	#1	#0
0204	RAM	OFS	MCC			FSA		

(1) Servo alarm No. 400 (Overload)

Bit converted to "1"	Alarm details
OVL	Overheat (Power supply module unit)
OVL + EXP	Overheat (Servo amplifier module unit)
OVL + ALD	Overheat (Servo motor)

(2) Servo alarm No. 414 (Detection error)

Bit converted to "1"	Alarm details
OFA	Digital servo overflow
DCA	Regenerative discharge circuit alarm
HVA	High voltage alarm (Power supply module unit)
HCA	Abnormal current alarm (Power supply module unit)
HCA + EXP	Abnormal current alarm (Servo amplifier module unit)
OVC	Overcurrent alarm
LV	Low voltage alarm (Power supply module unit) (Power source unit)
LV + EXP	Low voltage alarm (Servo amplifier module unit)
LV + ALD	Low voltage alarm (Power supply module unit) (Main unit)
LV + EXP + ALD	Low voltage alarm (Servo amplifier module unit) (Main unit)
FSA	Fan stop alarm (Power supply module unit)
FSA + EXP	Fan stop alarm (Servo amplifier module unit)
MCC	Charge failure (Power supply module unit)
OFS	Abnormal current detected in digital servo module
RAM	RAM check error in servo control module

(3) Servo alarm No. 416 (Disconnection alarm)

Bit converted to "1"	Alarm details
FBA	Pulse coder disconnection (Software)
FBA + ALD	Built–in serial pulse coder disconnection
FBA + EXP + ALD	Separate position detector disconnection

6) Over travel alarms

Number	Message	Contents
500	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit I. (Parameter No.1320 or 1326 **Notes**)
501	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit I. (Parameter No.1321 or 1327 **Notes**)
502	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit II. (Parameter No.1322)
503	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit II. (Parameter No.1323)
506	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side hardware OT.
507	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side hardware OT.

7) Overheat alarms

Number	Message	Contents
700	OVERHEAT: CONTROL UNIT	Control unit overheat Check that the fan motor operates normally, and clean the air filter.
701	OVERHEAT: FAN MOTOR	The fan motor on the top of the cabinet for the contorl unit is overheated. Check the operation of the fan motor and replace the motor if necessary.

8) Spindle alarms

Number	Message	Contents
750	SPINDLE SERIAL LINK START FAULT	This alarm is generated when the spindle control unit is not ready for starting correctly when the power is turned on in the system with the serial spindle. The four reasons can be considered as follows: 1) An improperly connected cable, or the spindle control unit's power is OFF. 2) When the NC power was turned on under alarm conditions other than SU–01 or AL–24 which are shown on the LED display of the spindle control unit. In this case, turn the spindle amplifier power off once and perform startup again. 3) Other reasons (improper combination of hardware) This alarm does not occur after the system including the spindle control unit is activated. 4) The second spindle (when SP2, bit 4 of parameter No. 3701, is 1) is in one of the above conditions 1) to 3). See diagnostic display No. 409 for details.
749	S–SPINDLE LSI ERROR	A communication error occurred for the serial spindle. The cause may be the disconnection of an optical cable or the interruption of the power to the spindle amplifier. (Note) Unlike alarm No. 750, this alarm occurs when a serial communication alarm is detected after the spindle amplifier is normally activated.

Number	Message	Contents
751	FIRST SPINDLE ALARM DETECTION (AL–XX)	This alarm indicates in the NC that an alarm is generated in the spindle unit of the system with the serial spindle. The alarm is displayed in form AL–XX (XX is a number). Refer to **(11) Alarms displayed on spindle servo unit** .The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.
752	FIRST SPINDLE MODE CHANGE FAULT	This alarm is generated if the system does not properly terminate a mode change. The modes include the rigid tapping and spindle control modes. The alarm is activated if the spindle control unit does not respond correctly to the mode change command issued by the NC.

- **The details of spindle alarm No.750**

The details of spindle alarm No. 750 are displayed in the diagnosis display (No. 409) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
409					SPE	S2E	S1E	SHE

SPE **0** : In the spindle serial control, the serial spindle parameters fulfill the spindle unit startup conditions.
1 : In the spindle serial control, the serial spindle parameters do not fulfill the spindle unit startup conditions.

S2E Unused (0 is always displayed.)

S1E **0** : The first spindle is normal during the spindle serial control startup.
1 : The first spindle was detected to have a fault during the spindle axis serial control startup.

SHE **0** : The serial communications module in the CNC is normal.
1 : The serial communications module in the CNC was detected to have a fault.

9) System alarms

(These alarms cannot be reset with reset key.)

Number	Message	Contents
900	ROM PARITY	F–ROM parity error in a ROM file (control software), such as CNC, macro, or digital servo. The F–ROM module may be defective.
910	DRAM PARITY : (Low)	For an SRAM parity error, initialize the memory. If the error subsequently recurs, or in the case of a DRAM parity error, replace the RAM module. Subsequently, set the parameters and all other data again.
911	DRAM PARITY: (High)	
912	SRAM PARITY: (Low)	
913	SRAM PARITY : (High)	
920	SERVO ALARM (1/2 AXIS)	Servo alarm (1st or 2nd axis). A watchdog alarm or a RAM parity error in the servo module occurred. Replace the servo control module on the main CPU board.
921	SERVO ALARM (3/4 AXIS)	Servo alarm (3rd or 4th axis). A watchdog alarm or a RAM parity error in the servo module occurred. Replace the servo control module on the main CPU board.
924	SERVO MODULE SETTING ERROR	The digital servo module is not installed. Check that the servo control module or servo interface module on the main board is mounted securely.
930	CPU INTERRUPUT	CPU error (abnormal interrupt) The main board is faulty.
940	PCB ERROR	PCB ID error. The main board or the memory module may be defective.
945	SERIAL SPINDLE COMMUNICATION ERROR	Communications error occurred in the serial spindle. The optical fiber may be disconnected.
950	PMC SYSTEM ALARM	Fault occurred in the PMC. The PMC control module on the main board or the RAM module may be defective.
971	NMI OCCURRED IN SLC	An alarm condition occurred in the interface with an I/O unit. For PMC–RA1 or PMC–RA3, check the connection between the PMC control module on the main board and the I/O Unit. Also, check that the power of the I/O Unit is on and that the interface module is operating normally.
972	NMI OCCURRED IN OTHER MODULE	NMI occurred in a board other than the main board. The main board or the back panel may be defective.
973	NON MASK INTERRUPT	NMI occurred for an unknown reason. The printed board of the power unit or the main board may be defective. Or, there may be noise interference.
974	BUS ERROR	Bus error The main board may be defective.

