FANUC Series 16*i*-LB
FANUC Series 160*i*-LB

OPERATOR'S MANUAL

B-63664EN/02

The export of this product is subject to the authorization of the government of the country from where the product is exported.

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

SAFETY PRECAUTIONS

This section describes the safety precautions related to the use of CNC units. It is essential that these precautions be observed by users to ensure the safe operation of machines equipped with a CNC unit (all descriptions in this section assume this configuration). Note that some precautions are related only to specific functions, and thus may not be applicable to certain CNC units.
Users must also observe the safety precautions related to the machine, as described in the relevant manual supplied by the machine tool builder. Before attempting to operate the machine or create a program to control the operation of the machine, the operator must become fully familiar with the contents of this manual and relevant manual supplied by the machine tool builder.

Contents

1 DEFINITION OF WARNING, CAUTION, AND NOTE

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

⚠ WARNING

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

⚠ CAUTION

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

NOTE

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

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

2 GENERAL WARNINGS AND CAUTIONS

WARNING

1. Never attempt to machine a workpiece without first checking the operation of the machine. Before starting a production run, ensure that the machine is operating correctly by performing a trial run using, for example, the single block, feedrate override, or machine lock function or by operating the machine with neither a tool nor workpiece mounted. Failure to confirm the correct operation of the machine may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

2. Before operating the machine, thoroughly check the entered data.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

3. Ensure that the specified feedrate is appropriate for the intended operation. Generally, for each machine, there is a maximum allowable feedrate. The appropriate feedrate varies with the intended operation. Refer to the manual provided with the machine to determine the maximum allowable feedrate. If a machine is run at other than the correct speed, it may behave unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

4. When using a tool compensation function, thoroughly check the direction and amount of compensation.
Operating the machine with incorrectly specified data may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

5. The parameters for the CNC and PMC are factory–set. Usually, there is not need to change them. When, however, there is not alternative other than to change a parameter, ensure that you fully understand the function of the parameter before making any change.
Failure to set a parameter correctly may result in the machine behaving unexpectedly, possibly causing damage to the workpiece and/or machine itself, or injury to the user.

6. Some laser beams are invisible (cannot be seen by the eye).
Before approaching the nozzle, check that the laser beam is off. Failure to do so subjects the user to the danger of serious injury.

7. A workpiece that has just undergone laser beam processing is very hot. Touching it, therefore, may cause injury to the user.

⚠ CAUTION

1. Immediately after switching on the power, do not touch any of the keys on the MDI panel until the position display or alarm screen appears on the CNC unit.
Some of the keys on the MDI panel are dedicated to maintenance or other special operations. Pressing any of these keys may place the CNC unit in other than its normal state. Starting the machine in this state may cause it to behave unexpectedly.

2. The operator’s manual and programming manual supplied with a CNC unit provide an overall description of the machine’s functions, including any optional functions. Note that the optional functions will vary from one machine model to another. Therefore, some functions described in the manuals may not actually be available for a particular model. Check the specification of the machine if in doubt.

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

NOTE

Programs, parameters, and macro variables are stored in nonvolatile memory in the CNC unit. Usually, they are retained even if the power is turned off. Such data may be deleted inadvertently, however, or it may prove necessary to delete all data from nonvolatile memory as part of error recovery.
To guard against the occurrence of the above, and assure quick restoration of deleted data, backup all vital data, and keep the backup copy in a safe place.

3 WARNINGS AND CAUTIONS RELATED TO PROGRAMMING

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

⚠ WARNING

1. Coordinate system setting

If a coordinate system is established incorrectly, the machine may behave unexpectedly as a result of the program issuing an otherwise valid move command.
Such an unexpected operation may damage the tool, the machine itself, the workpiece, or cause injury to the user.

2. Positioning by nonlinear interpolation

When performing positioning by nonlinear interpolation (positioning by nonlinear movement between the start and end points), the tool path must be carefully confirmed before performing programming.
Positioning involves rapid traverse. If the tool collides with the workpiece, it may damage the tool, the machine itself, the workpiece, or cause injury to the user.

3. Function involving a rotation axis

When programming polar coordinate interpolation or normal–direction (perpendicular) control, pay careful attention to the speed of the rotation axis. Incorrect programming may result in the rotation axis speed becoming excessively high, such that centrifugal force causes the chuck to lose its grip on the workpiece if the latter is not mounted securely.
Such mishap is likely to damage the tool, the machine itself, the workpiece, or cause injury to the user.

4. Inch/metric conversion

Switching between inch and metric inputs does not convert the measurement units of data such as the workpiece origin offset, parameter, and current position. Before starting the machine, therefore, determine which measurement units are being used. Attempting to perform an operation with invalid data specified may damage the tool, the machine itself, the workpiece, or cause injury to the user.

5. Stroke check

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

⚠ CAUTION

1. Absolute/incremental mode

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

2. Plane selection

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

3. Torque limit skip

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

4. Programmable mirror image

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

5. Compensation function

If a command based on the machine coordinate system or a reference position return command is issued in compensation function mode, compensation is temporarily canceled, resulting in the unexpected behavior of the machine.
Before issuing any of the above commands, therefore, always cancel compensation function mode.

6. Gap control

Tracing control is performed to keep the gap constant. During tracing control, ensure that the nozzle does not move to any location other than around the workpiece. Otherwise, the nozzle may collide with the machine, possibly causing damage to the nozzle and the machine.

4 WARNINGS AND CAUTIONS RELATED TO HANDLING

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

⚠ WARNING

1. Manual operation

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

2. Manual reference position return

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

3. Manual numeric command

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

4. Manual handle feed

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

5. Origin/preset operation

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

⚠ WARNING

6. Workpiece coordinate system shift

Manual intervention, machine lock, or mirror imaging may shift the workpiece coordinate system. Before attempting to operate the machine under the control of a program, confirm the coordinate system carefully.
If the machine is operated under the control of a program without making allowances for any shift in the workpiece coordinate system, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the operator.

7. Software operator's panel and menu switches

Using the software operator's panel and menu switches, in combination with the MDI panel, it is possible to specify operations not supported by the machine operator's panel, such as mode change, override value change, and jog feed commands.
Note, however, that if the MDI panel keys are operated inadvertently, the machine may behave unexpectedly, possibly damaging the tool, the machine itself, the workpiece, or causing injury to the user.

⚠ CAUTION

1. Manual intervention

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

2. Feed hold, override, and single block

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

3. Dry run

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

4. Cutter compensation in MDI mode

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

5. Program editing

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

5 WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. **Memory backup battery replacement**

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high–voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high–voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The CNC uses batteries to preserve the contents of its memory, because it must retain data such as programs, offsets, and parameters even while external power is not applied.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the contents of the CNC's memory will be lost.
Refer to the maintenance section of the operator's manual or programming manual for details of the battery replacement procedure.

⚠ WARNING

2. Absolute pulse coder battery replacement

When replacing the memory backup batteries, keep the power to the machine (CNC) turned on, and apply an emergency stop to the machine. Because this work is performed with the power on and the cabinet open, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing the batteries, be careful not to touch the high–voltage circuits (marked ⚠ and fitted with an insulating cover).
Touching the uncovered high–voltage circuits presents an extremely dangerous electric shock hazard.

NOTE

The absolute pulse coder uses batteries to preserve its absolute position.
If the battery voltage drops, a low battery voltage alarm is displayed on the machine operator's panel or screen.
When a low battery voltage alarm is displayed, replace the batteries within a week. Otherwise, the absolute position data held by the pulse coder will be lost.
Refer to the maintenance section of the operator's manual or programming manual for details of the battery replacement procedure.

WARNING

3. Fuse replacement

For some units, the chapter covering daily maintenance in the operator's manual or programming manual describes the fuse replacement procedure.
Before replacing a blown fuse, however, it is necessary to locate and remove the cause of the blown fuse.
For this reason, only those personnel who have received approved safety and maintenance training may perform this work.
When replacing a fuse with the cabinet open, be careful not to touch the high–voltage circuits (marked and fitted with an insulating cover).
Touching an uncovered high–voltage circuit presents an extremely dangerous electric shock hazard.

Table of Contents

III. OPERATION

IV. MAINTENANCE

APPENDIX

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.

IV. MAINTENANCE
Describes procedures for replacing batteries.

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

Some functions described in this manual may not be applied to some products. For detail, refer to the DESCRIPTIONS manual(B–63662EN).

This manual does not describe parameters in detail. For details on parameters mentioned in this manual, refer to the manual for parameters (B–63530EN, B–63670EN).

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

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

Product name	Abbreviations	
FANUC Series 16*i*–LB	16*i*–LB	Series 16*i*
FANUC Series 160*i*–LB	160*i*–LB	Series 160*i*

Special symbols

This manual uses the following symbols:

- IP_ : 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.

Manuals related to FANUC Series 16*i*/160*i*–LB

Table 1 (a) Manuals Related to the Series 16*i*/160*i*–LA

Manual name	Specification number	
FANUC Series 16*i*/18*i*/160*i*/180*i*–MODEL B DESCRIPTIONS	B–63522EN	
FANUC Series 16*i*/18*i*/160*i*/180*i*–MODEL B CONNECTION MANUAL (HARDWARE)	B–63523EN	
FANUC Series 16*i*/18*i*/160*i*/180*i*–MODEL B CONNECTION MANUAL (FUNCTION)	B–63523EN–1	
FANUC Series 16*i*/18*i*/160*i*/180*i*–MODEL B PARAMETER MANUAL	B–63530EN	
FANUC Series 16*i*/160*i*–LB DESCRIPTIONS	B–63662EN	
FANUC Series 16*i*/160*i*–LB CONNECTION MANUAL	B–63663EN	
FANUC Series 16*i*/160*i*–LB OPERATOR’S MANUAL	B–63664EN	*
FANUC Series 16*i*/160*i*–LB MAINTENANCE MANUAL	B–63665EN	
FANUC Series 16*i*/160*i*–LB PARAMETER MANUAL	B–63670EN	
Programming		
Macro Compiler/Macro Executor PROGRAMMING MANUAL	B–61803E–1	
C Language Executor PROGRAMMING MANUAL	B–62443EN–3	
FAPT MACRO COMPILER (For Personal Computer) PROGRAMMING MANUAL	B–66102E	
PMC		
PMC Ladder Language PROGRAMMING MANUAL	B–61863E	
PMC C Language PROGRAMMING MANUAL	B–61863E–1	
Network		
PROFIBUS–DP Board OPERATOR’S MANUAL	B–62924EN	
Ethernet Board/DATA SERVER Board OPERATOR’S MANUAL	B–63354EN	
DeviceNet Board OPERATOR’S MANUAL	B–63404EN	

Manuals related to FANUC SERVO MOTOR α series

Table 1 (a) Manuals Related to the SERVO MOTOR α series

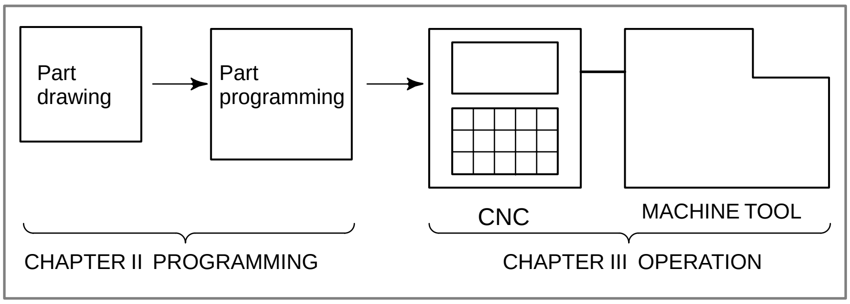
Manual name	Specification number
FANUC AC SERVO MOTOR α series DESCRIPTIONS	B–65142E
FANUC AC SERVO MOTOR α series PARAMETER MANUAL	B–65150E
FANUC SERVO AMPLIFIER α series DESCRIPTIONS	B–65162E
FANUC SERVO MOTOR α series MAINTENANCE MANUAL	B–65165E

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.

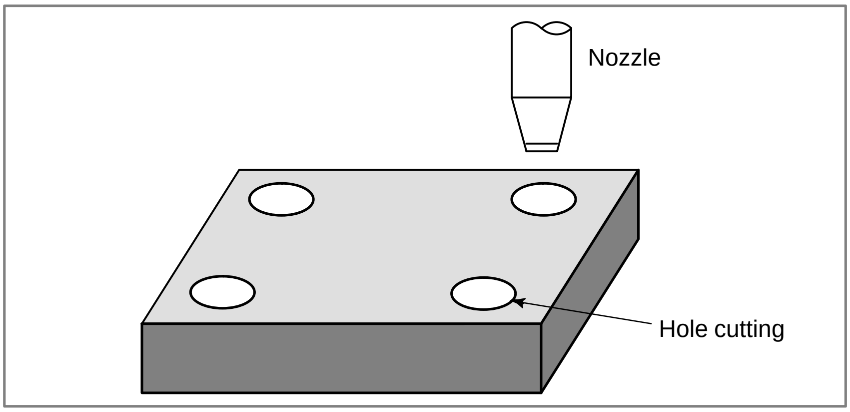
1) First, prepare the program from a part drawing to operate the CNC machine tool.
   How to prepare the program is described in the Chapter II. PROGRAMMING.

2) The program is to be read into the CNC system. Then, mount the workpieces and tools on the machine, and operate the nozzle 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 conditions

Cutting procedure \ Cutting process	1	2
	Peripheral machining	**Hole machining**
1. Cutting nozzle		
2. Cutting conditions : Feedrate, beam output		
3. Nozzle path		



Prepare the program of the nozzle path and cutting condition according to the workpiece figure, for each cutting.

1.2 NOTES ON READING THIS MANUAL

CAUTION

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.
3 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.

1.3 NOTES ON VARIOUS KINDS OF DATA

CAUTION

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.

II. PROGRAMMING

1 GENERAL

1.1 NOZZLE MOVEMENT ALONG WORKPIECE PARTS FIGURE– INTERPOLATION

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

Explanations

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

- **Nozzle movement along a straight line**

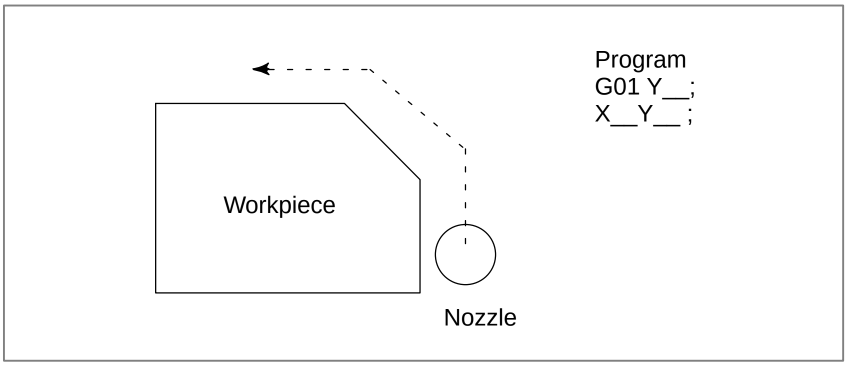


Fig.1.1 (a) Nozzle movement along a straight line

- **Nozzle movement along an arc**

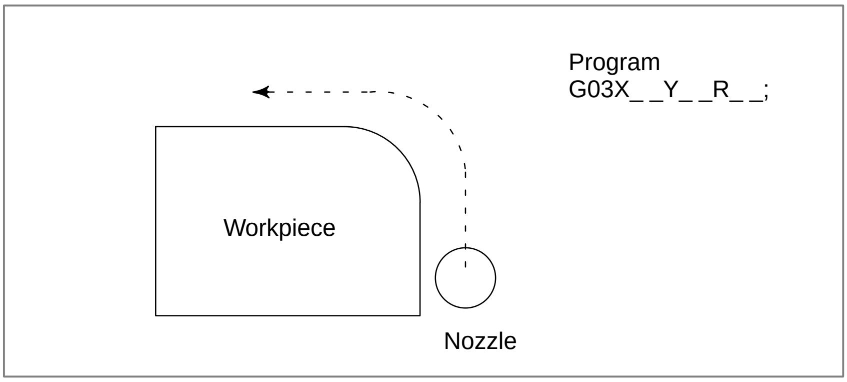


Fig.1.1 (b) Nozzle movement along an arc

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

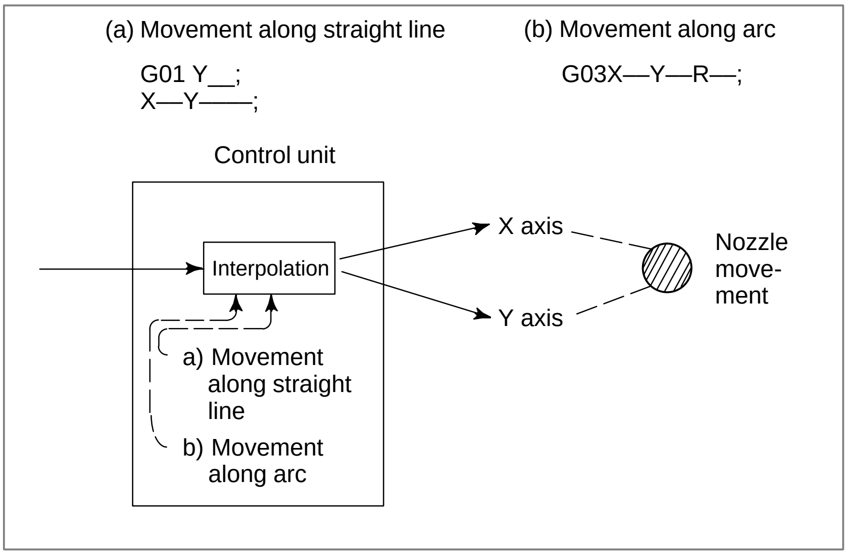


Fig.1.1 (c) Interpolation function

CAUTION

Some machines move tables instead of tools but this manual assumes that nozzle is moved against workpieces.

1.2 FEED–FEED FUNCTION

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

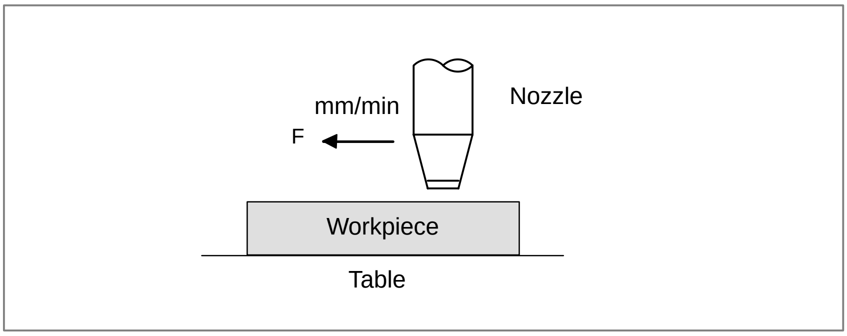


Fig.1.2 (a) Feed function

Feedrates can be specified by using actual numerics. For example, to feed the nozzle 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 NOZZLE MOVEMENT

1.3.1 Reference Position (Machine–Specific Position)

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

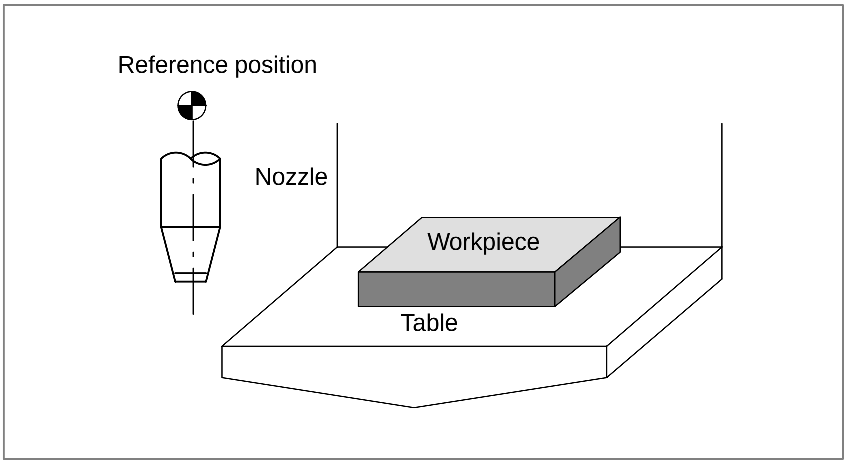


Fig.1.3.1 Reference position

Explanations

The nozzle 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 nozzle to the reference position for workpiece 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

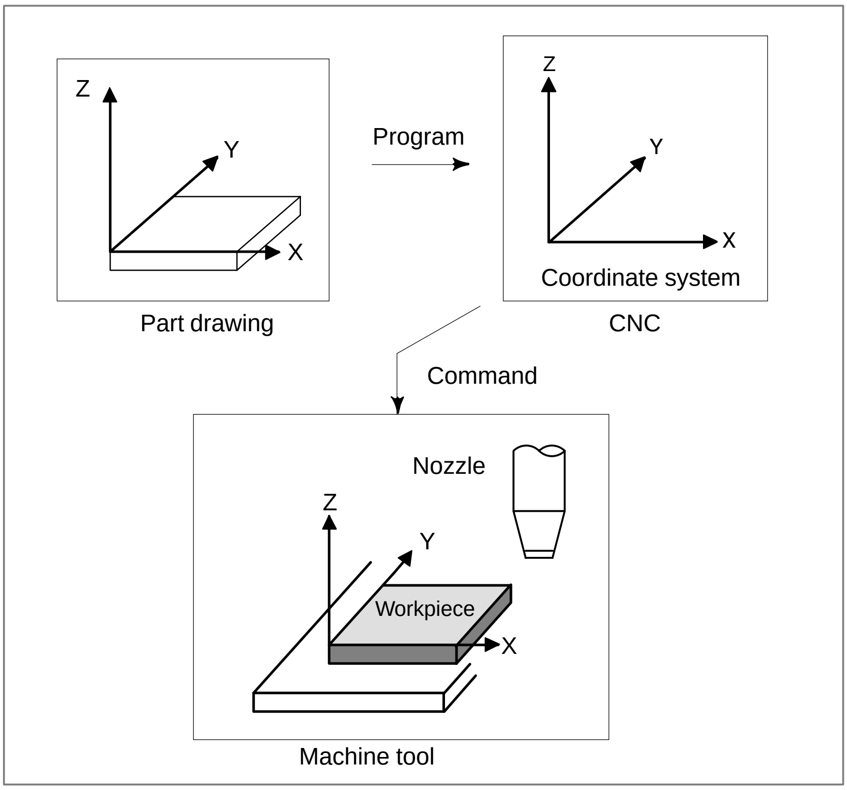


Fig.1.3.2 (a) Coordinate system

Explanations

- **Coordinate system**

The following two coordinate systems are specified at different locations: (See II–8)

(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 nozzle to the zero point of the coordinate system to be set.