(10) Alarms Displayed on spindle Servo Unit

Alarm No.	Meaning	Description	Remedy
"A" display	Program ROM abnormality (not installed)	Detects that control program is not started (due to program ROM not installed, etc.)	Install normal program ROM
AL01	Motor overheat	Detects motor speed exceeding specified speed excessively.	Check load status. Cool motor then reset alarm.
AL02	Excessive speed deviation	Detects motor speed exceeding specified speed excessively.	Check load status. Reset alarm.
AL03	DC link section fuse blown	Detects that fuse F4 in DC link section is blown (models 30S and 40S).	Check power transistors, and so forth. Replace fuse.
AL04	Input fuse blown. Input power open phase.	Detects blown fuse (F1 to F3), open phase or momentary failure of power (models 30S and 40S).	Replace fuse. Check open phase and power supply refenerative circuit operation.
AL05	Control power supply fuse blown	Detects that control power supply fuse AF2 or AF3 is blown (models 30S and 40S).	Check for control power supply short circuit . Replace fuse.
AL–07	Excessive speed	Detects that motor rotation has exceeded 115% of its rated speed.	Reset alarm.
AL–08	High input voltage	Detects that switch is flipped to 200 VAC when input voltage is 230 VAC or higher (models 30S and 40S).	Flip switch to 230 VAC.
AL–09	Excessive load on main circuit section	Detects abnormal temperature rise of power transistor radiator.	Cool radiator then reset alarm.
AL–10	Low input voltage	Detects drop in input power supply voltage.	Remove cause, then reset alarm.
AL–11	Overvoltage in DC link section	Detects abnormally high direct current power supply voltage in power circuit section.	Remove cause, then reset alarm.
AL–12	Overcurrent in DC link section	Detects flow of abnormally large current in direct current section of power circtcuit	Remove cause, then reset alarm.
AL–13	CPU internal data memory abnormality	Detects abnormality in CPU internal data memory. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–15	Spindle switch/output switch alarm	Detects incorrect switch sequence in spindle switch/output switch operation.	Check sequence.
AL–16	RAM abnormality	Detects abnormality in RAM for external data. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–18	Program ROM sum check error	Detects program ROM data error.This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–19	Excessive U phase current detection circuit offset	Detects excessive U phase current detection ciucuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–20	Excessive V phase current detection circuit offset	Detects excessive V phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–24	Serial transfer data error	Detects serial transfer data error (such as NC power supply turned off, etc.)	Remove cause, then reset alarm.
AL–25	Serial data transfer stopped	Detects that serial data transfer has stopped.	Remove cause, then reset alarm.
AL–26	Disconnection of speed detection signal for Cs contouring control	Detects abnormality in position coder signal(such as unconnected cable and parameter setting error).	Remove cause, then reset alarm.

Alarm No.	Meaning	Description	Remedy
AL–27	Position coder signal disconnection	Detects abnormality in position coder signal (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL–28	Disconnection of position detection signal for Cs contouring control	Detects abnormality in position detection signal for Cs contouring control (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL–29	Short–time overload	Detects that overload has been continuously applied for some period of time (such as restraining motor shaft in positioning).	Remove cause, then reset alarm.
AL–30	Input circuit overcurrent	Detects overcurrent flowing in input circuit.	Remove cause, then reset alarm.
AL–31	Speed detection signal disconnection motor restraint alarm or motor is clamped.	Detects that motor cannot rotate at specified speed or it is detected that the motor is clamped. (but rotates at very slow speed or has stopped). (This includes checking of speed detection signal cable.)	Remove cause, then reset alarm.
AL–32	Abnormality in RAM internal to LSI for serial data transfer. This check is made only when power is turned on.	Detects abnormality in RAM internalto LSI for serial data transfer. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–33	Insufficient DC link section charging	Detects insuffieient charging of direct current power supply voltage in power circuit section when magnetic contactor in amplifier is turned on (such as open phase and defectifve charging resistor).	Remove cause, then reset alarm.
AL–34	Parameter data setting beyond allowable range of values	Detects parameter data set beyond allowable range of values.	Set correct data.
AL–35	Excesive gear ratio data setting	Detects gear ratio data set beyond allowable range of values.	Set correct data.
AL–36	Error counter over flow	Detects error counter overflow.	Correct cause, then reset alarm.
AL–37	Speed detector parameter setting error	Detects incorrect setting of parameter for number of speed detection pulses.	Set correct data.
AL–39	Alarm for indicating failure in detecting 1–rotation signal for Cs contouring control	Detects 1–rotaion signal detection failure in Cs contouring contorl.	Make 1–rotaion signal adjustment. Check cable shield status.
AL–40	Alarm for indicating 1–rotation signal for Cs contouring control not detected	Detects that 1–rotation signal has not occurred in Cs contouring control.	Make 1–rotaion signal adjustment.
AL–41	Alarm for indicating failure in detecting position coder 1–rotaion signal.	Detects failure in detecting position coder 1–rotation signal.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL–42	Alarm for indicating position coder 1–rotation signal not detected	Detects that position coder 1–rotation signal has not issued.	Make 1–rotation signal adjustment for signal conversion circuit.
AL–43	Alarm for indicating disconnection of position coder signal for differential speed mode	Detects that main spindle position coder signal used for differential speed mode is not connected yet (or is disconnected).	Check that main spindle position coder signal is connected to connector CN12.
AL–46	Alarm for indicating failure in detecting position coder 1–rotation signal in thread cutting operation.	Detects failure in detecting position coder 1–rotation signasl in thread cutting operation.	Make 1–rotation signal adjustment for signal conversion circuit Check cable shield status.

Alarm No.	Meaning	Description	Remedy
AL–47	Position coder signal abnormality	Detects incorrect position coder signal count operation.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL–48	Position coder 1–rotation signal abnormailty	Detects that occurrence of position coder 1–rotation signal has stopped.	Make 1–rotation signal adjustment for signal conversion circuit.
AL–49	The converted differential speed is too high.	Detects that speed of other spindle converted to speed of local spindle has exceeded allowable limit in differential mode.	Check the position coder state of the other side.
AL–50	Excessive speed command calculation value in spindle synchronization control	Detects that speed command calculation value exceeded allowable range in spindle synchronization control.	Check parameters such as a position gain.
AL–51	Undervoltage at DC link section	Detects that DC power supply voltage of power ciucuit has dropped (due to momentary power failure or loose contact of magnetic contactor).	Remove cause, then reset alarm.
AL–52	ITP signal abnormality I	Detects abnormality in synchronization signal (ITP signal)with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.
AL–53	ITP signal abnormality II	Detects abnormality in synchronization signal (ITP signal) with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.
AL–54	Overload current alarm	Detects that excessive current flowed in motor for long time.	Check if overload operation or frequent acceleration/deceleration is performed.
AL–55	Power line abnormality in spindle switching/output switching	Detects that switch request signal does not match power line status check signal.	Check operation of magnetic contractor for power line switching. Check if power line status check signal is processed normally.

H OPERATION OF PORTABLE TAPE READER

Portable tape reader is the device which inputs the NC program and the data on the paper tape to CNC.

- **Names and descriptions of each section**

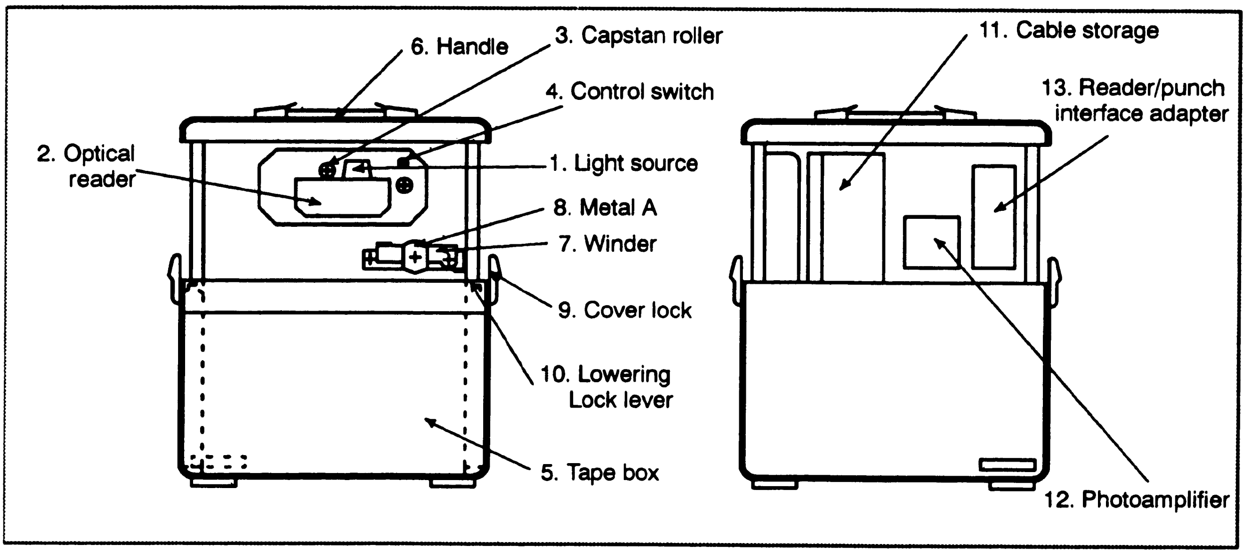
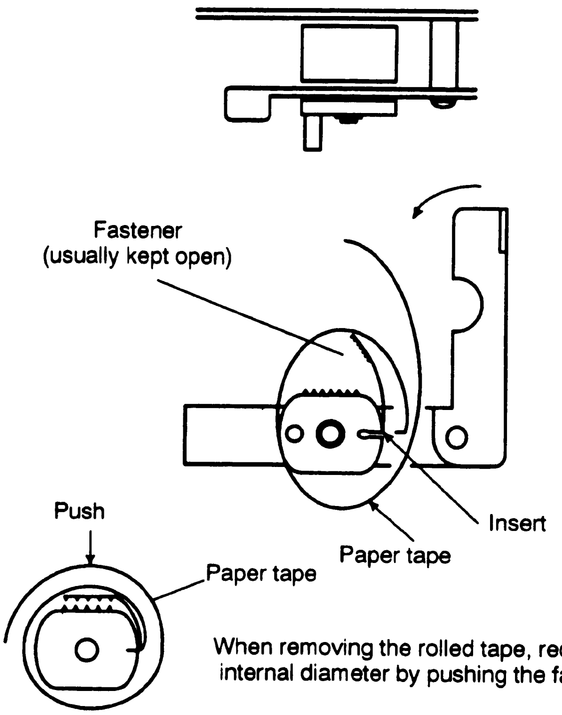


Table 1 Description of Each Section

No.	Name	Descriptions
1	Light Sources	An LED (Light emitting diode) is mounted for each channel and for the feed hole (9 diodes in total). A built–in Stop Shoe functions to decelerate the tape. The light source is attracted to the optical reader by a magnet so that the tape will be held in the correct position. This unit can be opened upward, by turning the tape reader control switch to the RELEASE position (this turns off the magnet).
2	Optical Reader	Reads data punched on the tape, through a glass window. Dust or scratches on the glass window can result in reading errors. Keep this window clean.
3	Capstan Roller	Controls the feeding of tape as specified by the control unit.
4	Tape Reader Control Switch	A 3–position switch used to control the Tape Reader. **RELEASE** ----- The tape is allowed to be free, or used to open the light-source. When loading or unloading the tape, select this position. **AUTO** -------- The tape is set to fixed position by the Stop Shoe. The feed and stop of the tape is controlled by the CNC. To input data from tape, the Light Source must be closed and this position must be selected. **MANUAL** ----- The tape can be fed in the forward reading direction. if another position is selected, the tape feed is stopped.
5	Tape Box	A Tape Box is located below the Tape Reader. A belt used to draw out a paper tape is located inside the box. The paper tape can easily be pulled out using this belt. The tape box accomodates 15 meters of tape.
6	Handle	Used to carry the tape reader.
7	Winder	Used to advance or rewind the tape.
8	Metal A	 When removing the rolled tape, reduce the internal diameter by pushing the fastener.
9	Cover lock	Be sure to use the lock for fastening the cover before carrying the tape reader.

No.	Name	Descriptions
10	Lowering lock lever	When the tape reader is raised, the latch mechanism is activated to fix the tape reader. Thus, the tape reader is not lowered. The latch is locked with the lowering lock lever. The latch is therefore not unlocked even when the tape reader is raised with the handle. When the latch is locked, the lever is horizontal. To store the tape reader in the box, push the lever to release the lock, then raise the tape reader with the handle to unlock the latch. When the latch is unlocked, the tape reader can be stored in the box. When storing the tape reader, secure it with the cover lock.
11	Cable storage	Used to store rolled power and signal cables. The cable length is 1.5 m.
12	Photoamplifier	For the tape reader
13	Reader/punch interface adapter	200 VAC input and 5 VDC output power and reader/punch interface adapter PCB

Procedure for Operating the Portable Tape Reader

Preparations

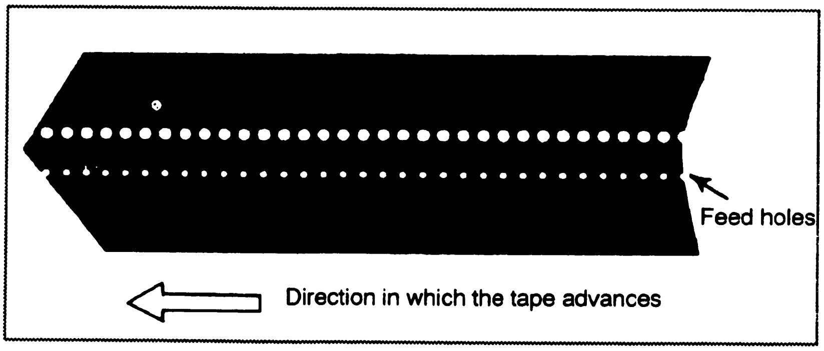
1 Unlock the cover locks **9**. Raise the tape reader with the handle **6** until it clicks, then lower the tape reader. The tape reader then appears and is secured. Check that the lowering lock levers **10** are horizontal.

2 Take out the signal and power cables from the cable storage **11** and connect the signal cable with the CNC reader/punch interface port and the power cable with the power supply.

Setting the tape

3 Turn the control switch to the RELEASE position.

4 Lift the Light Source Unit, and insert an NC tape between the gap. The tape must be positioned as shown in the figure, when viewed looking downward.



5 Pull the tape until the top of the tape goes past the Capstan roller.

6 Check that the NC tape is correctly positioned by the Tape Guide.

7 Lower the Light Source.

8 Turn the switch to the AUTO position.

9 Suspend the top and rear–end of the tape in the Tape Box.

Removing the tape

10 Turn the switch to the RELEASE position.

11 Lift the Light Source and remove the tape.

12 Lower the Light Source

Storage

13 Store the cables in the cable storage **11**.

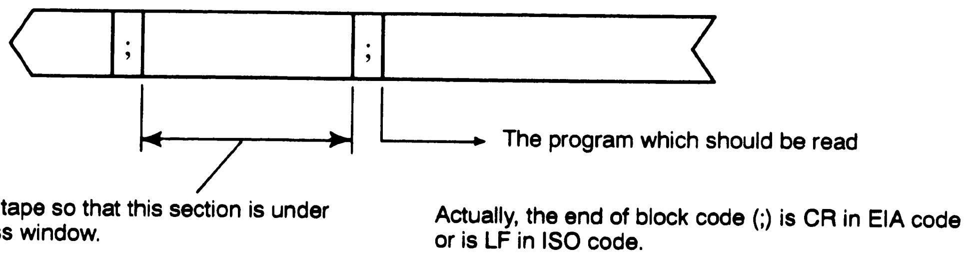
14 Push the lowering lock lever **10** at both sides down.

15 Raise the tape reader with the handle **6** to unlock the latch, then gently lower it.

16 Lock the cover lock **9** and carry the tape reader with the handle **6**.

NOTE 1 SETTING OF A TAPE

When the NC tape is loaded, the Label Skip function activates to read but skip data until first End of Block code (CR in EIA code or LF in ISO code) is read. When loading an NC tape, the location within the tape, from which data reading should be started must properly be selected and the NC tape should be set as shown in the figure below.