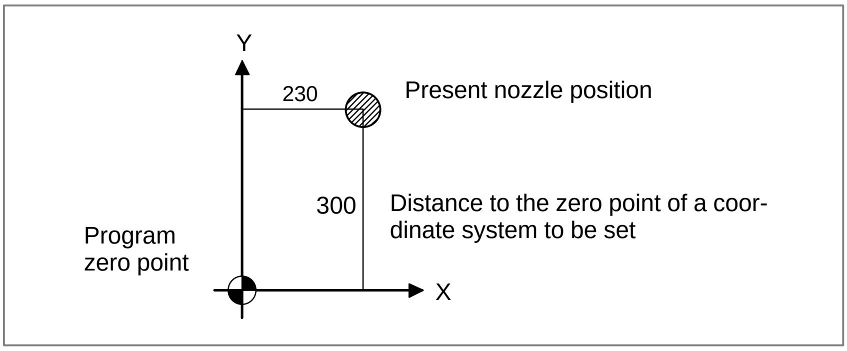


Fig.1.3.2 (b) Coordinate system specified by the CNC

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

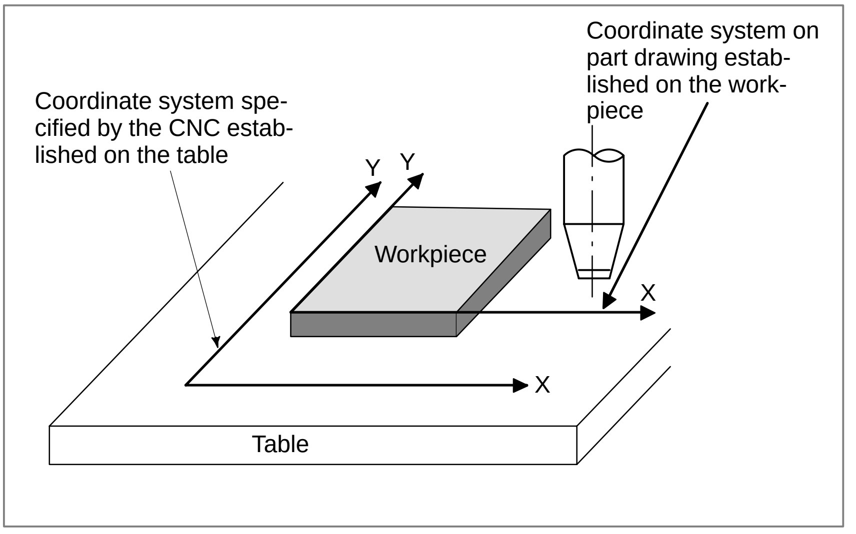


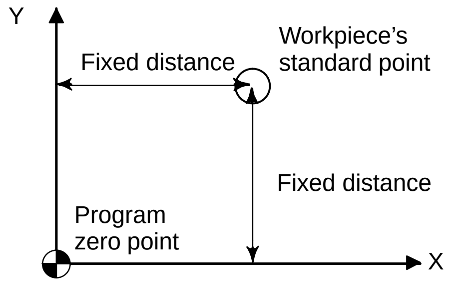
Fig.1.3.2 (c) Coordinate system specified by CNC and coordinate system on part drawing

The nozzle 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**

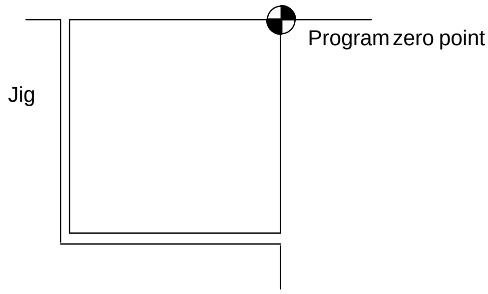
To set the two coordinate systems at the same position, simple methods shall be used according to workpiece shape, the number of machinings.

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



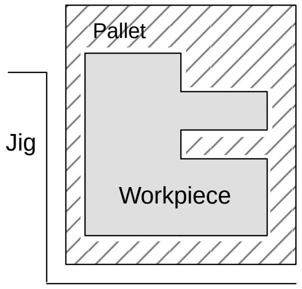
Bring the nozzle center to the workpiece standard point.
And set the coordinate system specified by CNC at this position.

(2) Mounting a workpiece directly against the jig



Meet the nozzle 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)).

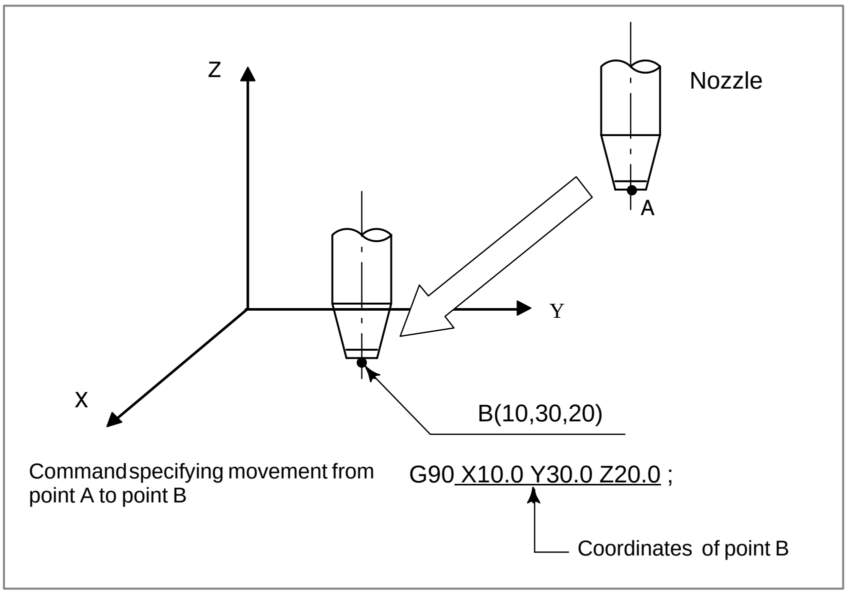
1.3.3 How to Indicate Command Dimensions for Moving the Machine Absolute, Incremental Commands

Explanations

Coordinate values of command for moving the tool can be indicated by absolute or incremental designation (See II–9.1).

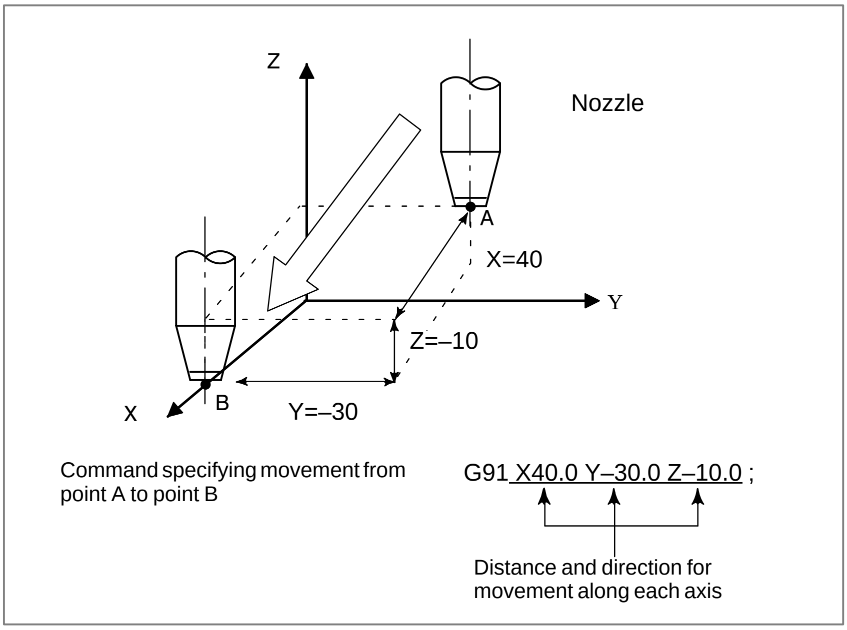
- **Absolute coordinates**

The nozzle moves to a point at ”the distance from zero point of the coordinate system” that is to the position of the coordinate values.



- **Incremental coordinates**

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



1.4 COMMAND FOR MACHINE OPERATIONS–MISCELLANEOUS FUNCTION

When laser arting is actually started, it is necessary to operate a work shooter, and tip conveyer. For this purpose, on–off operations of work shooter and tip conveyer should be controlled (See II–9).

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.

1.5 PROGRAM CONFIGURATION

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

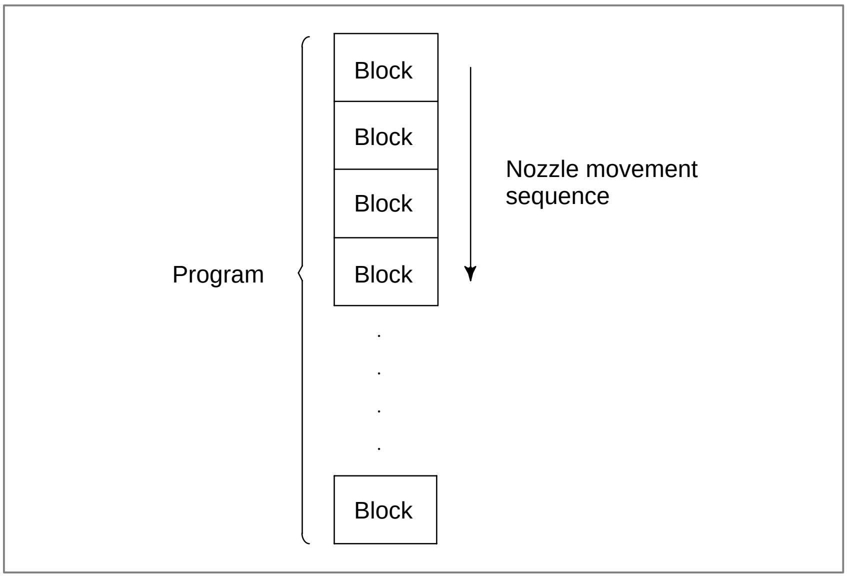


Fig.1.5 (a) Program configuration

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–10).

Explanations

The block and the program have the following configurations.

- **Block**

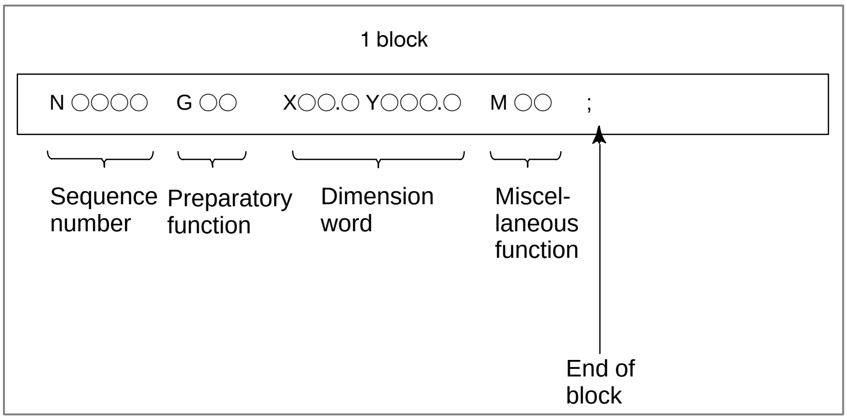


Fig.1.5 (b) Block configuration

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**

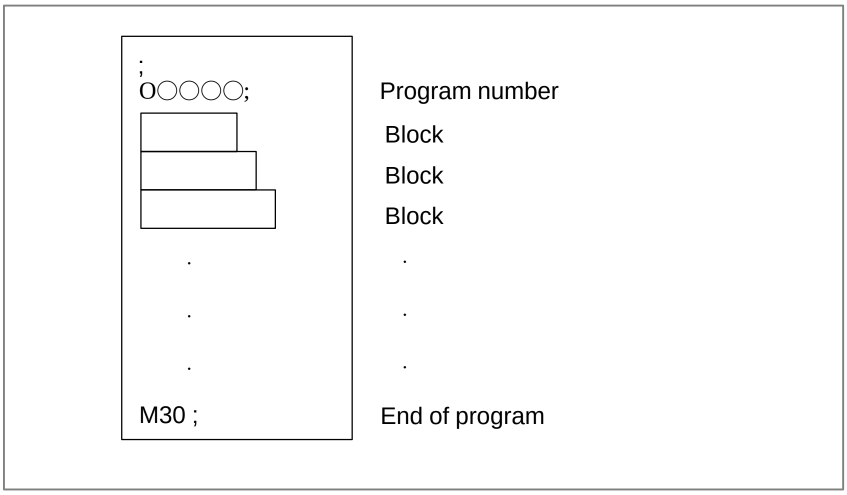
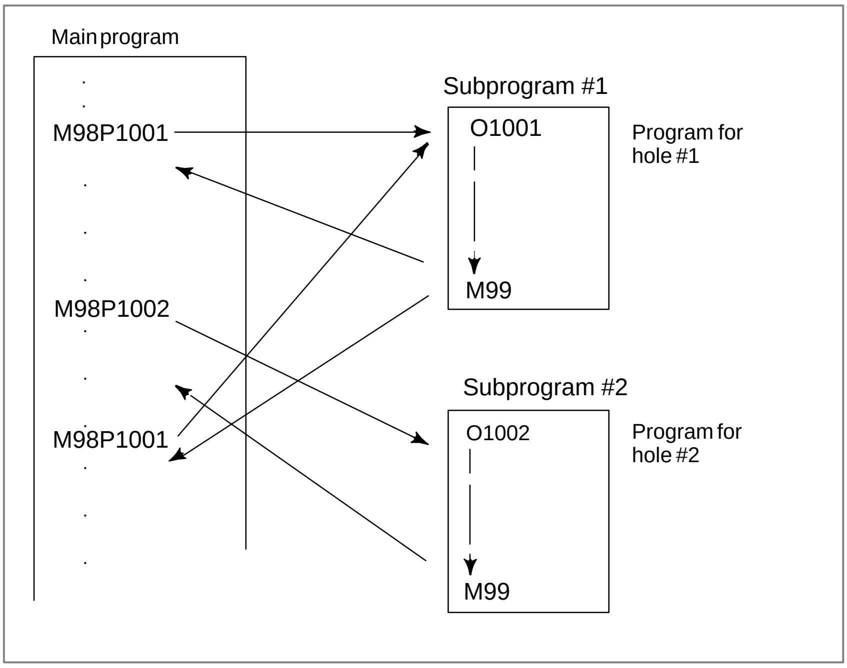


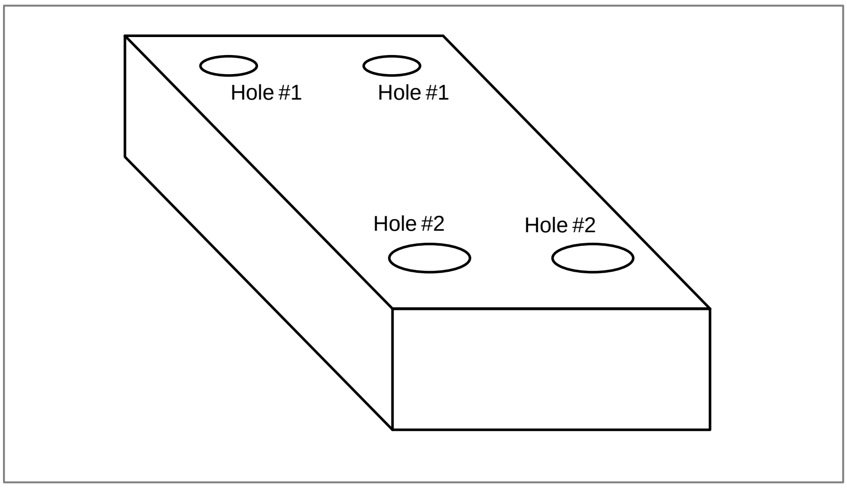
Fig.1.5 (c) Program configuration

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

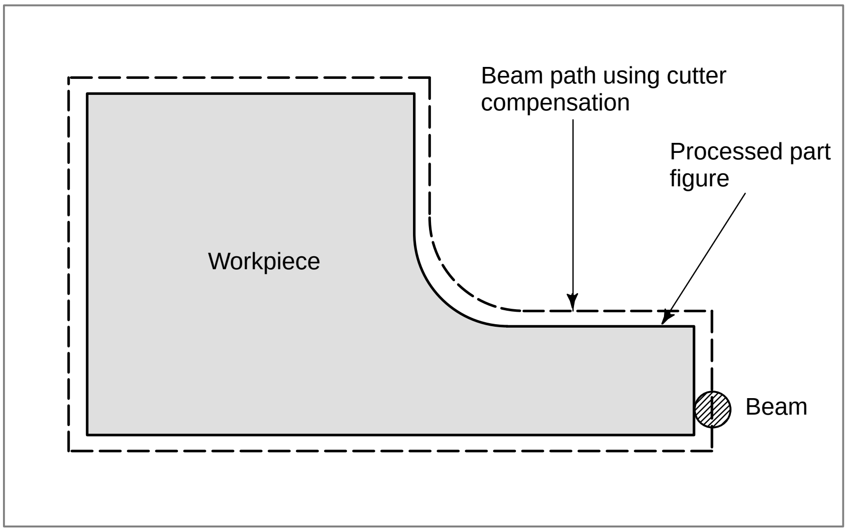
When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands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





- **Machining using the side of cutter – Cutter compensation function (See II–12.2, 12.3)**

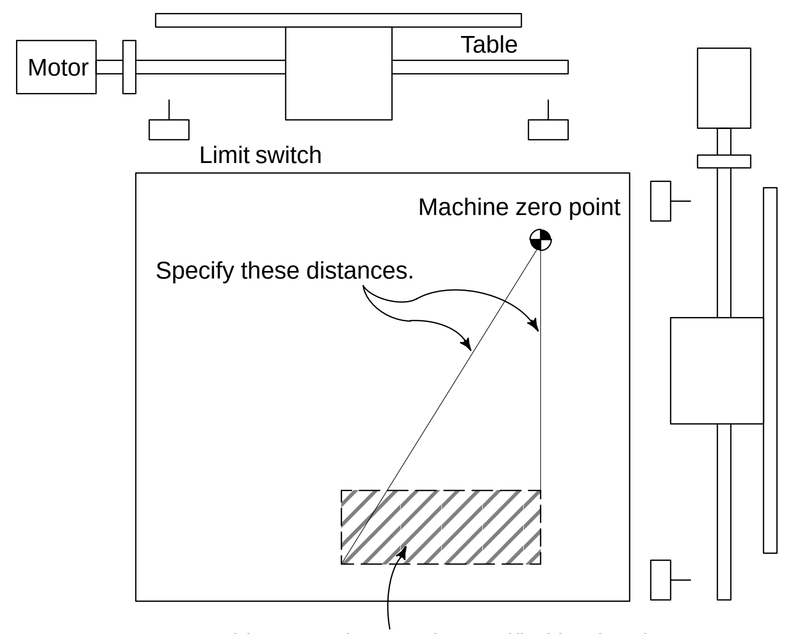
Because laserbeam has a radius, the center of the beam path goes around the workpiece with the cutter radius deviated.



If radius of beam 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 (See II–12.2 and 12.3).

1.6 NOZZLE 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.



Besides strokes defined with limit switches, the operator can define an area which the nozzle 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

Series 16*i*, Series 160*i*

Item	16*i*–LB 160*i*–LB
No. of basic controlled axes	3 axes
Controlled axes expansion (total)	Max. 8 axes
Basic simultaneously controlled axes	2 axes
Simultaneously controlled axes expansion (total)	Max. 6 axes

NOTE
The number of simultaneously controllable axes for manual operation jog feed, manual reference position 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 names of three basic axes are always X, Y, and Z. The name of an additional axis can be set to A, B, C, U, V, or W by using parameter 1020. Parameter No. 1020 is used to determine the name of each axis.
When this parameter is set to 0 or a character other than the valid characters is specified, an axis name from 1 to 8 is assigned by default.

Limitations

- **Default axis name**

When a default axis name (1 to 8) is used, operation in the MEM mode, MDI mode or RMT mode is disabled.

- **Duplicate axis names**

If a duplicate axis name is specified in the parameter, operation is enabled only for the axis specified first.

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 is 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, inches, or deg.
The increment system is classified into IS–B.

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

The least command increment is either metric or inch depending on the machine tool. Set metric or inch to the parameter INM (No.100#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

Maximum stroke = Least command increment × 99999999
See 2.3 Incremen System.

Table 2.4 Maximum strokes

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

NOTE
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) Which status G22 or G23 at power on is set by parameter G23 (No. 3402#7). However, G22 and G23 remain unchanged when the clear state is set at 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 (parameterG18) and bit 2 (parameter 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, P/S 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. G codes are indicated by group.
6. The group of G60 is switched according to the setting of the MDL bit (bit 0 of parameter 5431). (When the MDL bit is set to 0, the 00 group is selected. When the MDL bit is set to 1, the 01 group is selected.)