NOTE 2 DISCONNECTION AND CONNECTION OF A PORTABLE TAPE READER CONNECTION CABLE

Don't disconnect or connect CNC tape reader connection cable (signal cable) without turning off the CNC power supply, otherwise the PCB of the tape reader and master PCB of CNC controller may be broken.Turn off the CNC power supply before disconnecting or connecting the connection cable, accordingly.

Index

[Number]

2nd reference position return (G30), 61

[A]

A block without tool movement, 167

A block without tool movement specified at start–up, 155

A block without tool movement specified in offset mode, 167

A block without tool movement specified together with offset cancel, 173

A command to the MPG exceeding rapid traverse rate (HPF), 301

Abbreviations of arithmetic and logic operation commands, 211

Abbreviations of custom macro word, 412

About this manual, 3

Absolute and incremental programming (G90, G91), 74

Absolute commands, 19

Acceleration/decelera– tion for rapid traverse, 298

Actual feedrate display, 496

Actual feedrate display on the other screen, 496

Actual feedrate value, 496

Alarm, 325, 330, 385, 349, 365, 367

Alarm and self–diagnosis functions, 354

Alarm detail screen, 530

Alarm display, 258, 355

Alarm history display, 357

Alarm list, 567

Alarm screen, 355

Alarm status, 527

Alarms during background editing, 413

Altering a word, 403

An alarm while a program is output, 368

Angle units, 210

Another method for alarm displays, 355

Applicable models, 3

Arc radius, 42

Argument specification, 220, 226, 227, 228

Arithmetic and logic operation, 210

ATAN function, 210

Automatic acceleration/deceleration, 55

Automatic acceleration/deceleration, 55

Automatic acceleration/deceleration, 48

Automatic insertion of sequence numbers, 478

Automatic operation, 249, 307

Automatic operation control, 204

Automatic operation status, 527

Auxiliary function, 81

Auxiliary function (M function), 82

Auxiliary function lock, 341

Auxiliary function lock (See Section III–5.1), 252

Availability of manual pulse generator in jog mode (JHD), 301

Availability of manual pulse generator in TEACH IN JOG mode (THD), 301

Axis No.(P_), 241

Axis moving status/dwell status, 527

Axis name, 29

[B]

Background editing, 413

Battery for absolute pulse coder, 537

Battery for memory backup, 537

Block, 24

Block configuration (word and address), 91

Block containing G40 and I_J_K_, 174

Boring cycle (G85), 118

Boring cycle back boring cycle (G87), 122

Boring cycle (G86), 120

Boring cycle (G88), 124

Boring cycle (G89), 126

Bracket nesting, 212

Brackets, 212

Branch and repetition, 215

Buffering the next block in cutter compensation mode (G41, G42), 233

Buffering the next block in other than cutter compensation mode (G41, G42) (normally prereading one block), 232

[C]

Call, 219, 224, 229

Call nesting, 221, 224

Calling a subprogram stored in an external input/output device, 310

Cancel, 103

Cancel of the compensation function, 65

Cancellation, 224, 338

Canned cycle, 100

Canned cycle cancel (G80), 128, 138

Change from G23 to G22 in a forbidden area, 352

Change in the offset direction in the offset mode, 161

Change of modes, 298

Change of the cutter compensation value, 147

Changing by G10, 68

Changing by G92, 68

Changing parameter PASSWD, 415

Changing workpiece coordinate system, 68

character–to–codes correspondence table, 566

Check by running the machine, 251

Checking by self–diagnostic screen, 358

Checking contents of the memory, 482

Checking during search, 409

Checkpoint for the forbidden area, 351
Circular interpolation (G02,G03), 40
Clear, 329
Close command PCLOS, 238
Collation, 366, 385
Command for machine operations – miscellaneous function, 22
Comment, 421
Comment section, 87
Comments in a program, 477
Common variable, 377
Common variables, 376
Compensation function, 140
Concept of word and editing unit, 398
Conditional branch (IF statement), 215
Conditional expression, 215
Configuration of the help screen, 533
Controlled axes, 28, 29
Coordinate system, 16, 63
Coordinate system on part drawing and coordinate system specified by CNC – coordinate system, 16
Coordinate system rotation (G68, G69), 192
Coordinate value and dimension, 73
Copying a program, 394
Corner movement, 168
Correction of interference in advance, 178
Correspondence between parameter numbers and program numbers, 226, 227, 228
Creating a program with guidance programming, 417
Creating a program by cycle programming, 433
Creating programs, 416
Creating programs in TEACH IN mode, 480
Creating programs using the MDI panel, 476
Criteria for detecting interference, 176
CRT/MDI panels, 262
Current block display screen, 501
Current mode, 527
Current position , 208
Current position display , 258, 503
Current time, 528
Custom macro, 198
Cutter compensation C and coordinate system rotation, 195
Cutter compensation cancel, (offset mode cancel), 146
Cutter compensation G code in the offset mode, 166
Cutter compensation value setting, 148
Cutting feed, 50
Cutting feedrate clamp, 51
Cutting feedrate control, 52
Cutting mode, 52
Cutting speed – spindle speed function, 20
Cutting time, 514
Cycle time, 497, 514

[D]

Data and time, 514
Data input/output, 260, 361
Data output command BPRNT, 236
Data output command DPRNT, 237
Data protection key, 256, 483
decimal point programming, 76
Decimal point input, 509
Deleting blocks, 405
Deleting a block, 391, 405
Deleting a program, 393
Deleting a word, 404
Deleting all programs, 410
Deleting files, 384
Deleting more than one program by specifying a range, 411
Deleting multiple blocks, 406
Deleting one program, 410
Deleting programs, 410
Description of each display, 527
Detailed information of alarms, 529
Details of cutter compensation C, 151
Details of memory used, 506
Differences between macro calls and subprogram calls, 219
Differences from NC statements, 214
Direction of the circular interpolation, 41
Direction of the offset, 142
Disabling entry of compensation values, 510
Display, 257
Display including compensation values, 494
Display of run time and parts count, 497
Display of software configuration , 293
Display of the servo axes, 498
Display of the spindle axes, 498
Display on the other screen, 497
Display procedure for the actual feedrate on the current position display screen, 496
Display procedure for the current position screen in the workpiece coordinate system, 492
Display procedure for the current position screen with the relative coordinate system, 493
Displaying directory of floppy disk, 378
Displaying and setting the tool offset value, 509
Displaying and entering setting data, 511
Displaying and setting,, parameters, 256
Displaying and setting data, 254
Displaying and setting custom macro common variables, 518
Displaying and setting operator's setting data, 255
Displaying and setting parameters, 522
Displaying and setting pitch error compensation data, 524

Displaying and setting run time,parts count, and time, 513
Displaying and setting the software operator's panel, 519
Displaying and setting the workpiece origin offset value, 515
Displaying memory used and a list of programs, 505
Displaying the contents of a program, 388
Displaying the directory, 379
Displaying the directory of floppy disk files, 379
Displaying the floppy disk directory during file execution , 329
Displaying the program number and sequence number, 526
Displaying the program number, sequence number, and status, and warning messages for data setting or input/output operation, 526
Displaying the status and warning for data setting or input/output operation, 527
Displaying variable values, 202
Distance from the start point to the center of arc, 41
Distance moved on an arc, 41
Divisor, 213
DNC operation, 318
Drilling axis, 101
Drilling cycle counter boring cycle (G82), 112
Drilling cycle, spot drilling (G81), 110
Drilling mode, 102
Dry run, 344
Dry run (See Section III–5.4), 251
Dry run feedrate, 344
Dwell (G04), 54