Table 3 G Code List (1/3)

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	
G05		High–speed remote buffer/High–precision contour control	
G05.1		AI contour control / AI nano contour control	
G05.4		HRV3 on off	
G07		Hypothetical axis interpolation	
G07.1 (G107)		Cylindrical interpolation	
G08		Advanced preview control	
G09		Exact stop	
G10		Programmable data input	
G11		Programmable data input mode cancel	
G12	01	Spatial circular interpolation	
G12.1	25	Polar coordinate interpolation mode	
G13.1		Polar coordinate interpolation mode cancel	
G13	31	Gap control mode	
G14		Gap control mode cancel	
G15	17	Polar coordinates command cancel	
G16		Polar coordinates command	
G17	02	XpYp plane	Xp: X axis or a parallel axis
G18		ZpXp plane	Yp: Y axis or a parallel axis
G19		YpZp plane	Zp: Z axis or a parallel axis
G20	06	Inch input	
G21		Metric input	
G22	04	Stored stroke check function on	
G23		Stored stroke check function off	
G24	00	Piercing command	
G27		Reference position return check	
G28		Automatic return to reference position	
G29		Automatic return from reference position	
G30		2nd, 3rd and 4th reference position return	
G30.1		Floating reference point return	
G31		Skip function	
G32		Assist gas command	
G33	34	Spatial corner R insertion	
G34		Spatial corner R insertion cancel	
G39	00	Corner circular interpolation	

Table 3 G Code List (2/3)

Code	Group	Function
G40	07	Cutter compensation cancel
G41		Cutter compensation left
G42		Cutter compensation right
G40.1 (G150)	18	Normal direction control cancel mode
G41.1 (G151)		Normal direction control left on
G42.1 (G152)		Normal direction control right on
G45	00	Tool offset increase
G46		Tool offset decrease
G47		Tool offset double increase
G48		Tool offset double decrease
G50	11	Scaling cancel
G51		Scaling
G50.1	22	Programmable mirror image cancel
G51.1		Programmable mirror image
G52	00	Local coordinate system setting
G53		Machine coordinate system selection
G54	14	Workpiece coordinate system 1 selection
G54.1		Additional workpiece coordinate system 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	00/01	Unidirectional positioning
G61	15	Exact stop mode
G62		Automatic corner override
G63	00	Laser power control
G64	15	Machining mode
G65	00	Macro call
G66	12	Macro modal call
G67		Macro modal call cancel
G68	16	Coordinate rotation
G69		Coordinate rotation cancel
G68	32	Three–dimensional coordinate conversion
G69		Three–dimensional coordinate conversion cancel
G71	00	A–axis length compensation
G72.1		Rotation copy
G72.2		Linear copy
G84	16	Coordinate rotation
G85		Coordinate rotation cancel

Table 3 G Code List (3/3)

Code	Group	Function
G90	03	Absolute command
G91		Increment command
G92	00	Setting for workpiece coordinate system
G92.1		Workpiece coordinate system preset
G98	33	Three–dimensional conversion
G99		Three–dimensional conversion cancel

CAUTION

The G codes for coordinate rotation are G68, G69, G84, and G85.
If, however, the three–dimensional coordinate conversion option is added, G68 and G69 are the G codes for three–dimensional coordinate conversion.

4 INTERPOLATION FUNCTIONS

4.1 POSITIONING (G00)

In the absolute command, coordinate value of the end point is programmed.
In the incremental command the distance the nozzle moves is programmed.

Format

G00 IP_;

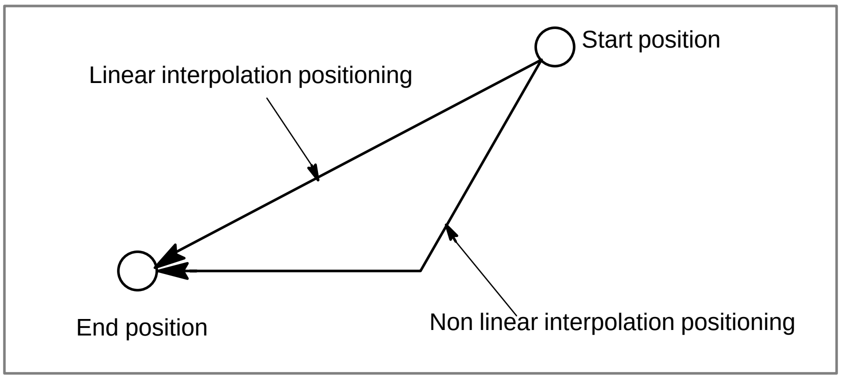
IP: For an absolute command, the coordinates of an end position, and for an incremental commnad, the distance the nozzle moves.

Explanations

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

- **Nonlinear interpolation positioning**
  The nozzle is positioned with the rapid traverse rate for each axis separately. The nozzle path is not normally straight.

- **Linear interpolation positioning**
  The nozzle moves along a line to the specified position. The nozzle is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis. However, the nozzle path is not the same as in linear interpolation (G01).



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).
In–position check for each block can be disabled by setting bit 5 (NCI) of parameter No.1601 accordingly.

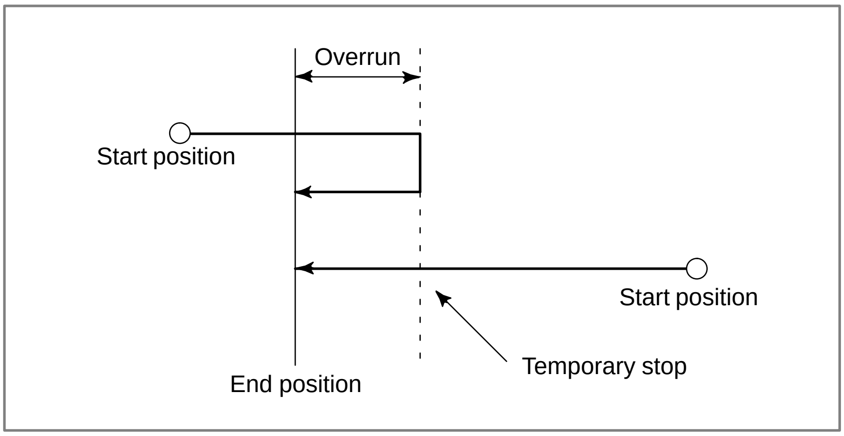
Limitations

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

G60IP_;

IP_ : For an absolute command, the coordinates of an end position, and for an incremental commnad, the distance the nozzle 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 (No. 5431 bit 0 MDL).
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

- 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.

4.3 LINEAR INTERPOLATION (G01)

Nozzle 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 nozzle moves.

F_:Speed of nozzle 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 nozzle path. If the F code is not commanded, the feedrate is regarded as zero.

The feedrate of each axis direction is as follows.

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

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 : $F\gamma = \frac{\gamma}{L} \times f$

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

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

The feed rate of the rotary axis is commanded in the unit of deg/min (the unit is decimal point position).

When the straight line axis α(such as X, Y, or Z) and the rotating axis β (such as A, B, or C) are linearly interpolated, the feed rate is that in which the tangential feed rate in the α and β cartesian coordinate system is commanded by F(mm/min).

β–axis feedrate is obtained ; at first, the time required for distribution is calculated by using the above fromula, then the β –axis feedrate unit is changed to deg 1min.

A calculation example is as follows.
G91 G01 X20.0B40.0 F300.0 ;
This changes the unit of the C axis from 40.0 deg to 40mm with metric input. The time required for distribution is calculated as follows:

$$\frac{\sqrt{20^2 + 40^2}}{300} \fallingdotseq 0.14907 \ (\text{min})$$

The feed rate for the C axis is

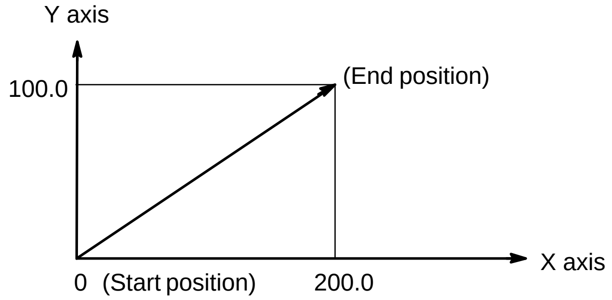
$$\frac{40}{0.14907} \fallingdotseq 268.3 \ \text{deg/min}$$

In simultaneous 3 axes control, the feed rate is calculated the same way as in 2 axes control.

Examples

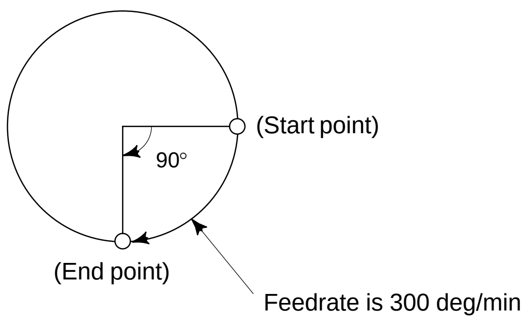
- **Linear interpolation**

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



- **Feedrate for the rotation axis**

G91G01C–90.0 G300.0 ;Feed rate of 300deg/min



4.4 CIRCULAR INTERPOLATION (G02, G03)

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

Format

Arc in the XpYp plane

$$G17 \left\{\begin{matrix}G02\\G03\end{matrix}\right\} Xp\_Yp\_ \left\{\begin{matrix}I\_\ J\_\\R\_\end{matrix}\right\} F\_\ ;$$

Arc in the ZpXp plane

$$G18 \left\{\begin{matrix}G02\\G03\end{matrix}\right\} Xp\_\ p\_ \left\{\begin{matrix}I\_\ K\_\\R\_\end{matrix}\right\} F\_$$

Arc in the YpZp plane

$$G19 \left\{\begin{matrix}G02\\G03\end{matrix}\right\} Yp\_\ Zp\_ \left\{\begin{matrix}J\_\ K\_\\R\_\end{matrix}\right\} F\_$$

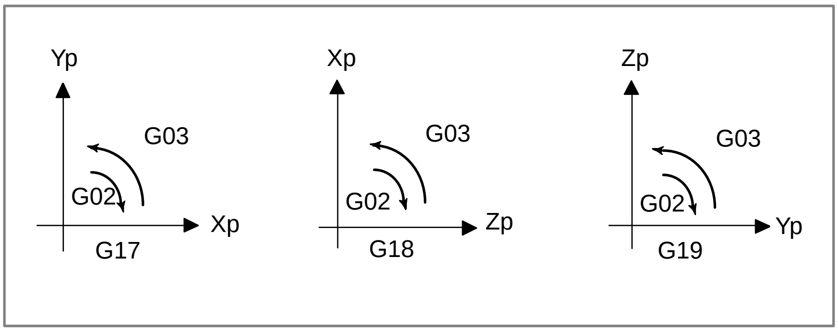
Table 4.4 Description of the Command Format

Command	Description
G17	Specification of arc on XpYp plane
G18	Specification of arc on ZpXp plane
G19	Specification of arc on YpZp plane
G02	Circular Interpolation Clockwise direction (CW)
G03	Circular Interpolation Counterclockwise direction (CCW)
X_{p}_	Command values of X axis or its parallel axis (set by parameter No. 1022)
Y_{p}_	Command values of Y axis or its parallel axis (set by parameter No. 1022)
Z_{p}_	Command values of Z axis or its parallel axis (set by parameter No. 1022)
I_	X_p axis distance from the start point to the center of an arc with sign
J_	Y_p axis distance from the start point to the center of an arc with sign
K_	Z_p 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 X_pY_p plane (Z_pX_p plane or Y_pZ_p plane) are defined when the X_pY_p plane is viewed in the positive–to–negative direction of the Z_p axis (Y_p axis or X_p axis, respectively) in the Cartesian coordinate system. See the figure below.

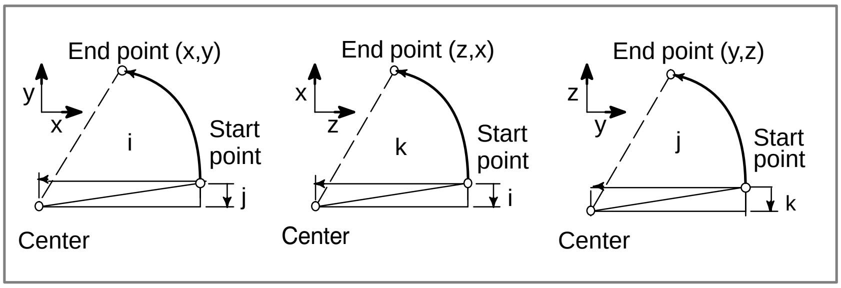


- **Distance moved on an arc**

The end point of an arc is specified by address Xp, Yp or Zp, 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 Xp, Yp, and Zp 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.

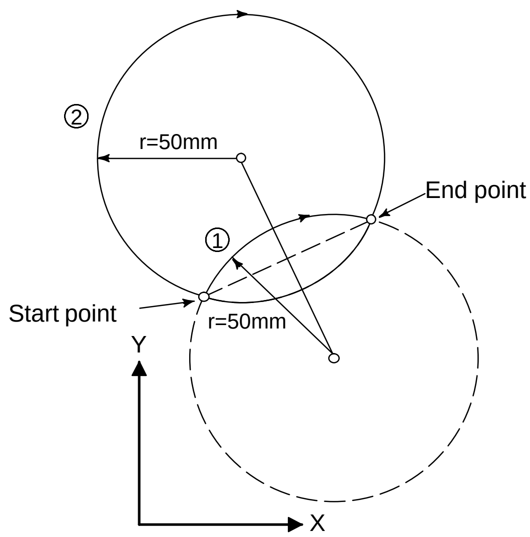


I0,J0, and K0 can be omitted. When X_p, Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.
G021; Command for a circle
If the difference between the radius at the start point and that at the end point exceeds the permitted value in a parameter (No.3410), an P/S alarm (No.020) 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 Xp, Yp, and Zp 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 <u>R50.0</u> F300.0 ;
For arc (2)**(greater than 180°)**
G91 G02 X_P60.0 Y_P20.0 <u>R–50.0</u> F300.0 ;



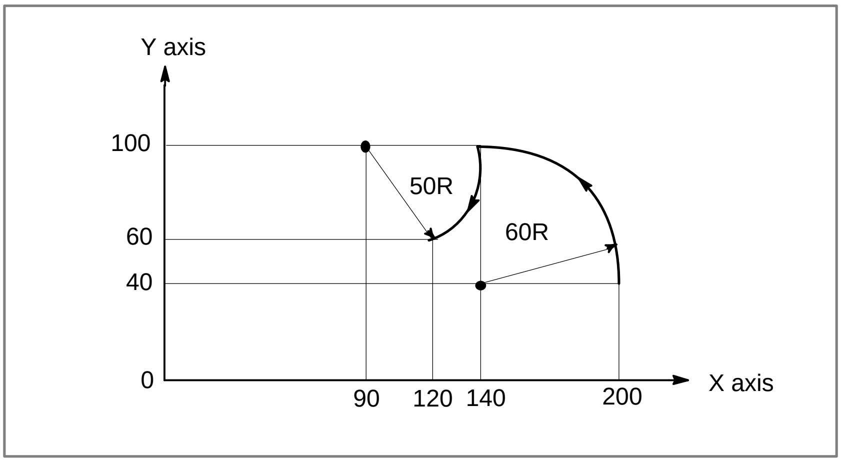
- **Feedrate**

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 nozzle 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 axis U is specified as a parallel axis to X axis when plane XY is specified, an P/S alarm (No.028)is displayed.
When an arc having a center angle approaching 180° is specified, the calculated center coordinates may contain an error. In such a case, specify the center of 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)

Helical interpolation which moved helically is enabled by specifying up to two other axes which move synchronously with the circular interpolation by circular commands.

Format

Synchronously with arc of XpYp plane

$$\mathbf{G17} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \mathbf{Xp\_Yp\_} \begin{Bmatrix} \mathbf{I\_J\_} \\ \mathbf{R\_} \end{Bmatrix} \alpha\_(\beta\_)\mathbf{F\_;}$$

Synchronously with arc of ZpXp plane

$$\mathbf{G18} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \mathbf{Xp\_Zp\_} \begin{Bmatrix} \mathbf{I\_K\_} \\ \mathbf{R\_} \end{Bmatrix} \alpha\_(\beta\_)\mathbf{F\_;}$$

Synchronously with arc of YpZp plane

$$\mathbf{G19} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \mathbf{Yp\_Zp\_} \begin{Bmatrix} \mathbf{J\_K\_} \\ \mathbf{R\_} \end{Bmatrix} \alpha\_(\beta\_)\mathbf{F\_;}$$

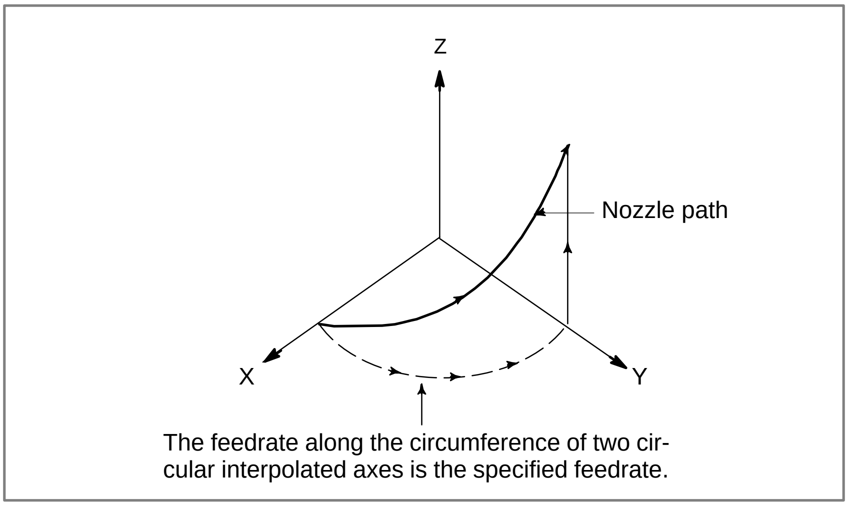
α,β: Any one axis where circular interpolation is not applied.
Up to two other axes can be specified.

Explanations

The command method is to simply or secondary 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.



Restrictions

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

4.6 HELICAL INTERPOLATION B (G02, G03)

Helical interpolation B moves the nozzle in a helical manner. This interpolation can be executed by specifying the circular interpolation command together with up to four additional axes in simple high–precision contour control mode (see II–NO TAG).

Format

With an arc in the XpYp plane

$$\mathbf{G17}\begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Xp\_Yp\_}\ \begin{Bmatrix}\mathbf{I\_J\_}\\ \mathbf{R\_}\end{Bmatrix}\ \alpha\_\beta\_\gamma\_\delta\_\mathbf{F\_;}$$

With an arc in the ZpXp plane

$$\mathbf{G18}\begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Xp\_Zp\_}\ \begin{Bmatrix}\mathbf{I\_K\_}\\ \mathbf{R\_}\end{Bmatrix}\ \alpha\_\beta\_\gamma\_\delta\_\mathbf{F\_;}$$

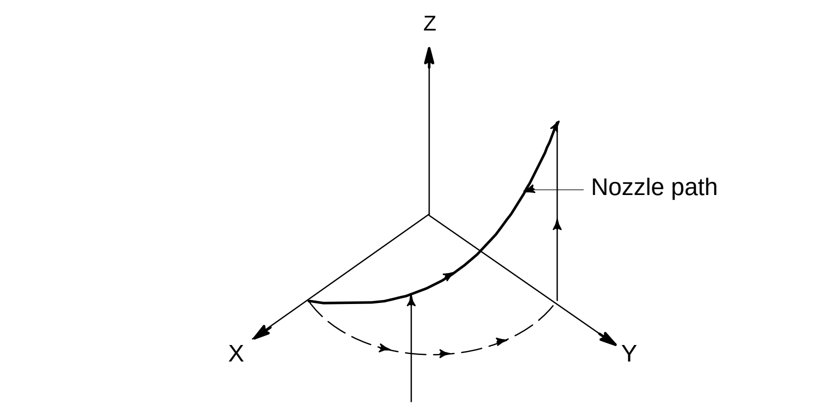
With an arc in the YpZp plane

$$\mathbf{G19}\begin{Bmatrix}\mathbf{G02}\\ \mathbf{G03}\end{Bmatrix}\ \mathbf{Yp\_Zp\_}\ \begin{Bmatrix}\mathbf{J\_K\_}\\ \mathbf{R\_}\end{Bmatrix}\ \alpha\_\beta\_\gamma\_\delta\_\mathbf{F\_;}$$

$\alpha, \beta, \gamma, \delta$: Any axis to which circular interpolation is not applied. Up to four axes can be specified.

Explanations

Basically, the command can be specified by adding two movement axes to a standard helical interpolation command (see II–4.5). Address F should be followed by a tangential velocity, which has been determined by also taking movement along the linear axes into consideration.



The feedrate equals the tangential velocity determined by also taking movement along the linear axes into consideration.

Limitations

- The command of helical interpolation B can be specified only in AI contour control mode.
- Cutter compensation is applied only to an arc.
- In a block containing the helical interpolation command, the tool offset command or tool length compensation command cannot be specified.

4.7 POLAR COORDINATE INTERPOLATION (G12.1, G13.1)

Polar coordinate interpolation is a function that exercises contour control in converting a command programmed in a Cartesian coordinate system to the movement of a linear axis (movement of a tool) and the movement of a rotary axis (rotation of a workpiece).

Format

G12.1 ;	Starts polar coordinate interpolation mode (enables polar coordinate interpolation)
	Specify linear or circular interpolation using coordinates in a Cartesian coordinate system consisting of a linear axis and rotary axis (virtual axis).
G13.1 ;	Polar coordinate interpolation mode is cancelled (for not performing polar coordinate interpolation)

Specify G12.1 and G13.1 in Separate Blocks.