[E]

Editing a PART program, 253
Editing in CNC language, 396
Editing of custom macros, 412
Editing programs, 386
Editing unit, 412
Editing with guidance programming, 388
Effective time for a forbidden area, 352
Efficient use of memory, 368
Emergency stop, 348
Emergency stop or reset status, 527
Enabling and disabling manual intervention and return, 338
End position for the arc is not on the arc, 160
Endless repetition, 329
Error codes, 356
Exact stop, 52
Exact stop (G09, G61) cutting mode (G64) tapping mode (G63), 53
Example of changing T15 to M15, 403
Example of deleting a block of No. 01234, 405
Example of deleting blocks from a block containing N01234 to a block containing N56789, 406
Example of deleting X100.0, 404
Example of inserting T15, 402
External I/O devices, 286
External motion function (G81), 139
External output commands, 236

[F]

Facing, 435
FANUC FA card, 289
FANUC floppy cassette, 288
FANUC handy file, 288
FANUC PPR, 289
FEED– FEED function, 14
Feed functions, 47, 48
Feed hold, 310
Feed per minute (G94), 50
Feedrate, 42
Feedrate override, 342
Feedrate override (See Section III–5.2), 251
File heading, 364
File output location, 368
File SEARCH, 364
File search by N–9999, 364
Files, 362
Fine boring cycle (G76), 108
First return to the reference position after the power has been turned on (without an absolute position detector), 61
Forbidden area over- lapping, 352
Format, 369
Free purpose, 514
Function keys, 266
Function keys and soft keys, 265
Function selection soft keys, 285
Functions to simplify programming, 99

[G]

G code for selecting a plane: G17,G18 or G19, 193
G28 in tool length offset mode, 143
G53 specification immediately after power–on, 65
General flow of operation of cnc machine tool, 5
General purpose switches, 520
General screen operations, 265
Guidance programming, 422

[H]

Handle interruption (See Section III–4.6), 250
Handle/incremental feed mode, 299

Heading a program, 401

Helical interpolation (G02,G03), 44

Help function, 529

Help function procedure, 529

Help screen, 281

High–speed peck drilling cycle (G73) , 104

High–speed peck tapping cycle, 137

Hole mahcining, 444

How to indicate command dimensions for moving the tool – absolute, incremental commands, 19

How to use nomograph, 557

How to view the position display change without running the machine, 252

[I]

I/O channel, 512

I/O devices, 385

Inch/metric conversion(G20,G21), 75

Incorrect threaded length, 556

Increment system, 29

Incremental commands, 19

Incremental feed, 299

Incrementing the number of machined parts, 497

Inner side and outer side, 151

Input command from MDI, 184

Input of measured workpiece origin offsets, 516

Input of tool compensation value by programing, 185

Input unit, 512

Inputting a program, 366

Inputting and outputting parameters and pitch error compensation data, 372

Inputting custom macro common variables, 376

Inputting file numbers and program numbers with keys, 384

Inputting multiple programs from an NC tape, 366

Inputting offset data, 370

Inputting parameters, 372

Inputting pitch error compensation data, 374

Inputting/outputting custom macro common variables, 376

Inserting, altering and deleting a word, 398

Inserting a block, 392

Inserting a word, 402

Interface signals, 203

Interference check, 176

Interpolation functions, 34

Interruption of manual operation, 169

Invalid scaling, 189

ISO code, 369

[J]

Jog feed, 297

Jog feed and arrow keys, 520

[K]

Key input and input buffer, 283

[L]

Leader section, 86

Left–handed tapping cycle (G74), 106

Left–handed rigid tapping cycle (G74), 134

Lighting the lamp when the programmed position does not coincide with the reference position, 62

Limitations, 235

Linear acceleration/deceleration after interpolation for cutting feed, 57

Linear interpolation, 39

Linear interpolation (G01), 39

List of functions and tape format, 548

Load meter, 498

Local coordinate system, 70

Local variable levels, 221

[M]

M code, 329

M codes not locked by auxiliary function lock, 341

M, S, T command by only machine lock, 341

M00 (Program stop), 82

M01 (Optional stop), 82

M02,M03 (End of program), 82

M98 (Calling of sub–program), 82

M99 (End of subprogram), 82

Machine coordinate system, 64

Machine lock, 341

Machine lock (See sections III–5.1), 252

Machine lock and auxiliary function lock, 341

Machine lock, mirror image, and scaling, 338

Machining a groove smaller than the tool radius, 181

Machining a step smaller than the tool radius, 182

Machining an inside corner at a radius smaller than the cutter radius, 181

Machining using the end of cutter – tool length compensation function (See II–14.1), 26

Machining using the side of cutter – cutter compensation function (See II–14.2, 14.3), 26

Macro call, 219

Macro call using G code, 226

Macro call using an M code, 227

Macro statements and NC statements , 214
Main program and subprogram, 25, 84
Major addresses and ranges of command values, 92
Manual absolute, 325
Manual absolute ON and OFF, 302
Manual absolute ON/OFF, 338
Manual handle feed, 300
Manual handle interruption, 333
Manual intervention, 325
manual intervention and return, 338
Manual operation, 246, 294
Manual operation after a feed hold, 303
Manual operation after single block stop, 306
Manual operation after the end of block, 303
Manual operation during cornering, 306
Manual operation during cutter compensation, 304
Manual operation performed in other than cornering, 305
Manual pulse generator, 300
Manual reference position return, 62, 295
Manual reference position return (See Section III–3.1), 246
Maximum stroke, 29
MDI mode, 338
MDI operation, 248, 504
MDI operation (Guidance programming MDI operation), 311
Meaning of symbols, 151
Memory operation, 248, 308, 309
Message screen, 281
Method of replacing battery, 537
Methods of setting the two coordinate systems in the same position, 17
Mirror image, 206, 336, 512
Modal information, 207
Modal call (G66), 224
Modal call nesting, 224
Modal information, 504
Modifying the contents of a program, 390
Movement direction of an axis to the rotation of MPG (HNGX), 301
Moving the tool again, 296
Multiple M commands in a single block, 83

[N]

NC statements that have the same property as macro statements, 214
Nesting, 217
Nesting of calls using G codes, 226
Next block display screen, 502
Nomographs, 555
Notes on reading this manual, 7
Number of files registered, 329
Number of machined parts, 206
Number of MPGs, 301
Number of repetitions, 329
Number of tool compensation values and the addresses to be specified, 185

[O]

Offset, 338
Offset cancel mode, 146
offset data input and output, 370
Offset mode, 146
Offset mode cancel, 147
Offset value, 254
Offset/setting screen, 277, 488
Omission of the decimal point , 200
On the memo record, 368
Open command POPEN, 236
Operating monitor display, 498
Operating time, 514
Operation during search, 409
Operation error, 212
Operation method, 529
Operation method,, screen, 531
Operation of portable tape reader, 582
Operational devices, 261
Operators, 216
Optional block skip, 93, 310
Optional stop (M01), 310
Order in which programs are registered, 507
Other method, 509
Output file name, 371, 373, 375, 377
Output format, 371, 373, 375, 377
Outputting a program, 368
Outputting a program after file heading, 368
Outputting custom macro common variable, 377
Outputting offset data, 371
Outputting parameters, 373
Outputting pitch error compensation data, 375
Outputting programs, 383
Overall position display, 495
Overcutting by cutter compensation, 181
Override, 48, 338
Override range, 342
Overrun amount of stored stroke limit, 353
Overtravel, 349
Overtravel during automatic operation, 349
Overtravel during manual operation, 349
Overview of cutter compensation C (G40 – G42), 145