Explanations

- **Polar coordinate interpolation plane**

G12.1 starts the polar coordinate interpolation mode and selects a polar coordinate interpolation plane (Fig. 4.7). Polar coordinate interpolation is performed on this plane.

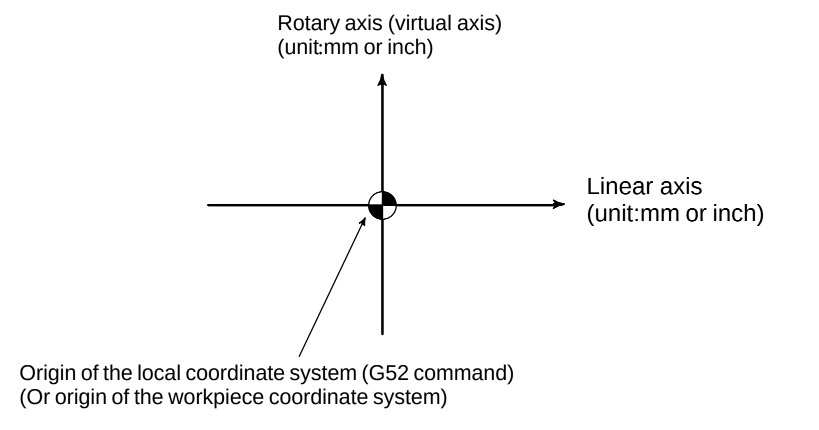


Fig.4.7 Polar coordinate interpolation plane.

When the power is turned on or the system is reset, polar coordinate interpolation is canceled (G13.1).
The linear and rotation axes for polar coordinate interpolation must be set in parameters (No. 5460 and 5461) beforehand.

> **CAUTION**
> The plane used before G12.1 is specified (plane selected by G17, G18, or G19) is canceled. It is restored when G13.1 (canceling polar coordinate interpolation) is specified.
> When the system is reset, polar coordinate interpolation is canceled and the plane specified by G17, G18, or G19 is used.

- **Distance moved and feedrate for polar coordinate interpolation**

  The unit for coordinates on the hypothetical axis is the same as the unit for the linear axis (mm/inch)

  The unit for the feedrate is mm/min or inch/min

In the polar coordinate interpolation mode, program commands are specified with Cartesian coordinates on the polar coordinate interpolation plane. The axis address for the rotation axis is used as the axis address for the second axis (virtual axis) in the plane. Whether a diameter or radius is specified for the first axis in the plane is the same as for the rotation axis regardless of the specification for the first axis in the plane. The virtual axis is at coordinate 0 immediately after G12.1 is specified. Polar interpolation is started assuming the angle of 0 for the position of the tool when G12.1 is specified.
Specify the feedrate as a speed (relative speed between the workpiece and tool) tangential to the polar coordinate interpolation plane (Cartesian coordinate system) using F.

- **G codes which can be specified in the polar coordinate interpolation mode**

G01 . . . . Linear interpolation
G02, G03 . . . . Circular interpolation
G04 . . . . Dwell, Exact stop
G40, G41, G42 . . . Cutter compensation
(Polar coordinate interpolation is applied to the path after cutter compensation.)
G65, G66, G67 . . . Custom macro command
G90, G91 . . . . Absolute command, incremental command

- **Circular interpolation in the polar coordinate plane**

The addresses for specifying the radius of an arc for circular interpolation (G02 or G03) in the polar coordinate interpolation plane depend on the first axis in the plane (linear axis).

· I and J in the Xp–Yp plane when the linear axis is the X–axis or an axis parallel to the X–axis.
· J and K in the Yp–Zp plane when the linear axis is the Y–axis or an axis parallel to the Y–axis.
· K and I in the Zp–Xp plane when the linear axis is the Z–axis or an axis parallel to the Z–axis.

The radius of an arc can be specified also with an R command.

- **Movement along axes not in the polar coordinate interpolation plane in the polar coordinate interpolation mode**

The tool moves along such axes normally, independent of polar coordinate interpolation.

- **Current position display in the polar coordinate interpolation mode**

Actual coordinates are displayed. However, the remaining distance to move in a block is displayed based on the coordinates in the polar coordinate interpolation plane (Cartesian coordinates).

Limitations

- **Coordinate system for the polar coordinate interpolation**

Before G12.1 is specified, a local coordinate system (or workpiece coordinate system) where the center of the rotary axis is the origin of the coordinate system must be set. In the G12.1 mode, the coordinate system must not be changed (G92, G52, G53, relative coordinate reset, G54 through G59, etc.).

- **Tool offset command**

The polar coordinate interpolation mode cannot be started or terminated (G12.1 or G13.1) in the tool offset mode (G41 or G42). G12.1 or G13.1 must be specified in the tool offset canceled mode (G40).

- **Tool offset command**

A tool offset must be specified before the G12.1 mode is set. No offset can be changed in the G12.1 mode.

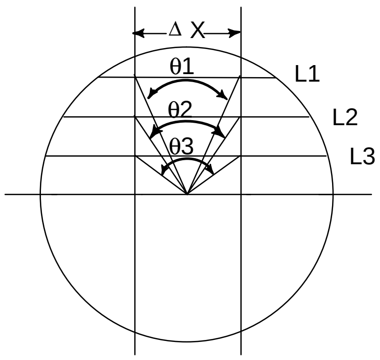
- **Program restart**

For a block in the G12.1 mode, the program cannot be restarted.

- **Cutting feedrate for the rotation axis**

Polar coordinate interpolation converts the tool movement for a figure programmed in a Cartesian coordinate system to the tool movement in the rotation axis (C–axis) and the linear axis (X–axis). When the tool moves closer to the center of the workpiece, the C–axis component of the feedrate becomes larger and may exceed the maximum cutting feedrate for the C–axis (set in parameter (No. 1422)), causing an alarm (see the figure below). To prevent the C–axis component from exceeding the maximum cutting feedrate for the C–axis, reduce the feedrate specified with address F or create a program so that the beam (center of the beam when cutter compensation is applied) does not move close to the center of the workpiece.

WARNING



Consider lines L1, L2, and L3. ΔX is the distance the tool moves per time unit at the feedrate specified with address F in the Cartesian coordinate system. As the tool moves from L1 to L2 to L3, the angle at which the tool moves per time unit corresponding to ΔX in the Cartesian coordinate system increases from θ1 to θ 2 to θ3.
In other words, the C–axis component of the feedrate becomes larger as the tool moves closer to the center of the workpiece. The C component of the feedrate may exceed the maximum cutting feedrate for the C–axis because the tool movement in the Cartesian coordinate system has been converted to the tool movement for the C–axis and the X–axis.

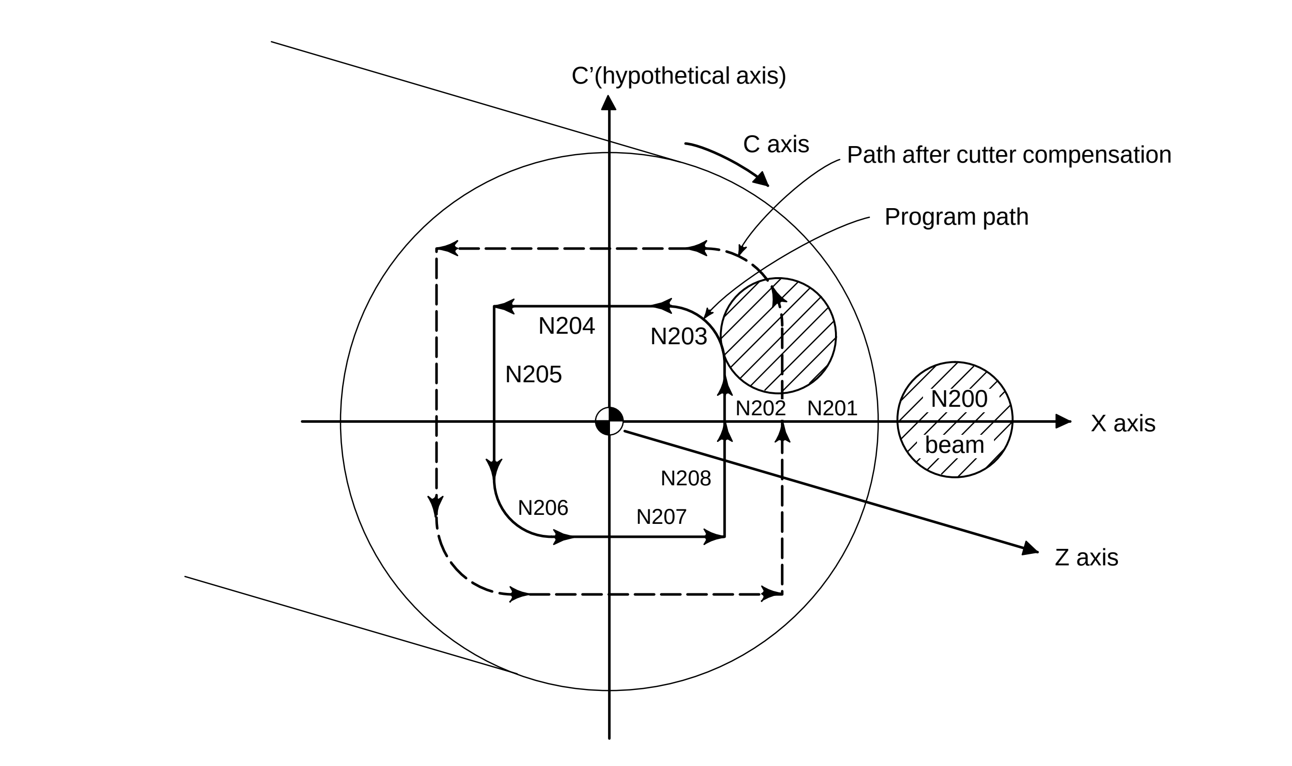
L : Distance (in mm) between the beam center and workpiece center when the beam center is the nearest to the workpiece center
R : Maximum cutting feedrate (deg/min) of the C axis
Then, a speed specifiable with address F in polar coordinate interpolation can be given by the formula below. Specify a speed allowed by the formula. The formula provides a theoretical value; in practice, a value slightly smaller than a theoretical value may need to be used due to a calculation error.

$$F < L \times R \times \frac{\pi}{180} \text{ (mm/min)}$$

The function cannot be used with a three–dimensional machine.

Examples

<u>Example of Polar Coordinate Interpolation Program Based on X Axis (Linear Axis) and C Axis (Rotary Axis)</u>




```
O0001 ;
  :
N010 T0101
  :
N0100 G90 G00 X60.0 C0 Z_ ;       Positioning to start position
N0200 G12.1 ;                      Start of polar coordinate interpolation
N0201 G42 G01 X20.0 F_ ;
N0202 C10.0 ;
N0203 G03 X10.0 C20.0 R10.0 ;
N0204 G01 X–20.0 ;                   Geometry program
N0205 C–10.0 ;                       (program based on cartesian coordinates on
N0206 G03 X–10.0 C–20.0 I10.0 J0 ;    X–C' plane)
N0207 G01 X20.0 ;
N0208 C0 ;
N0209 G40 X60.0 ;
N0210 G13.1 ;                      Cancellation of polar coordinate interpolation
N0300 Z_ ;
N0400 X_ C_ ;
  :
N0900M30 ;
```

4.8 CYLINDRICAL INTERPOLATION (G07.1)

The amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis along the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to the amount of travel of the rotary axis.

Format

G07.1IP r ; **Starts the cylindrical interpolation mode (enables cylindrical interpolation).**

:

G07.1IP 0 ; **The cylindrical interpolation mode is cancelled.**

IP: An address for the rotation axis
r : The radius of the cylinder

Specify G07.1 IP r ; and G07.1 IP 0; in separate blocks.
G107 can be used instead of G07.1.

Explanations

• Plane selection (G17, G18, G19)

Use parameter (No. 1022) to specify whether the rotation axis is the X–, Y–, or Z–axis, or an axis parallel to one of these axes. Specify the G code to select a plane for which the rotation axis is the specified linear axis.
For example, when the rotation axis is an axis parallel to the X–axis, G17 must specify an Xp–Yp plane, which is a plane defined by the rotation axis and the Y–axis or an axis parallel to the Y–axis.
Only one rotation axis can be set for cylindrical interpolation.

• Feedrate

A feedrate specified in the cylindrical interpolation mode is a speed on the developed cylindrical surface.

• Circular interpolation (G02, G03)

In the cylindrical interpolation mode, circular interpolation is possible with the rotation axis and another linear axis. Radius R is used in commands in the same way as described in II–4.4.
The unit for a radius is not degrees but millimeters (for metric input) or inches (for inch input).

< Example Circular interpolation between the Z axis and C axis >

For the C axis of parameter (No.1022), 5 (axis parallel with the X axis) is to be set. In this case, the command for circular interpolation is

G18 Z__C__;
G02 (G03) Z__C__R__;

For the C axis of parameter (No.1022), 6 (axis parallel with the Y axis) may be specified instead. In this case, however, the command for circular interpolation is

G19 C__Z__;
G02 (G03) Z__C__R__;

• Tool offset

To perform tool offset in the cylindrical interpolation mode, cancel any ongoing cutter compensation mode before entering the cylindrical interpolation mode. Then, start and terminate tool offset within the cylindrical interpolation mode.

- **Cylindrical interpolation accuracy**

In the cylindrical interpolation mode, the amount of travel of a rotary axis specified by an angle is once internally converted to a distance of a linear axis on the outer surface so that linear interpolation or circular interpolation can be performed with another axis. After interpolation, such a distance is converted back to an angle. For this conversion, the amount of travel is rounded to a least input increment.
So when the radius of a cylinder is small, the actual amount of travel can differ from a specified amount of travel. Note, however, that such an error is not accumulative.
If manual operation is performed in the cylindrical interpolation mode with manual absolute on, an error can occur for the reason described above.

$$\text{The actual amount of travel} = \left[\frac{\text{MOTION REV}}{2 \times 2\pi R} \left[\times \text{Specified value} \times \frac{2 \times 2\pi R}{\text{MOTION REV}} \right] \right]$$

MOTION REV : The amount of travel per rotation of the rotation axis (Setting value of parameter No. 1260)

R : Workpiece radius

$[\;\;]$:Rounded to the least input increment

Limitations

- **Arc radius specification in the cylindrical interpolation mode**

In the cylindrical interpolation mode, an arc radius cannot be specified with word address I, J, or K.

- **Circular interpolation and cutter compensation**

If the cylindrical interpolation mode is started when cutter compensation is already applied, circular interpolation is not correctly performed in the cylindrical interpolation mode.

- **Positioning**

In the cylindrical interpolation mode, positioning operations (including those that produce rapid traverse cycles such as G28, G53, G73, G74, G76, G80 through G89) cannot be specified. Before positioning can be specified, the cylindrical interpolation mode must be cancelled. Cylindrical interpolation (G07.1) cannot be performed in the positioning mode (G00).

- **Coordinate system setting**

In the cylindrical interpolation mode, a workpiece coordinate system (G92, G54 through G59) or local coordinate system (G52) cannot be specified.

- **Cylindrical interpolation mode setting**

In the cylindrical interpolation mode, the cylindrical interpolation mode cannot be reset. The cylindrical interpolation mode must be cancelled before the cylindrical interpolation mode can be reset.

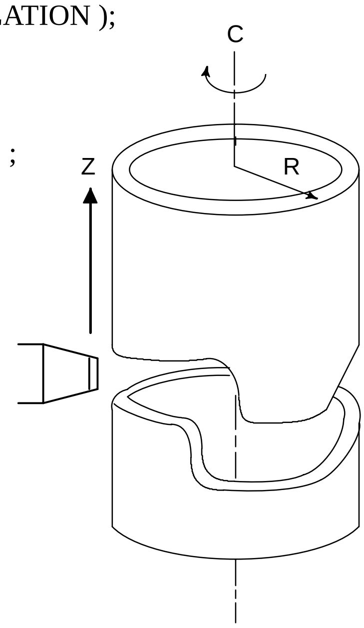
- **Tool offset**

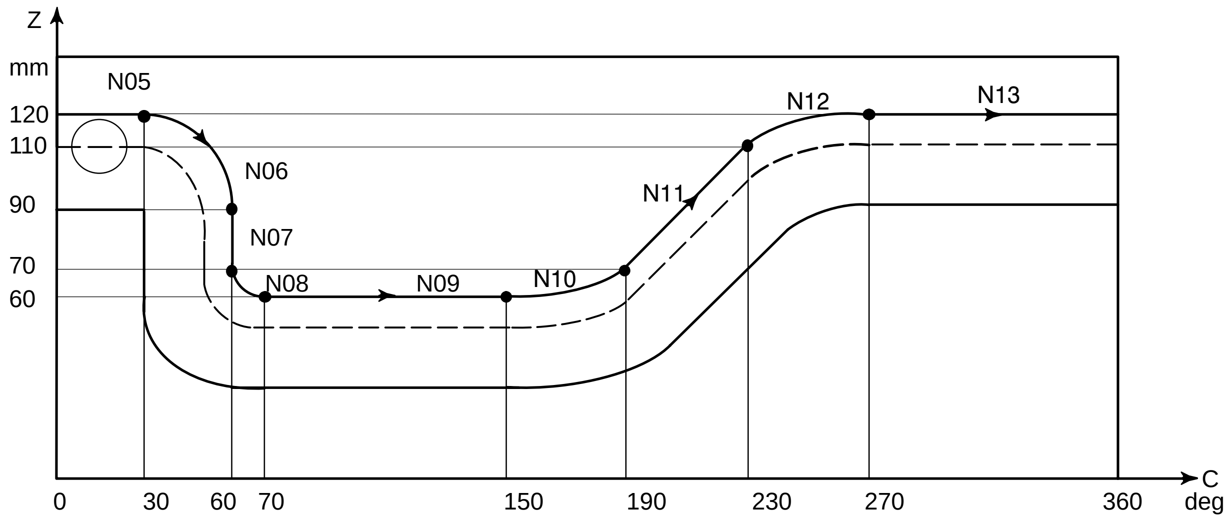
A tool offset must be specified before the cylindrical interpolation mode is set. No offset can be changed in the cylindrical interpolation mode.

Examples

Example of a Cylindrical Interpolation Program

```
O0001 (CYLINDRICAL INTERPOLATION );
N01 G00 G90 Z100.0 C0 ;
N02 G01 G91 G18 Z0 C0 ;
N03 G07.1 C57299 ;
N04 G90 G01 G42 Z120.0 D01 F250 ;
N05 C30.0 ;
N06 G03 Z90.0 C60.0 R30.0 ;
N07 G01 Z70.0 ;
N08 G02 Z60.0 C70.0 R10.0 ;
N09 G01 C150.0 ;
N10 G02 Z70.0 C190.0 R75.0 ;
N11 G01 Z110.0 C230.0 ;
N12 G03 Z120.0 C270.0 R75.0 ;
N13 G01 C360.0 ;
N14 G40 Z100.0 ;
N15 G07.1 C0 ;
N16 M30 ;
```





4.9 HYPOTHETICAL AXIS INTERPOLATION (G07)

One of the X–, Y–, and X–axes can be specified as a hypothetical axis. Although the axis specified as a hypothetical axis does not allow operation by any subsequent program commands, interpolation is performed internally. If, therefore, one circular interpolation axis is specified as a hypothetical axis, circular interpolation is performed on the remaining one axis only, allowing the feedrate to be changed in a sine wave fashion.
When circular interpolation is performed on the Y– and Z–axes with the Z–axis specified as a hypothetical axis, it is possible to machine the corners of a rectangular pipe placed in parallel with the X–axis by applying gap control to the Z–axis and rotating the rectangular pipe about a helical axis.

Format

G07 α0 ; Sets a hypothetical axis.
G07 α1 ; Cancels a hypothetical axis.
where, α is any of the addresses on a controlled axis.

Explanations

- **Operation**

When hypothetical axis interpolation mode is set by executing the G07 0 command, the axis specified as a hypothetical axis does not allow operation by any program instructions other than gap control commands. Thus, the machine, absolute, and relative coordinates are not updated.In hypothetical axis interpolation mode, interpolation is performed on those axes including the hypothetical axis. In the block where the hypothetical axis is specified, therefore, the speed of the nozzle end point will not be as specified.
The G07 1 command cancels hypothetical axis interpolation mode; in the coordinate system with the axis specified as the hypothetical axis, operation is restarted with the values assumed before hypothetical axis interpolation mode was set.

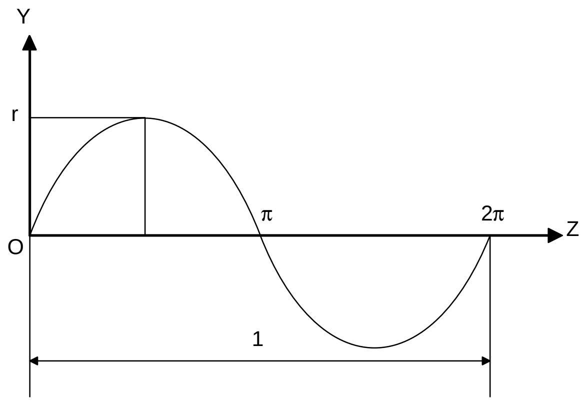
- **Sine interpolation**

Sine interpolation is possible by performing pulse distribution with one of the circular interpolation axes subject to helical interpolation used as a hypothetical axis.
If helical interpolation is performed on the YZ plane with the Z axis specified as a hypothetical axis, sine interpolation is performed on the X– and Y–axes. In sine interpolation on the YZ plane in one cycle, the hypothetical axis is the X axis.

$X^2 + Y^2 = r^2$ where r is the arc radius.