[P]

P–type restart, 324

Parameter list, 523

Parameter setting, 497

Parameter setting value (R_), 241

Parameter table, 529

Parameter table screen, 532

Parameter write, 512

Parameters that require turning off the power, 523

Part count, 497

Part drawing and tool movement, 15

Parts count, 514

Parts count display, run time display, 259

Parts required, 513

Parts total, 513

Password function, 414

Pattern positioning, 448

Peck drilling cycle (G83), 114

Peck rigid tapping cycle (G84 or G74), 136

Peck tapping cycle, 137

Performing tool length offset along two or more axes, 143

Pitch error compensation, 374

Pitch error compensation data, 374

Plane, selection, 72

Plane selection and vector, 149

Pocketing, 466

Portable tape reader, 290

Position display, 334

Position display in the relative coordinate system, 493

Position display in the work coordinate system, 492

Position display screen, 484

Position screen, 268

Positioning (G00), 35

Positioning plane, 101

Positive/negative cutter compensation value and tool center path, 148

Power disconnection, 293

Power ON, 514

Power ON/OFF, 291

Preparatory function (G function), 30

Presetting by setting a coordinate system, 494

Priority of operations, 211

Procedur for manual handle feed, 300

Procedure for alarm history display, 357

Procedure for altering a word, 403

Procedure for automatic insertion of sequence numbers, 478

Procedure for background editing, 413

Procedure for creating a CNC language with the MDI panel, 476

Procedure for creating programs in TEACH IN mode, 480

Procedure for deleting a block, 405

Procedure for deleting a word, 404

Procedure for deleting all programs, 410

Procedure for deleting more than one program by specifying a range, 411

Procedure for deleting multiple blocks, 406

Procedure for deleting one program, 410

Procedure for diagnois, 358

Procedure for displaying and setting custom macro common variables, 518

Procedure for displaying and setting parameters, 522

Procedure for displaying and setting run time, parts count and time , 513

Procedure for displaying and setting the pitch error compensation data, 525

Procedure for displaying and setting the software operator's panel, 519

Procedure for displaying and setting the workpiece origin offset value, 515

Procedure for displaying memory used and a list of programs, 505

Procedure for displaying overall position display screen, 495

Procedure for displaying run time and parts count on the current position display screen, 497

Procedure for displaying the current block display screen, 501

Procedure for displaying the next block display screen, 502

Procedure for displaying the operating monitor, 498

Procedure for displaying the program check screen, 503

Procedure for displaying the program contents, 500

Procedure for displaying the program screen for MDI operation, 504

Procedure for displaying variable values, 202

Procedure for dry run, 344

Procedure for enabling/displaying parameter writing , 523

Procedure for executing, one file, 326

Procedure for executing the scheduling function, 328

Procedure for feedrate override, 342

Procedure for heading a program, 401

Procedure for incremental feed, 299

Procedure for inserting a word, 402

Procedure for inserting, altering and deleting a word, 398

Procedure for jog feed, 297

Procedure for locking and unlocking, 414

Procedure for machine lock and auxiliary function lock, 341

Procedure for manual reference position return, 295

Procedure for program number search, 407

Procedure for replacing batteries for absolute pulse coder, 540, 541

Procedure for replacing battery for memory back–up, 538

Procedure for scanning a program, 399

Procedure for searching a word, 400

Procedure for searching an address, 400

Procedure for sequence number search, 408

Procedure for setting and displaying the cutter compensation value, 509

Procedure for setting the setting data, 511

Procedure for single block, 345

Procedure of turning on the power, 291
Procedure to reset all axes, 494
Procedure to set the axis coordinate to a specified value, 494
Processing macro statements, 232
Program, 24
Program configuration, 23
Program check screen, 503
Program components, 85
Program components other than program sections, 86
Program configuration, 84
Program contents display, 500
Program display , 257, 503
Program editing status, 528
Program end, 94
Program end (M02, M30), 310
program input/output, 366
Program library list, 506
Program menu screen, 417
Program name, 506
Program number, 89
Program number search, 407
Program numbers on a NC tape, 367
Program registration in the background, 367
Program restart, 319
Program screen, 269, 271, 272, 273, 274, 275, 485, 486, 487
Program screen for MDI operation, 504
Program section configuration, 85, 89
Program selection, 249
Program start, 86
Program stop (M00), 309
Programmable parameter entry (G10), 240
Protect switch, 363
Punch code, 512
Punching all programs, 369
Punching programs in the background, 368

[R]

Radius direction error at circle cutting, 563
Range of command value, 553
Range of variable values, 199
Rapid traverse, 49
Rapid traverse override, 343
Rapid traverse prior to reference position return, 298
Re–locking, 415
Reading files, 382
Reference position return check, 60
Reference position, 59
Reference position (Machine–specific position), 15
Reference position return, 60, 325
Reference position return (G28), 61
Reference position return and movement from the reference position, 60
Reference position return and single block, 346
Reference position return check, 60
Reference position return check (G27), 61
Reference position return check in an offset mode, 62
Reference position return completion LED, 296
Reference position return under machine lock, 341
Referencing variables, 200
Registering a position with compensation, 482
Registering a program, 418
Registering a program having the same number as that of a previously registered program, 367
Registering commands other than position commands, 482
Registering custom macro programs, 234
Related manuals, 4
Relation with other functions, 334
Releasing overtravel, 349
Releasing the alarms, 352
Removing the tape, 585
Renaming a program, 395
Repeat, 103
Repetition, 226, 227, 228
Repetition (While statement), 216
Repetitive commands for coordinate system rotation, 197
Replacing batteries for absolute pulse coder, 540
Replacing batteries for absolute pulse coder (Servo amplifier converter unit only for series 20 or α series servo amplifier module), 541
Replacing CNC battery for memory back–up, 538
Request for floppy replacement, 362
Reset, 310, 325
Reset of the alarm, 356
Restart block, 324
Restarting automatic operation, 330
Return from reference position, 60
Return from the reference position (G29), 61
Return operation, 338
Return point level G98/G99, 103
Return to the program screen, 329
Rigid tapping, 131
Rigid tapping (G84), 132
Round function , 210
Rounding up and down to an integer, 211
Run time, 497
Run time and part count settings, 514

[S]

Safety functions, 347

Sample program, 230

Scaling (G50,G51), 186

Scaling and coordinate system rotation, 195

Scaling of circular interpolation, 188

Scaling of each axis, programmable mirror image (negative magnification), 187

Scaling up or down along all axes at the same rate of magnification, 187

Scheduling function, 326

Screen displayed at power–on, 292

Screen fields and their meanings, 381

Screen indicating module setting status, 293

Screen transition chart, 483

Screen transition triggered by the function key, 484, 488, 489

Screen transition triggered by the function key in the MEMORY mode, 485

Screens displayed by function key, 491, 508, 521

Screens displayed by function key, , (In MEMORY mode or MDI mode), 499

Screens displayed by function key, (In the EDIT mode), 505

Screens on which jog feed is valid, 520

Searching in sub–program, 409

Selecting a machine coordinate system (G53), 64

Selecting a workpiece coordinate system, 67

Selection of the offset plane, 146

Selection of tool length offset, 142

Selection of tool used for various machining – tool function, 21

Sequence number and block, 90

Sequence number search, 408

Sequence stop, 512

Setting 0 in parameter PASSWD, 415

Setting a machine coordinate system, 64

Setting a workpiece coordinate system by G92, 66

Setting a workpiece coordinate system, 66

Setting and displaying data, 483

Setting data, 523

Setting parameter PASSWD, 415

Setting parameters with external input/output devices, 523

Setting screens, 490

Setting the relative coordinates, 494

Setting the tape, 584

Side cutting, 453

Significant digits, 385

simple calculation of incorrect thread length, 558

Simple call (G65), 219

Single block, 325, 345, 338

Single block (See Section III–5.5), 252

Single block during a canned cycle, 346

Single direction positioning (G60), 37

Skip function(g31), 45

Soft key transition triggered by the function key, 268, 277, 278, 281, 282

Soft key transition triggered by the function key (When the soft key [BG–EDT] is pressed in all modes), 275

Soft key transition triggered by the function key in the EDIT mode, 271, 487

Soft key transition triggered by the function key in the HNDL, JOG, or REF mode, 274

Soft key transition triggered by the function key in the MDI mode, 272, 273, 486

Soft key transition triggered by the function key in the MEM mode, 269

Soft key transition triggered by the function key in the TJOG or THDL mode, 274

Soft keys, 267

Specifiable range of rapid traverse, 49

Specification of the tool length offset value, 142

Specifying G28 (automatic return to the reference position) in the offset mode, 165

Specifying G29 (automatic return from the reference position) in the offset mode , 165

Specifying no file number, 329

Specifying the sequence number for the return destination in the main program, 97

Specifying the spindle speed value directly (S5–digit command), 78

Specifying the spindle speed with a code, 78

Speedometer, 498

spindle speed function (S function), 77

Start and stop (See Section III–4), 249

Start up, 146

Start up, (Tool compensation start), 146

Starting compensation and cutting along the Z–axis, 182

State in which an auxiliary function is being executed, 527

Status the machine lock being turned on, 61

Status when turning power on, when clear and when reset, 564

Stopping and terminating memory operation, 309

Stopping the punch, 369

Stroke check, 350

Subprogram, 95

Subprogram call, 95

Subprogram call and single block, 346

Subprogram call function, 331

Subprogram call using an M code, 228

Subprogram calls using a T code, 229

Subprogram configuration, 95

Symbols in figures, 103

System screen, 278, 489

System variables, 203

[T]

Tangential speed constant control, 50

Tape code list, 545

Tape end, 88

Tape start, 86

Tapping cycle (G84), 116

Tapping mode, 52

Temporary cutter compensation cancel, 165

Test operation, 340

Testing a program, 251

The center of the arc is identical with the start position or the end position, 161

The distance to return to reference position, 296

The length of the tool center path larger than the circumference of a circle, 175

The length of tool center path larger than the circumference of a circle, 164

The next block to G31 is an absolute command for 1 axis, 46

The next block to G31 is an incremental command, 46

The previous block contains G41 or G42, 174

The tool movement by manual operation, 247

There is no inner intersection, 160

Time information, 204

Time settings, 514

Tool center path with an intersection , 162

Tool center path without an intersection, 163

Tool compensation values, number of compensation values, and entering values from the program (G10), 185

Tool compensation values, 203

Tool figure and tool motion by program, 26

Tool function (T function), 79

Tool length offset (G43,G44,G49), 141

Tool length offset cancel, 143

Tool movement along a straight line, 12

Tool movement along an arc, 12

Tool movement along workpiece parts figure–interpolation, 12

Tool movement around an inside corner (180°≦α), 170

Tool movement around an outside corner at an acute angle (α<90°), 172

Tool movement around an outside corner at an obtuse angle (90°≦α<180°), 171

Tool movement around the inside (α<1°) with an abnormally long vector, linear " linear, 157

Tool movement around the inside of a corner (180°≦α), 156

Tool movement around the outside corner at an acute angle (α<90°), 159

Tool movement around the outside corner at an obtuse angle (90°≦α<180°), 158

Tool movement around the outside linear"linear at an acute angle less than 1 degree (α<1°), 154

Tool movement around the outside linear"linear at an acute angle less than 1 degree (α<1°), 173

Tool movement around the outside of a corner at an obtuse angle (90°≦α<180°), 153

Tool movement around the outside of an acute angle (α<90°), 154

Tool movement by programing – automatic operation, 248

Tool movement in offset mode, 156

Tool movement in offset mode cancel, 170

Tool movement in start–up, 152

Tool movement range – stroke, 27

Tool path at corner, 560

Tool selection function, 80

Travel distance along the drilling axis G90/G91, 102

Travel distance display, 334

Turning on the power, 291

TV check, 369, 512

TV check (Vertical parity check along tape), 90

Types of variables, 199

[U]

Unconditional branch (GOTO statement), 215

Undefined variable, 200

Unit of graph, 498

Using a subprogram only, 98

Using M99 in the main program, 97

[V]

Valid operations, 520

Valid range of tool compensation values, 185

Variable representation, 199

Variables, 199

[W]

Warning for data setting or input/output operation, 528

Warning messages, 284

What is a file, 362

When a movement command in the next block is only one axis, 304

When interference is assumed although actual interference does not occur, 180

When reset after a manual operation following a feed hold, 304

When the next block involves no movement in cutter compensation C (G41, G42) mode, 233

When the next block is not buffered (M codes that are not buffered, G31, etc.), 232

When the next move block is an incremental, 304

When the switch is ON during cutter compensation, 305

Word search, 399

Workpiece coordinate system, 66

Workpiece coordinate system compensation values (workpiece zero point offset values), 209

Writing memo, 363

Revision Record

FANUC Series 20–FA OPERATOR'S MANUAL (B–62174E)

Edition	Date	Contents	Edition	Date	Contents
03	Jan., '95	All pages are revised.			
02	Jan., '94	• Section 5.1.3 for guidance programming in MDI mode was added. • Chapter 9 : Additions and modifications were made to the description of program registration and editing. • Appedix 7 : Program errors were added.			
01	Apr., '93	——————			

Headquarters, Hauptsitz, Siége Européen, Sede Principale, Sede Central:

GE Fanuc Automation Europe S.A.
L-6468 Echternach
Grand Duché de Luxembourg
Tel.: (+352) - 727979-1
Fax : (+352) - 727979-214

Branch Offices, Zweigniederlassungen, Filiales Européennes, Filiali Europee, Filiales Europeas:

GE Fanuc Automation
Deutschland GmbH
Bernhäuser Straße 22
D-73765 Neuhausen a.d.F.
Tel.: (+49)-(0)7158/187 400
Fax: (+49)-(0)7158/187 455

GE Fanuc Automation
Europe S.A.
-Zweigniederlassung Port-
Müllerstrasse 3
CH-2562 Port
Tel.: (+41)-(0)32-3328700
Fax: (+41)-(0)32-3328701

GE Fanuc Automation
Nordic AB
Hammarbacken 4
191 49 Sollentuna
Sweden
Tel.: (+46)-(0)8-4445520
Fax: (+46)-(0)8-4445521

GE Fanuc Automation
France S.A.
45, rue du Bois Chaland
CE 2904-Lisses
F-91029 Evry Cedex
Tel.: (+33)-1-69897039
Fax: (+33)-1-69897049

GE Fanuc Automation
Italia S.r.l.
Piazza Tirana 24/4 B
I-20147 Milano
Tel.: (+39)-(0)2-417176
Fax: (+39)-(0)2-419669

GE Fanuc Automation
Deutschland GmbH
Elberfelder Straße 45
D-40724 Hilden
Tel.: (+49)-(0)2103/87011
Fax: (+49)-(0)2103/87160

GE Fanuc Automation
Europe S.A.
-Netherlands Branch-
Industrieterrein Hoogeind
Minervum 1603A
NL-4817ZL Breda
Tel.: (+31)-(0)76-5783212
Fax: (+31)-(0)76-5870181