$Y = rSIN\left(\frac{2\pi}{l}Z\right)$ where l is the amount of movement on the

Z–axis in one cycle.



- **Interlock, stroke limits, and external deceleration**

Interlock, stroke limits, and external deceleration are effective even for the hypothetical axis.

- **Handle interrupts**

Handle interrupts are effective even on the hypothetical axis. Thus, movement is performed by a handle interrupt.

- **Gap control**

An axis can be specified as both a hypothetical and gap control axis at the same time. An axis specified as a gap control axis can also be specified as a hypothetical axis, while an axis specified as a hypothetical axis can also be specified as a gap control axis.

Limitations

- **Manual operation**

A hypothetical axis is effective during automatic operation only. It is not effective during manual operation.

- **Move command**

Hypothetical axis interpolation must be specified by incremental values only. If it is specified by absolute values, the feedrate for each axis on which interpolation has been performed may not be correct.

- **Coordinate rotation**

Coordinate rotation is not effective. Coordinate rotation cannot cause rotation about a helical axis.

- **Cutter compensation**

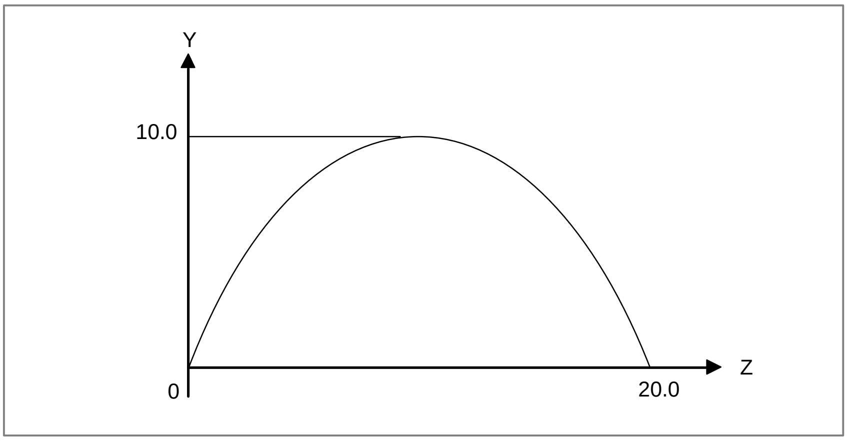
Cutter compensation cannot be specified. Even if it is specified, it is not performed normally.

- **Pre–interpolation acceleration/deceleration and advanced preview control**

If pre–interpolation acceleration/deceleration and advanced preview control are used, hypothetical axis interpolation cannot be specified. Advanced preview control mode must be turned off.

Examples

- **Sine interpolation**



N001 G07 X0 ;
N002 G91 G17 G03 X–20.2 Y0.0 I–10.0 Z20.0 F100 ;
N003 G01 X10.0 ;
N004 G07 X1 ;

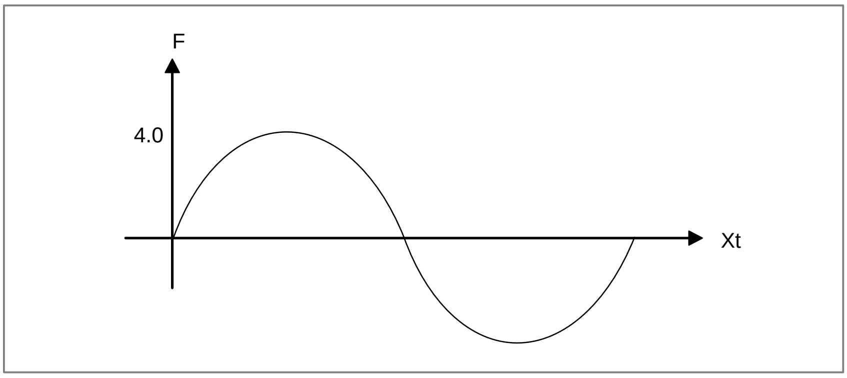
From the N002 to N003 blocks, the X–axis is set to a hypothetical axis. The N002 block specifies helical cutting in which the Z–axis is the linear axis. Since no movement takes place along the X axis, movement along the Y–axis is performed while performing sine interpolation along the Z–axis.

In the N003 block, there is no movement along the X–axis, and so the machine dwells until interpolation terminates.

- **Changing the feedrate to form a sine curve**

(Sample program)

G07Z0 ;	The Z–axis is set to a hypothetical axis.
G02X0Z0I10.0F4. ;	The feedrate on the X–axis changes sinusoidally.
G07Z1 ;	The use of the Z–axis as a hypothetical axis is canceled.



4.10 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. It is used also for measuring the dimensions of a workpiece.

Format

G31 IP_ ;

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 #5068, as follows:

#5061 X axis coordinate value
#5062 Y axis coordinate value
#5063 Z axis coordinate value
#5064 4th axis coordinate value
#5065 5th axis coordinate value
#5066 6th axis coordinate value
#5067 7th axis coordinate value
#5068 8th axis coordinate value

WARNING
Disable feedrate override, dry run, and automatic acceleration/deceleration (however, these become available by setting the parameter SKF No.6200#7 to 1.) allowing for an error in the position of the beam when a skip signal is input.
To disable beam output, set parameter No. 15004#2 to 1.

NOTE
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.

Examples

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

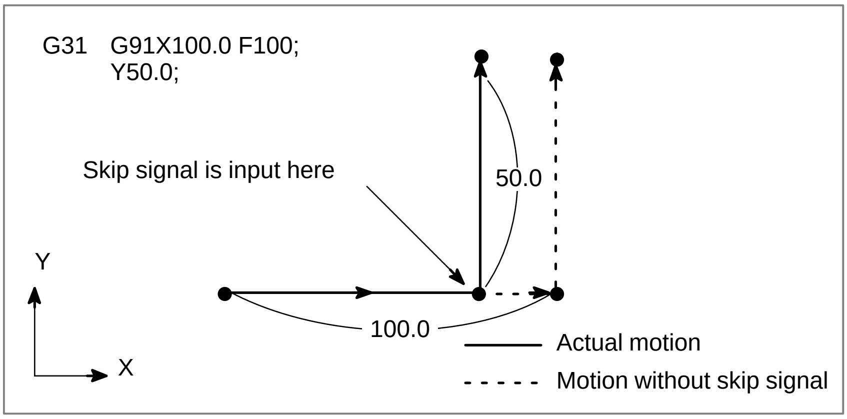


Fig.4.10 (a) The next block is an incremental command

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

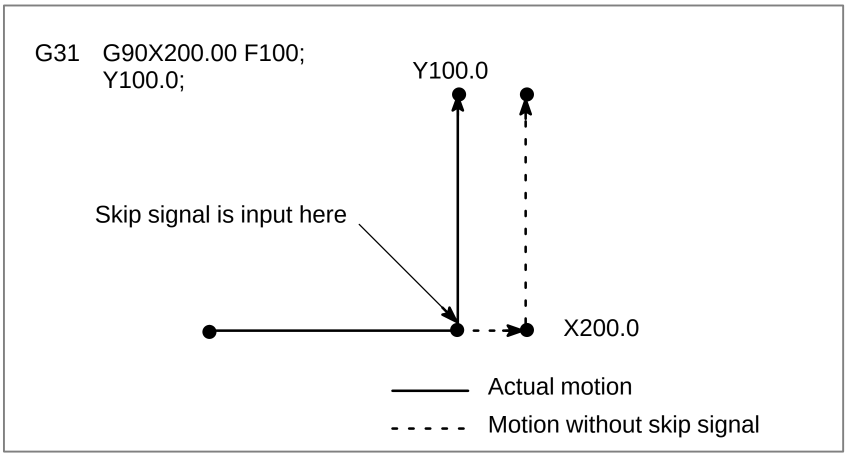


Fig.4.10 (b) The next block is an absolute command for 1 axis

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

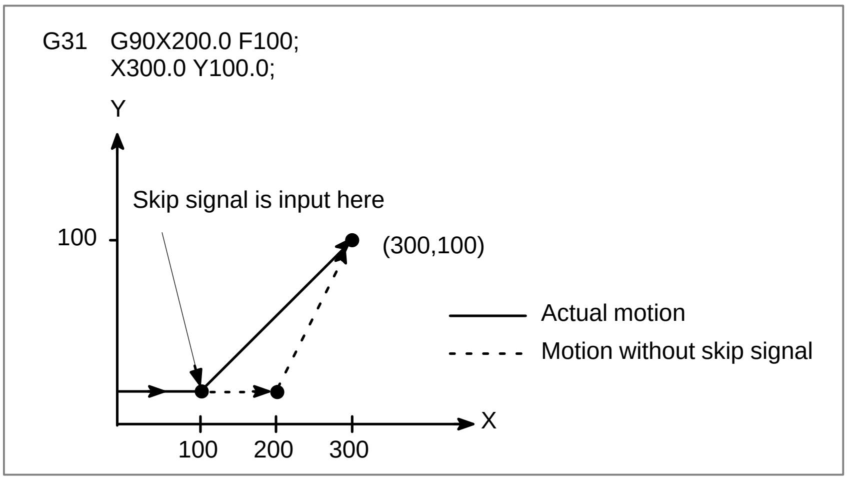


Fig.4.10 (c) The next block is an absolute command for 2 axes

4.11 MULTI–STEP SKIP (G31)

In a block specifying P1 to P4 after G31, the multi–step skip function stores coordinates in a custom macro variable when a skip signal (4–point or 8–point) is turned on.
Parameters No. 6202 to No. 6205 can be used to select a 4–point or 8–point skip signal. One skip signal can be set to match multiple Pn or Qn (n=1,2,3,4) as well as to match a Pn or Qn on a one–to–one basis. Parameters DS1 to DS8 (No. 6206 #0A#7) can be used for dwell.

Format

Move command

G31 IP__ F __ P __ ;

IP_ : End point
F_ : Feedrate
P_ : P1–P4

Dwell

G04 X (U, P)__ (Q__)
;

X(U, P)_ : Dwell time
Q_ : Q1 – Q4

Explanations

Multi–step skip is caused by specifying P1, P2, P3, or P4 in a G31 block. For an explanation of selecting (P1, P2, P3, or P4), refer to the manual supplied by the machine tool builder.
Specifying Q1, Q2, Q3, or Q4 in G04 (dwell command) enables dwell skip in a similar way to specifying G31. A skip may occur even if Q is not specified. For an explanation of selecting (Q1, Q2, Q3, or Q4), refer to the manual supplied by the machine tool builder.

- **Correspondence to skip signals**

Parameter Nos. 6202 to 6205 can be used to specify whether the 4–point or 8–point skip signal is used. Specification is not limited to one–to–one correspondence. It is possible to specify that one skip signal correspond to two or more Pn's or Qn's (n=1, 2, 3, 4). Also, bits 0 (DS1) to 7 (DS8) of parameter No. 6206 can be used to specify dwell.

CAUTION

Dwell is not skipped when Qn is not specified and parameters DS1–DS8 (No. 6206#0–#7) are not set.

4.12 HIGH SPEED SKIP SIGNAL (G31)

The skip function operates based on a high–speed skip signal (connected directly to the NC; not via the PMC) instead of an ordinary skip signal. In this case, up to eight signals can be input.
Delay and error of skip signal input is 0 – 2 msec at the NC side (not considering those at the PMC side).
This high–speed skip signal input function keeps this value to 0.1 msec or less, thus allowing high precision measurement.

For details, refer to the appropriate manual supplied from the machine tool builder.

Format

G31 IP_ ;

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

5 FEED FUNCTIONS

5.1 GENERAL

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

- **Feed functions**

1. Rapid traverse
   When the positioning command (G00) is specified, the nozzle moves at a rapid traverse feedrate set in the CNC (parameter No. 1420).
2. Cutting feed
   The nozzle 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 nozzle starts and ends its movement (Fig.5.1 (a)).

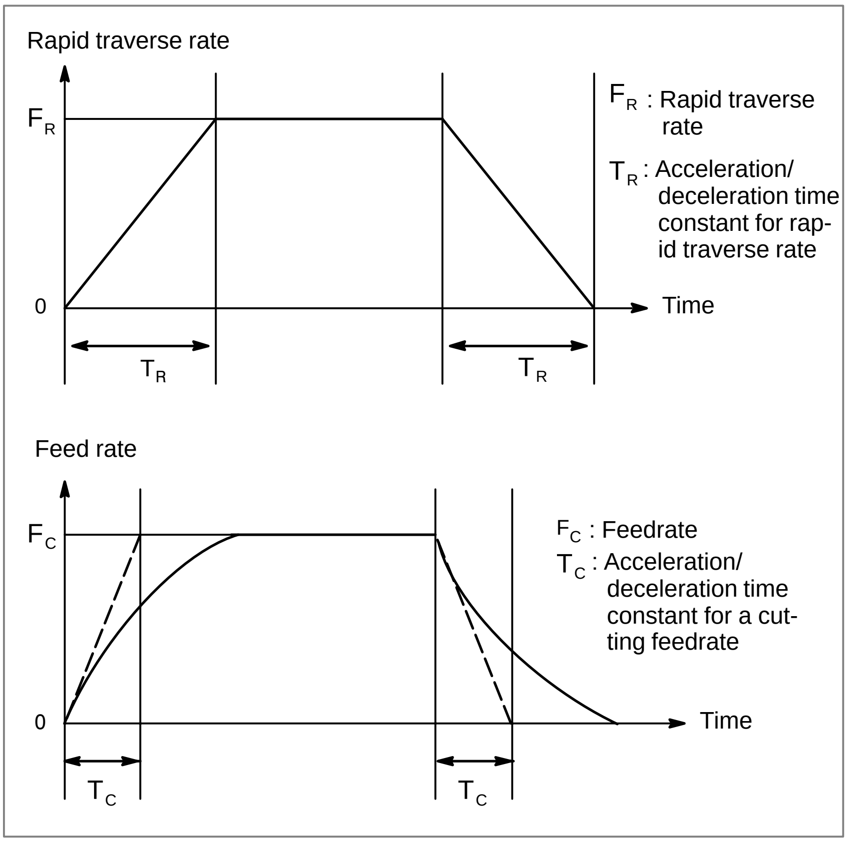


Fig.5.1 (a) Automatic acceleration/deceleration (example)

- **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.5.1 (b)).

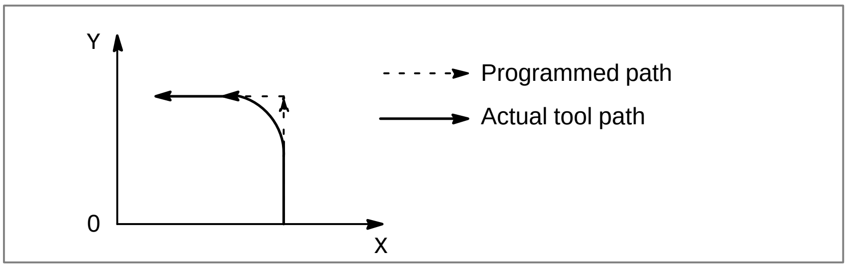


Fig.5.1 (b) Example of Tool Path between Two Blocks

In circular interpolation, a radial error occurs (Fig.5.1 (c)).

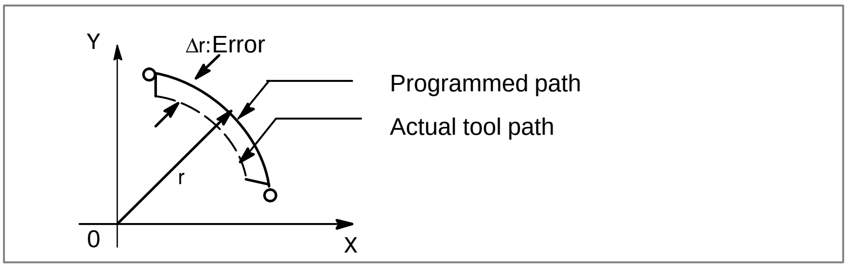


Fig.5.1 (c) Example of Radial Error in Circular Interpolation

The rounded–corner path shown in Fig.5.1 (b) and the error shown in Fig.5.1 (c) depend on the feedrate. So, the feedrate needs to be controlled for the nozzle to move as programmed.

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 nozzle 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).
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.

5.3 MACHINING FEED

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

1. Feed per minute
   After F, specify the amount of feed of the tool per minute.
2. One–digit F code feed
   Specify a desired one–digit number after F. Then, the feedrate set with the CNC for that number is set.

Format

Feed per minute
F_ ; Feedrate command (mm/min or inch/min)

One–digit F code feed
FN ;
N : Number from 1 to 9

Explanations

• Tangential speed constant control

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

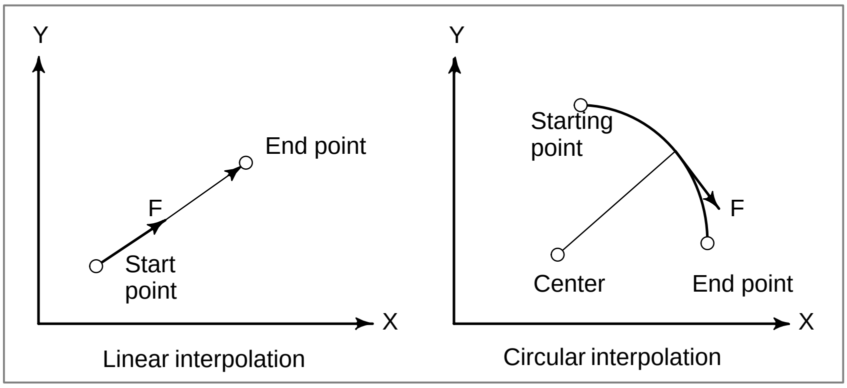


Fig.5.3 (a) Tangential feedrate (F)

• Feed per minute

Amount of feed of the nozzle per minute is to be directly specified by setting a number after F.
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 the appropriate manual of the machine tool builder.

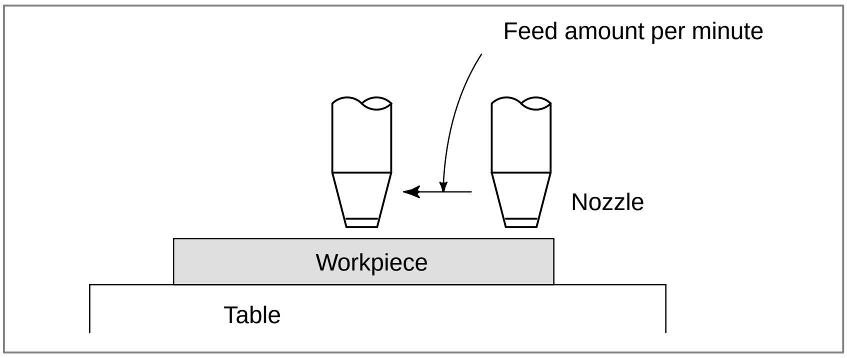


Fig.5.3 (b) Feed per minute

WARNING

No override can be used for some commands.

- **One–digit F code feed**

When a one–digit number from 1 to 9 is specified after F, the feedrate set for that number in a parameter (Nos. 1451 to 1459) is used. When F0 is specified, the rapid traverse rate is applied.
The feedrate corresponding to the number currently selected can be increased or decreased by turning on the switch for changing F1–digit feedrate on the machine operator's panel, then by rotating the manual pulse generator.
The increment/decrement, ΔF, in feedrate per scale of the manual pulse generator is as follows:

$$\Delta F = \frac{Fmax}{100X}$$

Fmax : feedrate upper limit for F1–F4 set by parameter (No.1460), or feedrate upper limit for F5–F9 set by parameter (No.1461)
X :any value of 1–127 set by parameter (No.1450)
The feedrate set or altered is kept even while the power is off. The current feed rate is displayed on the screen.

- **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

An upper limit is set in mm/min or inch/min. 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 range of feedrate command value.

5.4 CUTTING FEEDRATE CONTROL

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

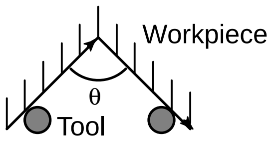
Table 5.4 Cutting Feedrate Control

Function name		G code	Validity of G code	Description
Exact stop		G09	This function is valid for specified blocks only.	The nozzle 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 G62 or G64 is specified.	The nozzle is decelerated at the end point of a block, then an in–position check is made. Then the next block is executed.
Machining mode		G64	Once specified, this function is valid until G61, G62, or G63 is specified.	The nozzle is not decelerated at the end point of a block, but the next block is executed.
Auto–matic corner over-ride	Automatic override for inner corners	G62	Once specified, this function is valid until G61 or G64 is specified.	When the nozzle moves along an inner corner during cutter compensation, over-ride is applied to the cutting feedrate to suppress the amount of cutting per unit of time so that a good surface finish can be produced.
	Internal circular cut-ting feedrate change	–	This function is valid in the cutter compensation mode, regardless of the G code.	The internal circular cutting feedrate is changed.

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.

2 Inner corner angle θ: $2° < \theta \leqq \alpha \leqq 178°$
(α is a set value)



Format

Exact stop	**G09 IP_ ;**
Exact stop mode	**G61 ;**
Machining mode	**G64 ;**
Automatic corner override	**G62 ;**

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

Explanations

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

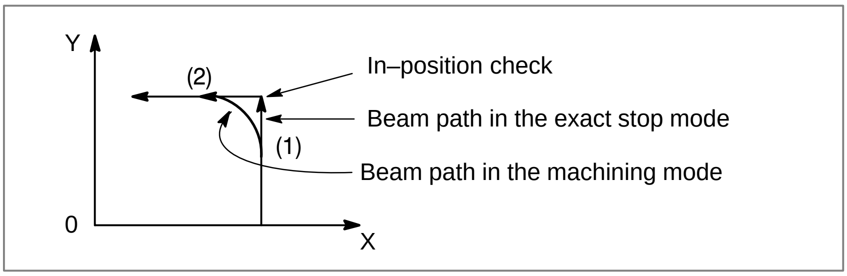


Fig.5.4.1 Example of Beam Paths from Block (1) to Block (2)

> **⚠ CAUTION**
> The machining mode (G64 mode) is set at power–on or system clear.

5.4.2 Automatic Corner Override

When cutter compensation is performed, the movement of the nozzle is automatically decelerated at an inner corner and internal circular area. This produces a smoothly machined surface.

5.4.2.1 Automatic override for inner corners (G62)

Explanations

- **Override condition**

When G62 is specified, and the nozzle path with cutter compensation applied forms an inner corner, the feedrate is automatically overridden at both ends of the corner.
There are four types of inner corners (Fig.5.4.2.1 (a)).
$2° \leqq \theta \leqq \theta p \leqq 178°$ in Fig.5.4.2.1 (a)
θp is a value set with parameter No. 1711. When θ is approximately equal to θp, the inner corner is determined with an error of 0.001,or less.

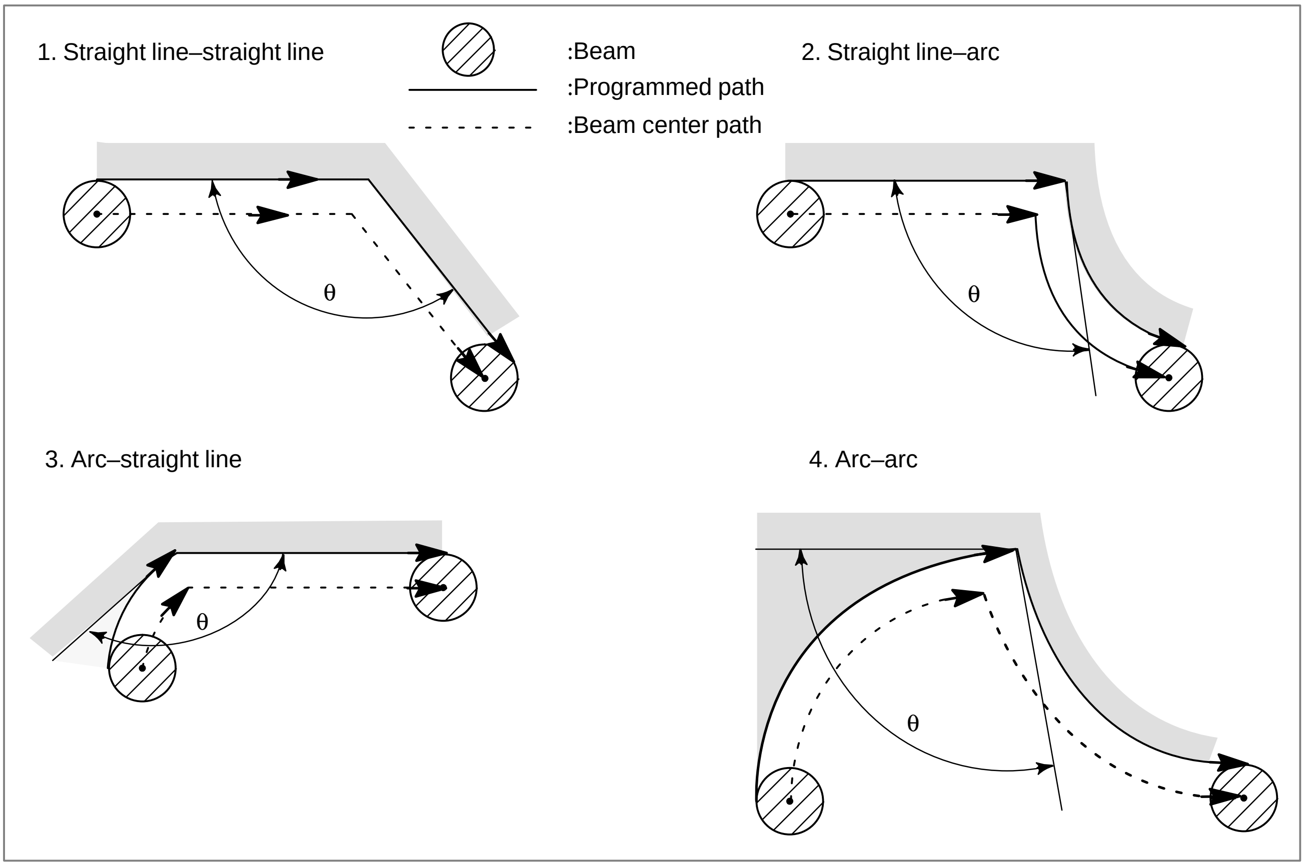


Fig.5.4.2.1 (a) Inner corner

Override range

When a corner is determined to be an inner corner, the feedrate is overridden before and after the inner corner. The distances Ls and Le, where the feedrate is overridden, are distances from points on the beam center path to the corner (Fig.5.4.2.1 (b), Fig.5.4.2.1 (c), Fig.5.4.2.1 (d)). Ls and Le are set with parameter Nos. 1713 and 1714.

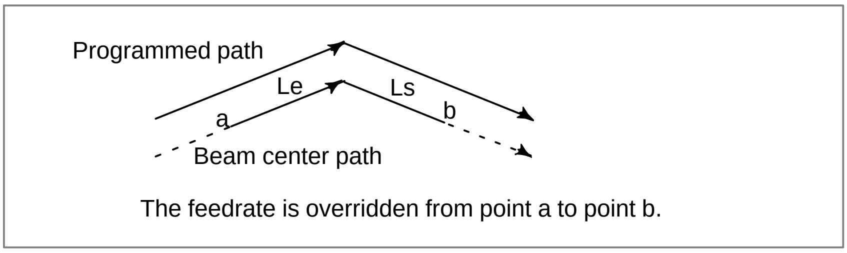


FIg.5.4.2.1 (b) Override Range (Straight Line to Straight Line)

When a programmed path consists of two arcs, the feedrate is overridden if the start and end points are in the same quadrant or in adjacent quadrants (Fig.5.4.2.1 (c)).

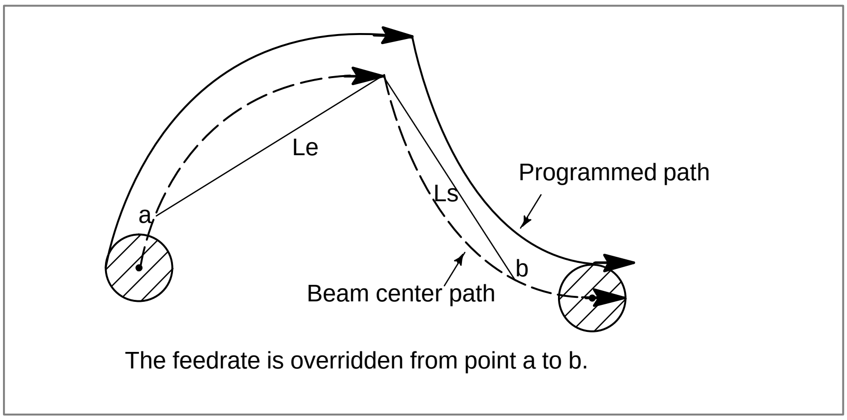


Fig.5.4.2.1 (c) Override Range (Arc to Arc)

Regarding program (2) of an arc, the feedrate is overridden from point a to point b and from point c to point d (Fig.5.4.2.1 (d)).

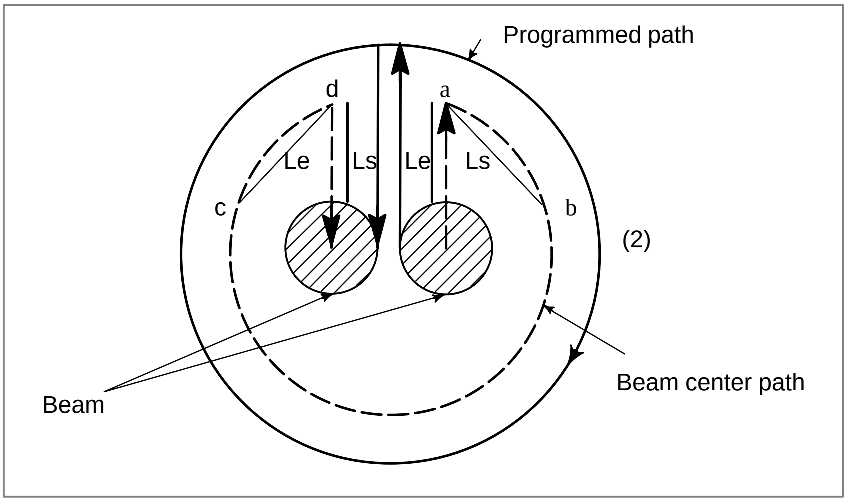


Fig.5.4.2.1 (d) Override Range (Straight Line to Arc, Arc to Straight Line)

Override value

An override value is set with parameter No. 1712. An override value is valid even for dry run and F1–digit specification.
In the feed per minute mode, the actual feedrate is as follows:

F × (automatic override for inner corners) × (feedrate override)

Limitations

- **Acceleration/deceleration before interpolation**

  Override for inner corners is disabled during acceleration/deceleration before interpolation.

- **Start–up/G41, G42**

  Override for inner corners is disabled if the corner is preceded by a start–up block or followed by a block including G41 or G42.

- **Offset**

  Override for inner corners is not performed if the offset is zero.

5.4.2.2 Internal circular machining feedrate change

For internally offset circular cutting, the feedrate on a programmed path is set to a specified feedrate (F) by specifying the circular cutting feedrate with respect to F, as indicated below (Fig.5.4.2.2). This function is valid in the cutter compensation mode, regardless of the G62 code.

$$F \times \frac{Rc}{Rp}$$

Rc : Beam center path radius
Rp : Programmed radius

It is also valid for the dry run and the one–digit F command.

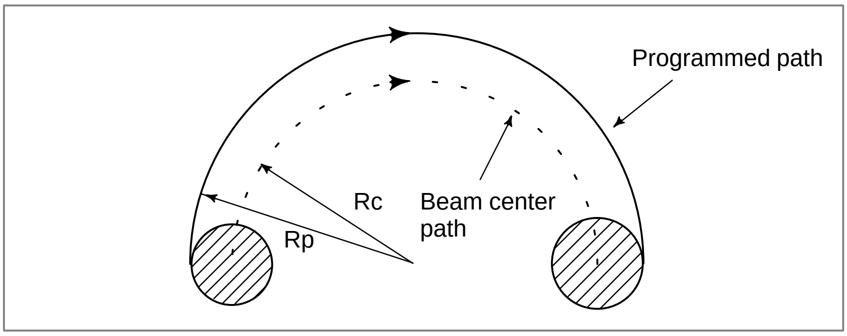


Fig.5.4.2.2 Internal circular cutting feedrate change

If Rc is much smaller than Rp, Rc/Rp ≐ 0; the tool stops. A minimum deceleration ratio (MDR) is to be specified with parameter No. 1710. When Rc/Rp ≦ MDR, the feedrate of the tool is (F×MDR).

⚠ CAUTION

When internal circular machining must be performed together with override for inner corners, the feedrate of the nozzle is as follows:

$$F \times \frac{Rc}{Rp} \times \text{(override for the inner corners)} \times \text{(feedrate override)}$$

5.4.3 Automatic Corner Deceleration

This function automatically controls the feedrate at a corner according to the corner angle between the machining blocks or the feedrate difference between the blocks along each axis.

This function is effective when ACD, bit 6 of parameter No. 1601, is set to 1, the system is in G64 mode (machining mode), and a cutting–feed block (block A) is followed by another cutting–feed block (block B).

The feedrate between machining blocks is controlled according to the corner angle between the blocks or the feedrate difference between the blocks along each axis. These two methods can be switched with CSD, bit 4 of parameter No. 1602.

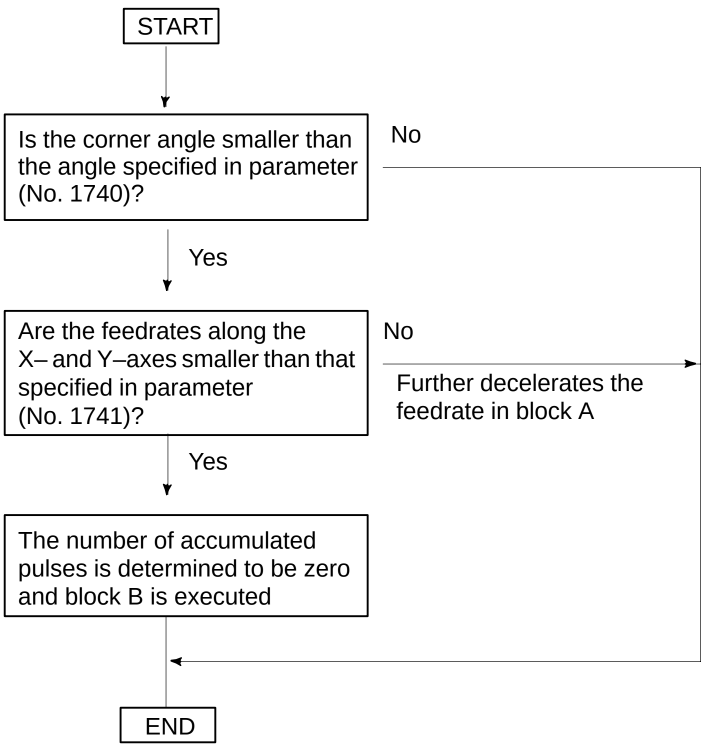
5.4.3.1 Corner deceleration according to the corner angle

This function decelerates the feedrate when the angle between blocks A and B on the selected plane is smaller than the angle specified in parameter No. 1740. The function executes block B when the feedrates along both the first and second axes are smaller than the feedrate specified in parameter No. 1741. In this case, the function determines that the number of accumulated pulses is zero.

Explanations

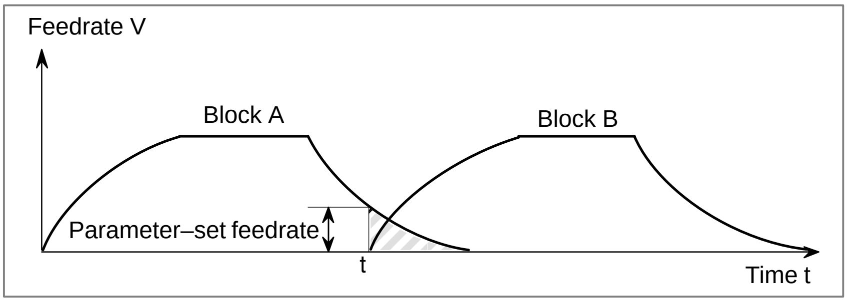
- **Flowchart for feedrate control**

The flowchart for feedrate control is shown below.



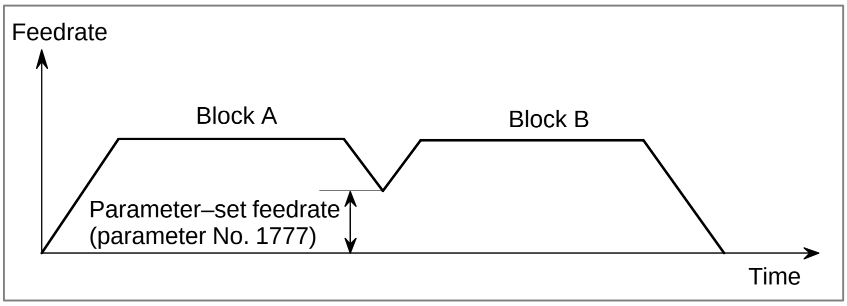
- **Feedrate and time**

When the corner angle is smaller than the angle specified in the parameter, the relationship between the feedrate and time is as shown below. Although accumulated pulses equivalent to the hatched area remain at time t, the next block is executed because the feedrate of the automatic acceleration/deceleration circuit is smaller than the parameter–set value. This function is effective only for movement on the selected plane.



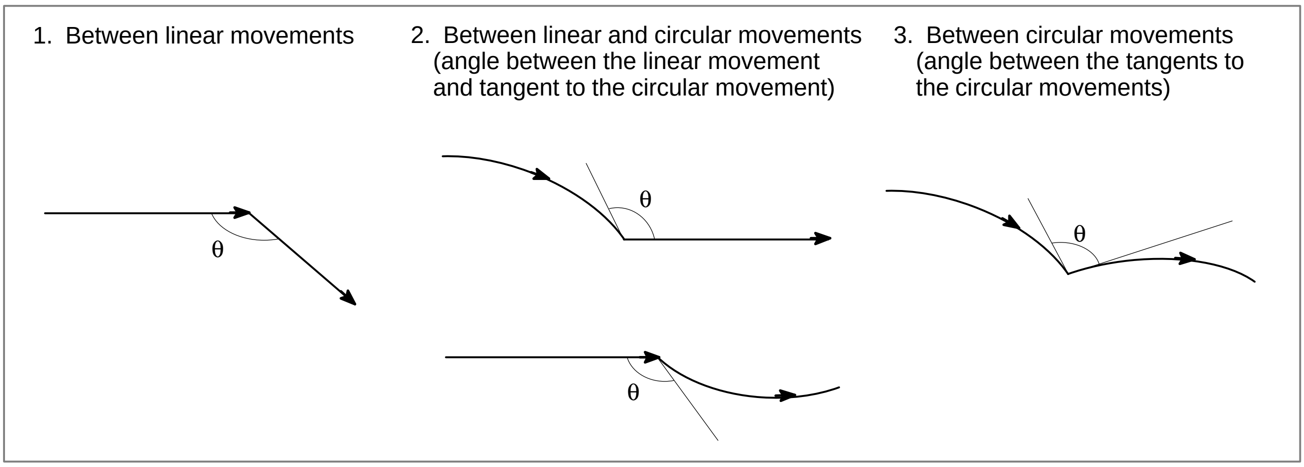
- **Acceleration/ deceleration before interpolation**

When acceleration/deceleration before interpolation is effective, the relationship between the feedrate and time is as shown below. When the angle between blocks A and B on the selected plane is smaller than the angle specified in parameter (No. 1740), and the feedrates specified in blocks A and B are larger than that specified in parameter (No. 1777), the feedrate is decelerated to the parameter–set value in block A, and accelerated to the feedrate specified in block B. The acceleration depends on the parameter for acceleration/deceleration before interpolation.



- **Angle between two blocks**

The angle between two blocks (blocks A and B) is assumed to be angle θ, as shown below.



- **Selected plane**

The machining angle is compared with the angle specified in parameter (No. 1740) for movements on the selected plane only. Machining feedrates are compared with that specified in parameter (No. 1741) for movement along the first and second axes on the selected plane only. This means, when movement occurs along three or more axes, only that movement along the first and second axes on the selected plane is considered.

- **Corner roundness**

Corner roundness is determined by the angle and feedrate specified in parameter (Nos. 1740 and 1741). To always make a sharp corner, set the angle to zero and the feedrate to 180000 (equivalent to 180 degrees).

- **Exact stop**

When G90 (exact stop) is specified, exact stop is performed irrespective of the angle and feedrate specified in parameter (Nos. 1740 and 1741).

- **Advanced preview control**

Those parameters related to automatic corner deceleration in advanced preview control mode are shown below.

Parameter description	Normal mode	Advanced preview control mode
Switching the methods for automatic corner deceleration	No.1602#4	No.1602#4
Lower limit of feedrate in automatic corner deceleration based on the angle	No.1777	No.1778
Limit angle in corner deceleration based on the angle	No.1740	No.1779

Limitations

This function cannot be enabled for a single block or during dry run.

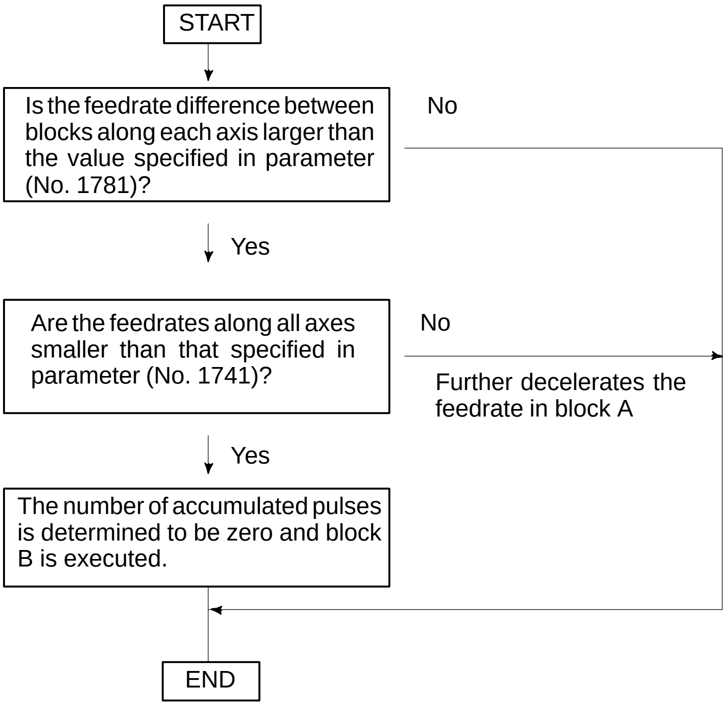
5.4.3.2 Corner deceleration according to the feedrate difference between blocks along each axis

This function decelerates the feedrate when the difference between the feedrates at the end point of block A and the start point of block B along each axis is larger than the value specified in parameter No. 1781. The function executes block B when the feedrates along all axes are smaller than the feedrate specified in parameter No. 1741. In this case, the function determines that the number of accumulated pulses is zero.