GE Fanuc Automation
Deutschland GmbH
Praunheimer Landstraße 50
D-60488 Frankfurt/Main
Tel.: (+49)-(0)69/97607-341
Fax (+49)-(0)69/97681-668

GE Fanuc Automation
(UK) Ltd.
Unit 1, Mill Square
Featherstone Road
Wolverton Mill South
Milton Keynes, MK 125BZ
U.K.
Tel.: (+44)-(0)1908-844010
Fax: (+44)-(0)1908-844001

GE Fanuc Automation
Espana S.A.
Poligono Industrial Olaso
Olaso, 27-Locales 10 & 11
E-20870 Elgoibar (Guipuzcao)
Tel.: (+34)-(9)43-744450
(+34)-(9)43-744420
Fax : (+34)-(9)43-744421

GE Fanuc Automation
Deutschland GmbH
Otto-Schmerbach-Str.20
D-09117 Chemnitz
Tel.: (+49)-(0)371/448111
Fax: (+49)-(0)371/448115

TECHNICAL REPORT (MANUAL)	No. TMN 07/093E

Date : 19, July, 2007

General Manager of
Software Development Laboratory

ソフトウェア開発研究所 19.7.19 所長寒川

FS 0*i*-TA/MA OPERATOR'S MANUAL

FS 0*i*-TB/MB OPERATOR'S MANUAL

FS 0*i*-TC/MC OPERATOR'S MANUAL

FS 20-FA/TA OPERATOR'S MANUAL

FS 20*i*-FA/TA OPERATOR'S MANUAL

FS 20*i*-FB/TB OPERATOR'S MANUAL

Addition of caution sentence of "parameter OLV(No.3202#1)"

1. Communicate this report to:

O	Your information
O	GE Fanuc Americas, GE Fanuc CNC Europe
	FANUC Robotics America, FANUC Robotics Europe
O	Machine tool builder
	Sales agency
	End user

2. Summary for Sales Documents

3. Notice

4. Attached Document:

Export Control:

☐ Controlled (Related item No. of Foreign Exchange Order Attachment List of Japan:______)

■ Non-controlled for item No. 2 to 15 of Foreign Exchange Order Attachment List of Japan

Drawing No.	B-63504EN/01-2, B-63514EN/01-3, B-63834EN/02-2, B-63844EN/02-2, B-64114EN/01-2, B-64124EN/01-2, B-62174E/03-2, B-62204E/02-2, B-63384EN/02-1, B-63374EN/01-2, B-64204EN/01-1, B-64194EN/01-1

COPY : SDLB5
SDLB1

No. SDLB1-07/4018

Date: 11. July, 2007

Acceptance (Technical Administrative Division)			
General Manager	Manager	Section Manager	Person in Charge
研究管理本部 19.7.26 本部長石川	研究管理)技術管理部 19.7.26 部長佐藤 研究管理本部 19.7.26 次長藤田	研究管理)技術管理)管理室 19.7.26 室長深谷	滝 19.7.25 口

Original section of issuer of issue			
Vice General Manager	Department Manager	Vice Department	Section Manager
ソフトウェア開発研究所 19.7.19 次長香西	ソフトウェア開発研)一部 19.7.19 部長遠藤	ソフトウェア開発研)一部 19.7.18 次長弦間	ソフトウェア開発研)一部)開発一課 19.7.18 課長渡邊

柴 19.7.11 田

FS 0*i*-TA/MA OPERATOR'S MANUAL
FS 0*i*-TB/MB OPERATOR'S MANUAL
FS 0*i*-TC/MC OPERATOR'S MANUAL
FS 20-FA/TA OPERATOR'S MANUAL
FS 20*i*-FA/TA OPERATOR'S MANUAL
FS 20*i*-FB/TB OPERATOR'S MANUAL
Addition of caution sentence of "parameter OLV(No.3202#1)"

1.Type of applied technical documents

Name	FS 0*i*-TA OPERATOR'S MANUAL, FS 0*i*-MA OPERATOR'S MANUAL, FS 0*i*-TB OPERATOR'S MANUAL, FS 0*i*-MB OPERATOR'S MANUAL, FS 0*i*-TC OPERATOR'S MANUAL, FS 0*i*-MC OPERATOR'S MANUAL, FS 20-FA OPERATOR'S MANUAL, FS 20-TA OPERATOR'S MANUAL, FS 20*i*-FA OPERATOR'S MANUAL, FS 20*i*-TA OPERATOR'S MANUAL FS 20*i*-FB OPERATOR'S MANUAL, FS 20*i*-TB OPERATOR'S MANUAL
Spec.No./Ed.	B-63504EN/01, B-63514EN/01, B-63834EN/02, B-63844EN/02, B-64114EN/01, B-64124EN/01, B-62174E/03, B-62204E/02, B-63384EN/02, B-63374EN/01, B-64204EN/01, B-64194EN/01

				Title	FS 0*i*-TA/MA/TB/MB/TC/MC, FS 20-FA/TA, FS 20*i*-A/B OPERATOR'S MANUAL Addition of caution sentence of "parameter OLV(No.3202#1)"
				Draw No.	B-63504EN/01-2, B-63514EN/01-3, B-63834EN/02-2, B-63844EN/02-2, B-64114EN/01-2, B-64124EN/01-2, B-62174E/03-2, B-62204E/02-2, B-63384EN/02-1, B-63374EN/01-2, B-64204EN/01-1, B-64194EN/01-1
Ed.	**Date**	**Design**	**Description**	**FANUC LTD**	Sheet 1/3
	Date 2007.07.11	**Desig.**	**Check** **Apprv.**		

2.Summary of Change

Group	Name/Outline	New, Add, Correct, Delete	Applicable Date
Basic			
Optional Function			
Unit			
Maintenance Parts			
Notice	The caution sentence of "parameter OLV(No.3202#1)" is added to the following chapters. ・ FS 0*i*-TA OPERATOR'S MANUAL ・ FS 0*i*-MA OPERATOR'S MANUAL ・ FS 0*i*-TB OPERATOR'S MANUAL ・ FS 0*i*-MB OPERATOR'S MANUAL ・ FS 0*i*-TC OPERATOR'S MANUAL ・ FS 0*i*-MC OPERATOR'S MANUAL ・ FS 20-FA OPERATOR'S MANUAL ・ FS 20-TA OPERATOR'S MANUAL ・ FS 20*i*-FA OPERATOR'S MANUAL ・ FS 20*i*-TA OPERATOR'S MANUAL ・ FS 20*i*-FB OPERATOR'S MANUAL ・ FS 20*i*-TB OPERATOR'S MANUAL III.OPERATION 8.4.2 Outputting a Program	Add	Immediately
Correction			
Another			

In "Outputting a Program", the following CAUTION is added.

Outputting a Program

> **CAUTION**
> When parameter OLV(No.3202#1) is 0, please note that the selected program number for foreground operation is changed by deleting the program or outputting all programs.
> Please confirm the selected program number for foreground operation before the program is executed.

Ed.	Date	Design	Description

Date 2007.07.11 Desig. Check Apprv.

Title	FS 0*i*-TA/MA/TB/MB/TC/MC, FS 20-FA/TA, FS 20*i*-A/B OPERATOR'S MANUAL Addition of caution sentence of "parameter OLV(No.3202#1)"
Draw No.	B-63504EN/01-2, B-63514EN/01-3, B-63834EN/02-2, B-63844EN/02-2, B-64114EN/01-2, B-64124EN/01-2, B-62174E/03-2, B-62204E/02-2, B-63384EN/02-1, B-63374EN/01-2, B-64204EN/01-1, B-64194EN/01-1

FANUC LTD