Explanations

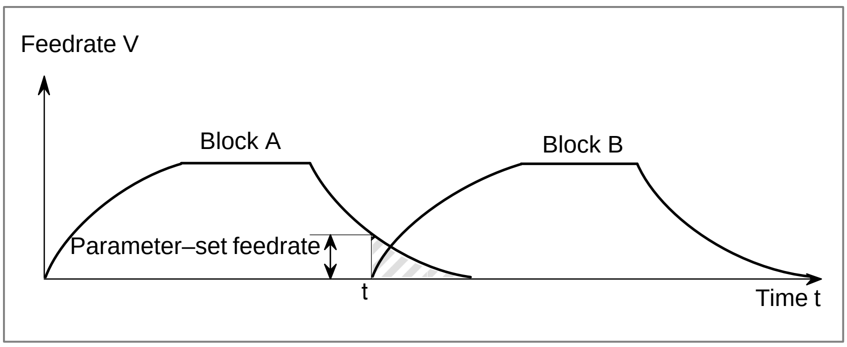
- **Flowchart for feedrate control**

The flowchart for feedrate control is shown below.



- **Feedrate and time**

When the feedrate difference between blocks along each axis is larger than the value specified in parameter No. 1781, the relationship between the feedrate and time is as shown below. Although accumulated pulses equivalent to the hatched area remain at time t, the next block is executed because the feedrate of the automatic acceleration/deceleration circuit is smaller than the feedrate specified in parameter No. 1741.



• Acceleration / deceleration before interpolation

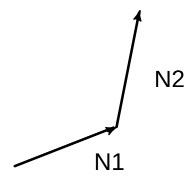
When acceleration/deceleration before interpolation is effective, the relationship between the feedrate and time is as described below.
When the feedrate difference between blocks A and B along each axis is larger than the value specified in parameter No. 1780, the feedrate is decelerated to the corner feedrate calculated from the feedrate difference along each axis.
Let the feedrate be F. Compare the feedrate difference along each axis (Vc[X], Vc[Y], ...) with the value specified in parameter No. 1780, Vmax. When the difference exceeds Vmax, calculate R as shown below.

$$R=\frac{Vc}{Vmax}$$

Find the maximum value for R among the calculated values for the axes. Let it be Rmax. Then, the corner feedrate can be obtained as follows:

$$Fc=F \times \frac{1}{Rmax}$$

(Example)



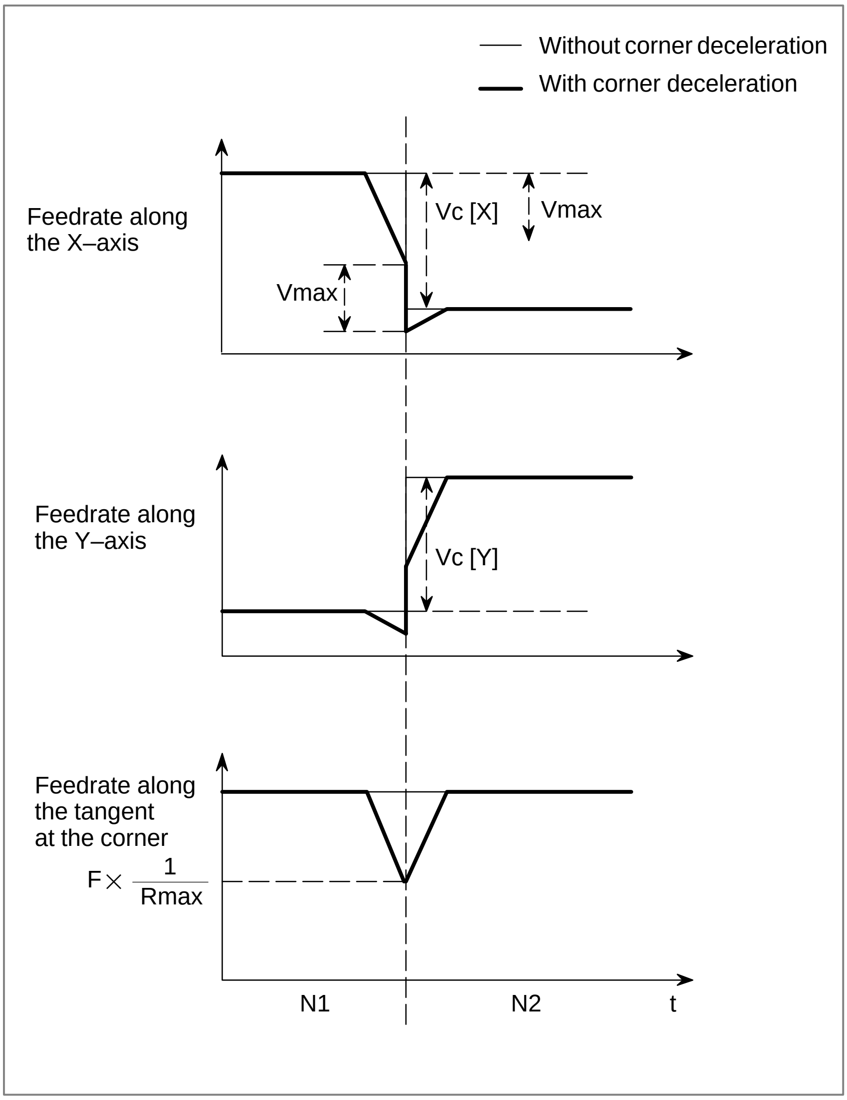
N1 G01 G91 X80. Y20. F3000 ;
N2 X20. Y80. ;

When this movement is specified, the feedrate along each axis is as shown in the next figure.

$$Rmax=\frac{Vc\ [X(Y)]}{Vmax}$$

$$F \times \frac{1}{Rmax}$$

From the figure, it can be seen that the feedrate differences along the X– and Y–axes (Vc[X] and Vc[Y]) exceed Vmax. Calculate Rmax to get Fc. When the feedrate is decelerated to Fc at the corner, the feedrate difference along each axis do not exceed Vmax.



- **Setting the allowable feedrate difference along each axis**

The allowable feedrate difference can be specified for each axis in parameter No. 1783.

- **Checking the feedrate difference**

The feedrate difference is also checked during dry–run operation or during deceleration caused by an external signal, using feedrate commands specified in a program.

- **Exact stop**

When G90 (exact stop) is specified, exact stop is performed irrespective of the parameter settings.

- **Override**

If an override is changed during operation, the feedrate difference will not be checked correctly.

• Advanced preview control

Parameters related to automatic corner deceleration in advanced preview control mode are shown below.

Parameter description	Normal mode	Advanced preview control mode
Switching the methods for automatic corner deceleration	No.1602#4	No.1602#4
Allowable feedrate difference (for all axis) in automatic corner deceleration based on the feedrate difference	No.1780	No.1780
Allowable feedrate difference (for each axis) in automatic corner deceleration based on the feedrate difference	No.1783	No.1783

Limitations

This function is not effective for feed–per–rotation commands, address–F–with–one–digit commands, rigid tapping, and a single block.

5.5 DWELL (G04)

Format

Dwell G04 X_ ; or G04 P_ ;

X_ : Specify a time
P_ : Specify a time

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 machining mode (G64 mode).

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

Increment system	Command value range	Dwell time unit
IS–B	0.001 to 99999.999	sec

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

Increment system	Command value range	Dwell time unit
IS–B	1 to 99999999	0.001 sec

6 REFERENCE POSITION

A CNC machine tool has a special position where, generally, the nozzle is exchanged or the coordinate system is set, as described later. This position is referred to as a reference position.

6.1 REFERENCE POSITION RETURN

General

- **Reference position**

The reference position is a fixed position on a machine nozzle to which the tool can easily be moved by the reference position return function. Up to four reference positions can be specified by setting coordinates in the machine coordinate system in parameters (No. 1240 to 1243).

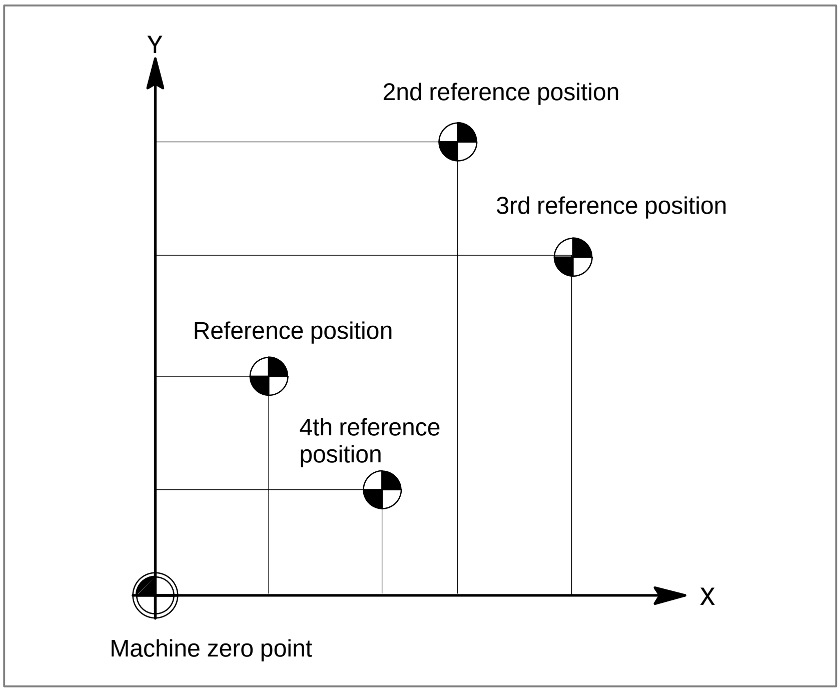


Fig.6.1 (a) Machine zero point and reference positions

- **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.

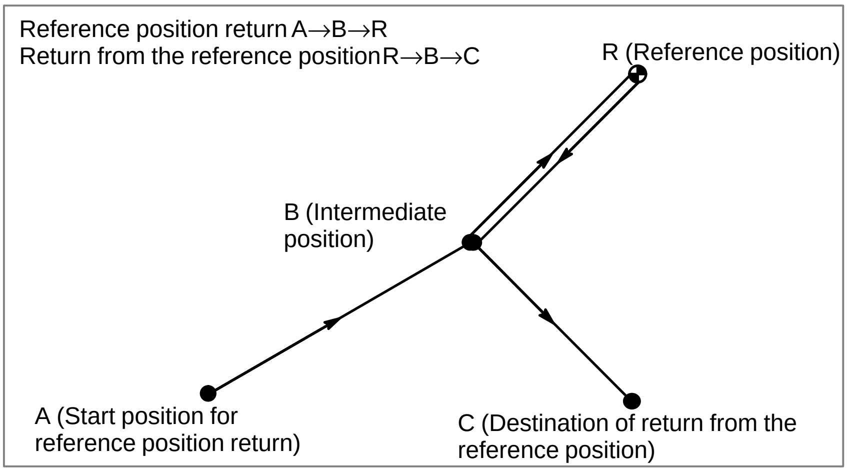


Fig.6.1 (b) Reference position return and return form the reference position

- **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.)**

G30 P3 IP_ ; 3rd reference position return

G30 P4 IP_ ; 4th reference position return

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, 3rd, and 4th reference position return (G30)**

In a system without an absolute–position detector, the first, third, and fourth 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.

- **Setting of the reference position return feedrate**

Before a machine coordinate system is established with the first return to the reference position after power–on, the manual and automatic reference position return feedrates and automatic rapid traverse rate conform to the setting of parameter No. 1428, if any, for each axis. Even after a machine coordinate system is established upon the completion of reference position return, the manual reference position return feedrate conforms to the setting of the parameter. Even after a machine coordinate system is established upon the completion of reference position return, the manual reference postiion return feedrate conforms to the setting of the parameter.

NOTE

1 To this feedrate, a rapid traverse override (F0 ,25,50,100%) is applied, for which the setting is 100%.
2 After a machine coordinate system has been established upon the completion of reference position return, the automatic reference position return feedrate will conform to the ordinary rapid traverse rate.
3 For the manual rapid traverse rate used before a machine coordinate system is estavlished upon the completion of reference position return a jog feedrate or manual rapid traverse rate can be selected usting RPD (bit 0 of parameter No. 1401).

	Before a coordinate system is established	After a coordinate system is established
Automatic reference position return (G28)	No. 1428	No.1420
Automatic rapid traverse (G00)	No.1428	No.1420
Manual reference position return	No.1428	No.1428
Manual rapid traverse rate	No.1423 *1	No.1424

NOTE

4 When parameter No. 1428 is set to 0, the feedrates conform to the parameter settings shown below.

	Before a coordinate system is established	After a coordinate system is established
Automatic reference position return (G28)	No. 1420	No.1420
Automatic rapid traverse (G00)	No.1420	No.1420
Manual reference position return	No.1424	No.1424
Manual rapid traverse rate	No.1423 *1	No.1424

1420 : Rapid traverse rate
1423 : Jog feedrate
1424 : Manual rapid traverse rate

*1 Setting of parameter No.1424 when RPD (bit 0 of parameter No.1401) is set to 1.

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 setting increment. This is because the least setting 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)
M1111 ; (Changing the tool at the reference position)
G29X1300.0Y200.0 ; (Programs movement from B to C)

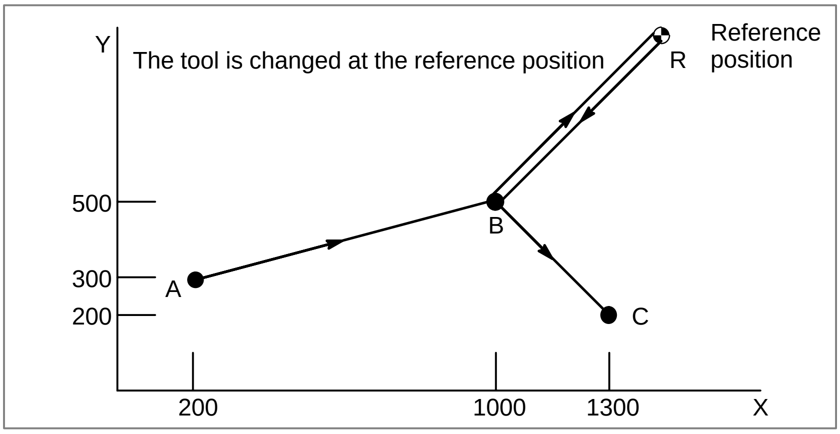


Fig.6.1 (c) Reference position return and return from the reference position

6.2 FLOATING REFERENCE POSITION RETURN (G30.1)

Tools ca be returned to the floating reference position.
A floating reference point is a position on a machine tool, and serves as a reference point for machine tool operation.
A floating reference point need not always be fixed, but can be moved as required.

Format

G30.1 IP_ ;

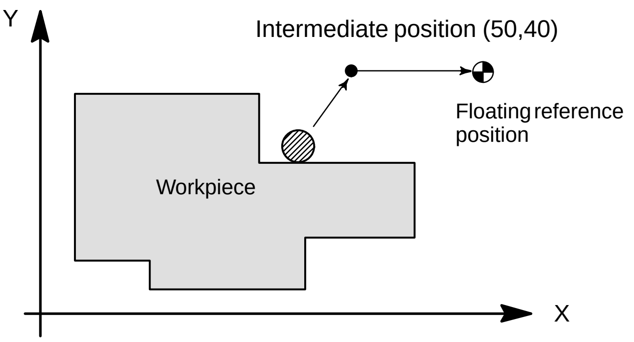
IP_ : Command of the intermediate position of the floating reference position
(Absolute command/incremental command)

Explanations

Generally speaking, on a machining center or milling machine, cutting tools can be replaced only at specific positions. A position where tools can be replaced is defined as the second or third reference point. Using G30 can easily move the cutting tools back to these points. On some machine tools, the cutting tools can be replaced at any position unless they interfere with the workpiece.
With these machines, the cutting tools should be replaced at a position as close to the workpiece as possible so as to minimize the machine cycle time. For this purpose, the tool change position is to be changed, depending on the figure of the workpiece. This operation can easily be performed using this function. That is, a tool change position suitable for the workpiece is memorized as a floating reference point. Then command G30. 1 can easily cause return to the tool change position.
A floating reference point becomes a machine coordinate position memorized by pressing the soft key **[SET FRP]** on the current positions display screen (see III–11.1.7). The G30.1 block first positions the tool at the intermediate point along the specified axes at rapid traverse rate, then further moves the tool from the intermediate point to the floating reference point at rapid traverse rate.
Before using G30.1, cancel cutter compensation and tool length compensation.
A floating reference point is not lost even if power is turned off.
The function for returning from the reference position (G29) can be used for moving the tool from the floating reference position (see II–6.1).

Examples

G30.1 G90 X50.0 Y40.0 ;



7 COORDINATE SYSTEM

By teaching the CNC a desired nozzle position, the nozzle can be moved to the position. Such a nozzle 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.

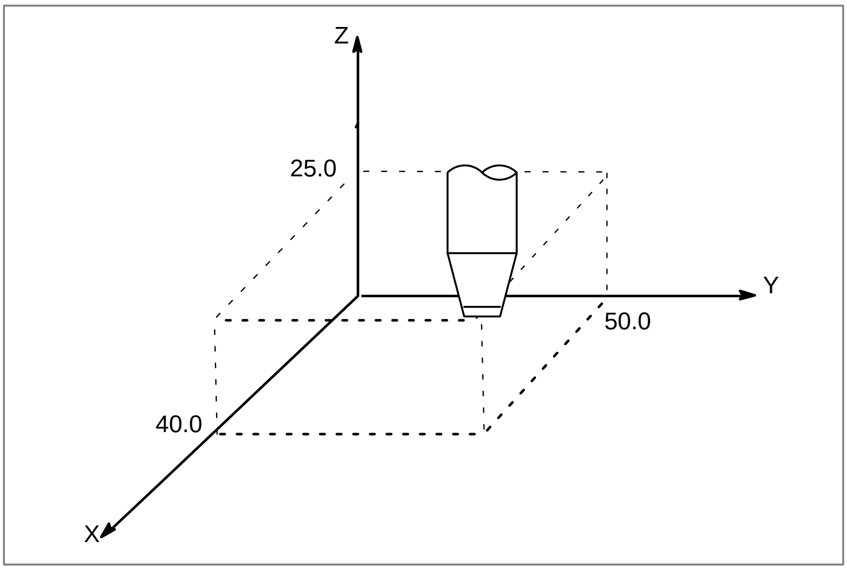


Fig.7 Nozzle position specified by X40.0Y50.0Z25.0

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 and serves as the reference of the 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.

Format

(G90)G53 IP_ ;
IP_; Absolute dimension word

Explanations

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

When a command is specified the position on a machine coordinate system, the nozzle moves to the position by rapid traverse. G53, which is used to select a machine coordinate system, is a one–shot G code; that is, it is valid only in the block in which it is specified on a machine coordinate system. Specify an absolute command (G90) for G53. When an incremental command (G91) is specified, the G53 command is ignored. When the nozzle is to be moved to a machine–specific position such as a nozzle change position, program the movement in a machine coordinate system based on G53.

Restrictions

- **Cancel of the compensation function**

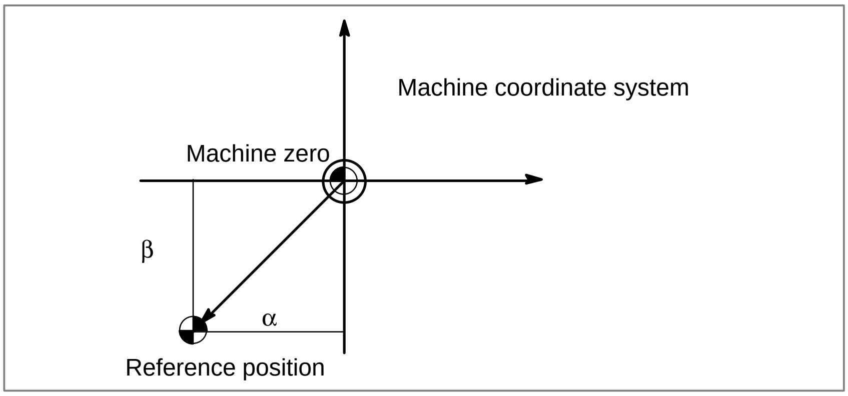
When the G53 command is specified, cancel the cutter compensation, tool length offset, and tool offset.

- **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.

Reference

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.



7.2 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 bit 0 of parameter SPR No. 1201 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed (see Part III–3.1.).
This function is, however, disabled when the workpiece coordinate system option is being used.

(3) Input using the MDI panel
Six workpiece coordinate systems can be set beforehand using the MDI panel (see Part III–11.4.5.).
When using an absolute command, establish the workpiece coordinate system in any of the above ways.

Format

- **Setting a workpiece coordinate system by G92**

(G90) G92 IP_

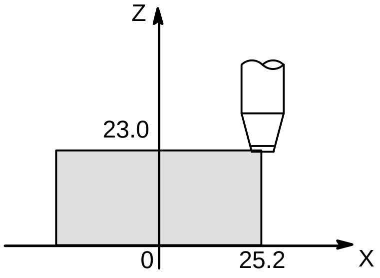
Explanations

A workpiece coordinate system is set so that a point on the nozzle, such as the nozzle tip, is at specified coordinates. If a coordinate system is set using G92 during nozzle 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

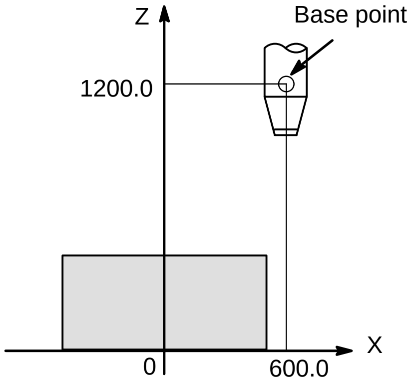
Example 1

Setting the coordinate system by the G92X25.2Z23.0; command
(The nozzle 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 nozzle 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 nozzle tip to the commanded position, the difference from the nozzle tip to the base point is compensated by tool length offset.

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 II– 7.2.1.)

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

(2) Choosing from six workpiece coordinate systems set using the MDI panel

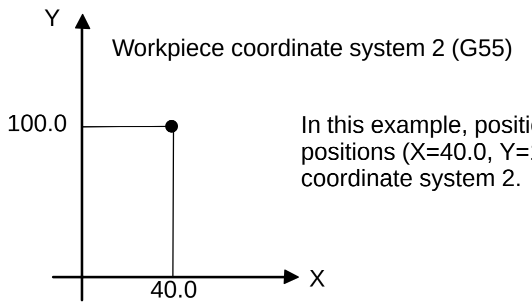
By specifying a G code from G54 to G59, one of the workpiece coordinate systems 1 to 6 can be selected.

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

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 MDI panel (see III–11.4.5)

(2) Programming by G10 or G92

(3) Using the external data input function
An external workpiece zero point offset value can be changed by input signal to CNC. Refer to machine tool builder's manual for details

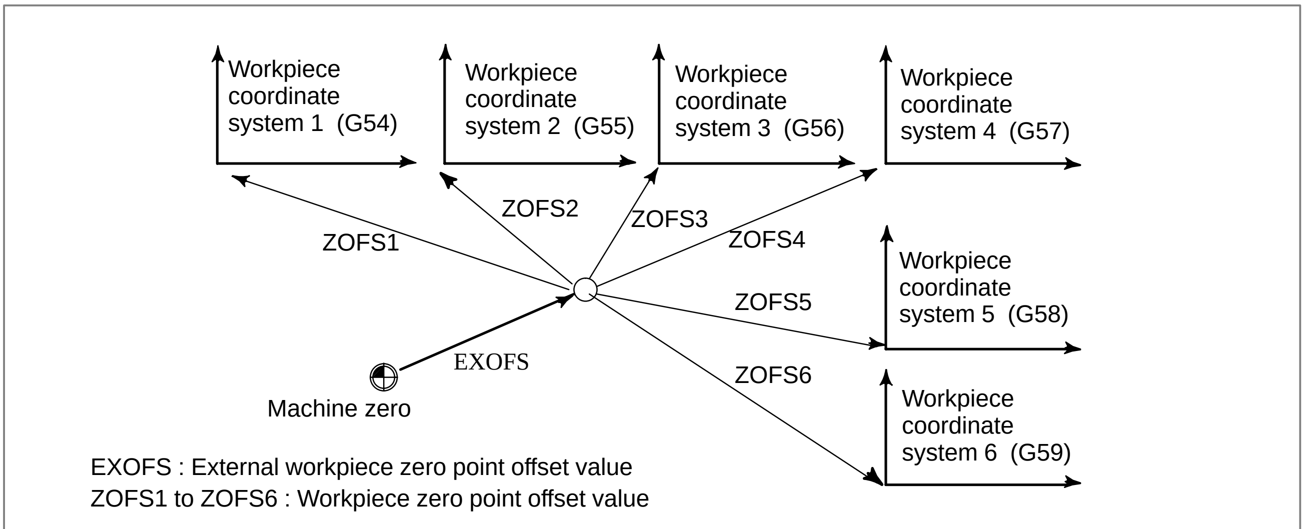


Fig.7.2.3 Changing an external workpiece zero point offset value or workpiece zero point offset value

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 : For an absolute command (G90), workpiece zero point offset for each axis.
For an incremental command (G91), value to be added to the set workpiece zero point offset for each axis (the result of addition becomes the new workpiece zero point offset).

- **Changing by G92**

G92 IP_;

Explanations

- **Changing by G10**

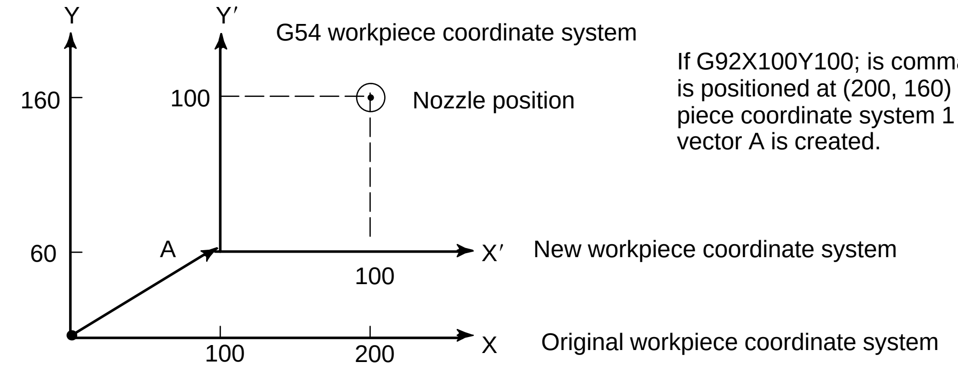
With the G10 command, each workpiece coordinate system can be changed separately.

- **Changing by G92**

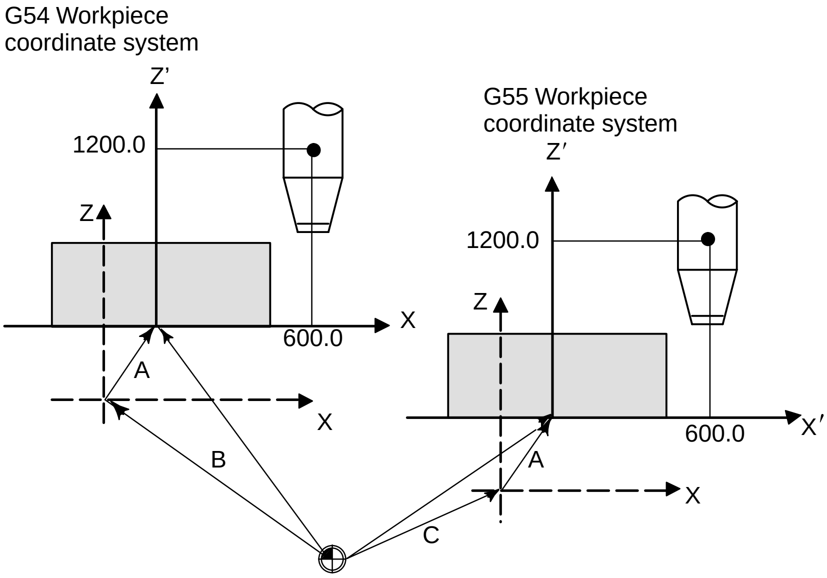
By specifying G92IP_;, a workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).
Then, the amount of coordinate system shift is added to all the workpiece zero point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

> **WARNING**
> 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 nozzle 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,C : Workpiece zero point offset value in the original workpiece coordinate system

Suppose that a G54 workpiece coordinate system is specified. Then, a G55 workpiece coordinate system where the black circle on the nozzle (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.

7.2.4 Workpiece Coordinate System Preset (G92.1)

The workpiece coordinate system preset function presets a workpiece coordinate system shifted by manual intervention to the pre–shift workpiece coordinate system. The latter system is displaced from the machine zero point by a workpiece zero point offset value.
There are two methods for using the workpiece coordinate system preset function. One method uses a programmed command (G92.1). The other uses MDI operations on the absolute position display screen, relative position display screen, and overall position display screen (III– 11.1.4).

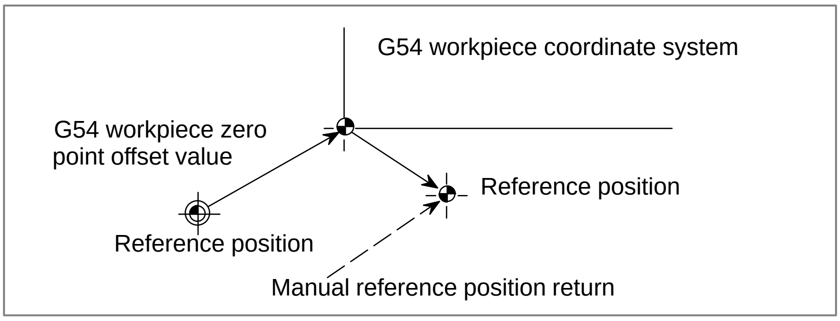
Format

G92.1 IP 0 ;

IP 0 ; Specifies axis addresses subject to the workpiece coordinate system preset operation. Axes that are not specified are not subject to the preset operation.

Explanations

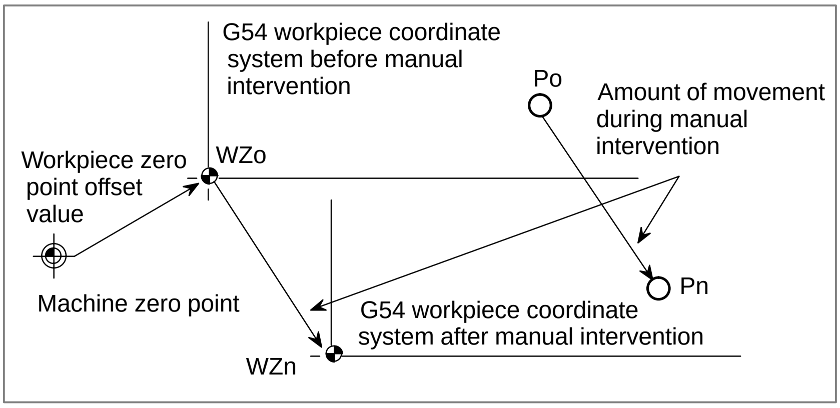
When manual reference position return operation is performed in the reset state, a workpiece coordinate system is shifted by the workpiece zero point offset value from the machine coordinate system zero point. Suppose that the manual reference position return operation is performed when a workpiece coordinate system is selected with G54. In this case, a workpiece coordinate system is automatically set which has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value; the distance from the zero point of the workpiece coordinate system to the reference position represents the current position in the workpiece coordinate system.



If an absolute position detector is provided, the workpiece coordinate system automatically set at power–up has its zero point displaced from the machine zero point by the G54 workpiece zero point offset value. The machine position at the time of power–up is read from the absolute position detector and the current position in the workpiece coordinate system is set by subtracting the G54 workpiece zero point offset value from this machine position. The workpiece coordinate system set by these operations is shifted from the machine coordinate system using the commands and operations listed next page.

(a) Manual intervention performed when the manual absolute signal is off
(b) Move command executed in the machine lock state
(c) Movement by handle interrupt
(d) Operation using the mirror image function
(e) Setting the local coordinate system using G52, or shifting the workpiece coordinate system using G92

In the case of (a) above, the workpiece coordinate system is shifted by the amount of movement during manual intervention.



In the operation above, a workpiece coordinate system once shifted can be preset using G code specification or MDI operation to a workpiece coordinate system displaced by a workpiece zero point offset value from the machine zero point. This is the same as when manual reference position return operation is performed on a workpiece coordinate system that has been shifted. In this example, such G code specification or MDI operation has the effect of returning workpiece coordinate system zero point WZn to the original zero point WZo, and the distance from WZo to Pn is used to represent the current position in the workpiece coordinate system.

Bit 3 (PPD) of parameter No. 3104 specifies whether to preset relative coordinates (RELATIVE) as well as absolute coordinates.

When no workpiece coordinate system option (G54 to G59) is selected, the workpiece coordinate system is preset to the coordinate system set by automatic workpiece coordinate system setting. When automatic workpiece coordinate system setting is not selected, the workpiece coordinate system is preset with its zero point placed at the reference position.

Limitations

- **Cutter compensation, tool offset**

When using the workpiece coordinate system preset function, cancel compensation modes: cutter compensation and tool offset. If the function is executed without cancelling these modes, compensation vectors are temporarily cancelled.

- **Program restart**

The workpiece coordinate system preset function is not executed during program restart.

- **Prohibited modes**

Do not use the workpiece coordinate system preset function when the scaling, coordinate system rotation, programmable image, or drawing copy mode is set.

7.2.5 Adding Workpiece Coordinate Systems (G54.1 or G54)

Besides the six workpiece coordinate systems (standard workpiece coordinate systems) selectable with G54 to G59, 48 additional workpiece coordinate systems (additional workpiece coordinate systems) can be used. Alternatively, up to 300 additional workpiece coordinate systems can be used.

Format

- **Selecting the additional workpiece coordinate systems**

G54.1Pn ; or G54Pn ;

Pn : Codes specifying the additional workpiece coordinate systems
n : 1 to 48 or 1 to 300

- **Setting the workpiece zero point offset value in the additional workpiece coordinate systems**

G10L20 Pn IP_;

Pn : Codes specifying the workpiece coordinate system for setting the workpiece zero point offset value
n : 1 to 48 or 1 to 300
IP_ : Axis addresses and a value set as the workpiece zero point offset

Explanations

- **Selecting the additional workpiece coordinate systems**

When a P code is specified together with G54.1 (G54), the corresponding coordinate system is selected from the additional workpiece coordinate systems (1 to 48 or 1 to 300).
A workpiece coordinate system, once selected, is valid until another workpiece coordinate system is selected. Standard workpiece coordinate system 1 (selectable with G54) is selected at power–on.

G54.1 P1 . . . . . . Additional workpiece coordinate system 1
G54.1 P2 . . . . . . Additional workpiece coordinate system 2
:
G54.1 P48 . . . . . Additional workpiece coordinate system 48
:
G54.1 P300 . . . . Additional workpiece coordinate system 300

As with the standard workpiece coordinate systems, the following operations can be performed for a workpiece zero point offset in an additional workpiece coordinate system:

(1) The OFFSET function key can be used to display and set a workpiece zero point offset value.

(2) The G10 function enables a workpiece zero point offset value to be set by programming (refer to II–7.2.3).

(3) A custom macro allows a workpiece zero point offset value to be handled as a system variable.

(4) Workpiece zero point offset data can be entered or output as external data.

(5) The PMC window function enables workpiece zero point offset data to be read as program command modal data.

- **Setting the workpiece zero point offset value in the additional workpiece coordinate systems**

When an absolute workpiece zero point offset value is specified, the specified value becomes a new offset value. When an incremental workpiece zero point offset value is specified, the specified value is added to the current offset value to produce a new offset value.

Limitations

- **Specifying P codes**

A P code must be specified after G54.1 (G54). If G54.1 is not followed by a P code in the same block, additional workpiece coordinate system 1 (G54.1P1) is assumed.
If a value not within the specifiable range is specified in a P code, an P/S alarm (No. 030) is issued.
P codes other than workpiece offset numbers cannot be specified in a G54.1 (G54) block.

Example) G54.1 (G54) G04 P1000 ;

7.3 LOCAL COORDINATE SYSTEM

When a program is created in a workpiece coordinate system, a child workpiece coordinate system can 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.

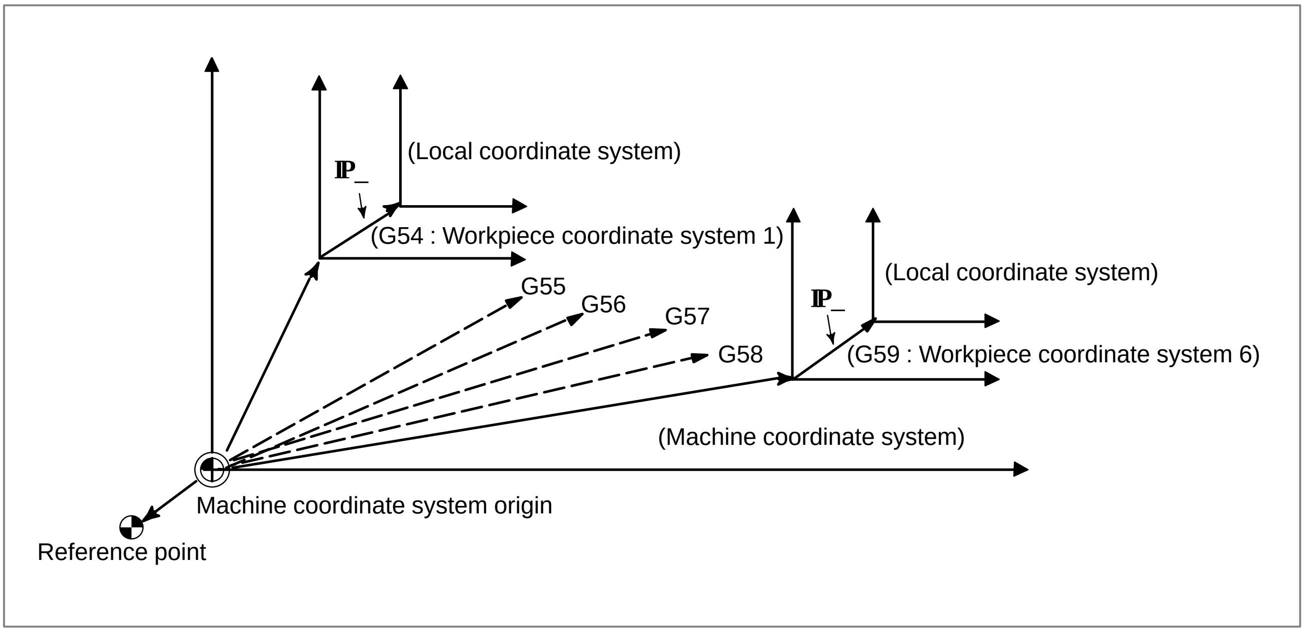


Fig.7.3 Setting the local coordinate system

WARNING

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 of parameter No.1202 is set to 1.

4 If coordinate values are not specified for all axes when setting a workpiece coordinate system with the G92 command, the local coordinate systems of axes for which coordinate values were not specified are not cancelled, but remain unchanged.

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.
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			

Xp, Yp, Zp are determined by the axis address appeared in the block in which G17, G18 or G19 is commanded.
When an axis address is omitted in G17, G18 or G19 block, it is assumed that the addresses of basic three axes are omitted.
Parameter No. 1022 is used to specify that an optional axis be parallel to the each axis of the X, Y–, and Z–axes as the basic three axes.
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,
G17U_Y_	UY plane
G18X_Z_	ZX plane
X_Y_	Plane is unchanged (ZX plane)
G17	XY plane
G18	ZX plane
G17 U_	UY 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)

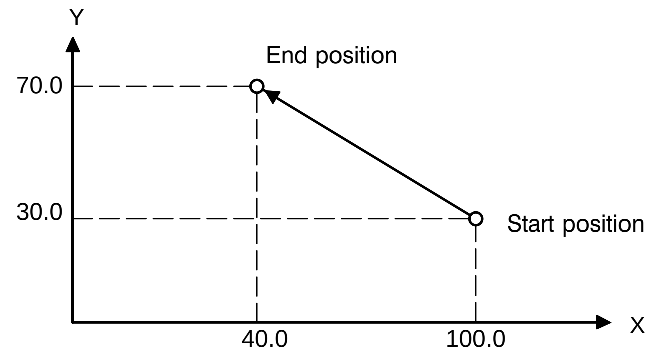
There are two ways to command travels of the nozzle; the absolute command, and the incremental command. In the absolute command, coordinate value of the end position is programmed; in the incremental command, move distance of the position itself is programmed. G90 and G91 are used to command absolute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

Format

Absolute command	**G90 IP_ ;**
Incremental command	**G91 IP_ ;**

Examples

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



8.2 POLAR COORDINATE COMMAND (G15, G16)

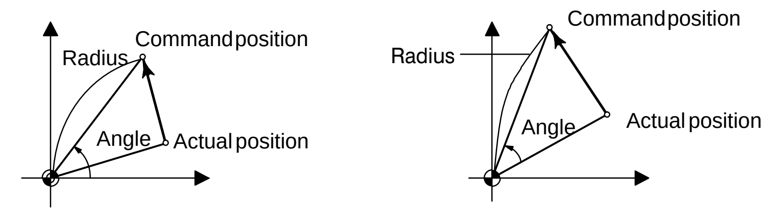
The end point coordinate value can be input in polar coordinates (radius and angle).
The plus direction of the angle is counterclockwise of the selected plane first axis + direction, and the minus direction is clockwise.
Both radius and angle can be commanded in either absolute or incremental command (G90, G91).

Format

G□□ G○○ G16 ;	Starting the polar coordinate command (polar coordinate mode)
G00 IP _ ; ⋮	Polar coordinate command
G15 ;	Canceling the polar coordinate command (polar coordinate mode)
G16	Polar coordinate command
G15	Polar coordinate command cancel
G□□	Plane selection of the polar coordinate command (G17, G18 or G19)
G○○	G90 specifies the zero point of the work coordinate system as the origin of the polar coordinate system, from which a radius is measured. G91 specifies the current position as the origin of the polar coordinate system, from which a radius is measured.
IP_	Specifying the addresses of axes constituting the plane selected for the polar coordinate system, and their values First axis : radius of polar coordinate Second axis : radius of polar coordinate

- **Setting the zero point of the workpiece coordinate system as the origin of the polar coordinate system**

Specify the radius (the distance between the zero point and the point) to be programmed with an absolute command. The zero point of the work coordinate system is set as the origin of the polar coordinate system.
When a local coordinate system (G52) is used, the origin of the local coordinate system becomes the center of the polar coordinates.

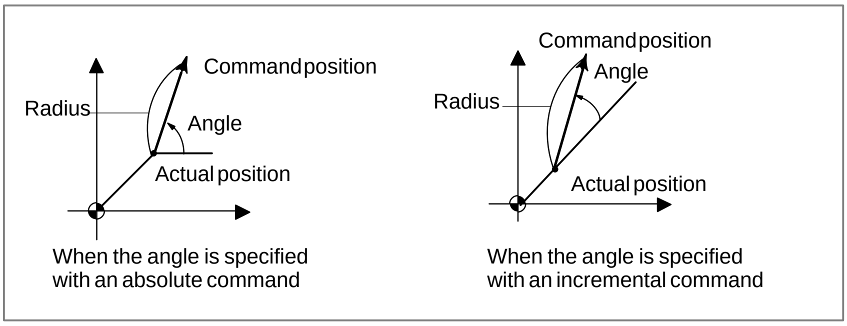


When the angle is specified with an absolute command

When the angle is specified with an incremental command

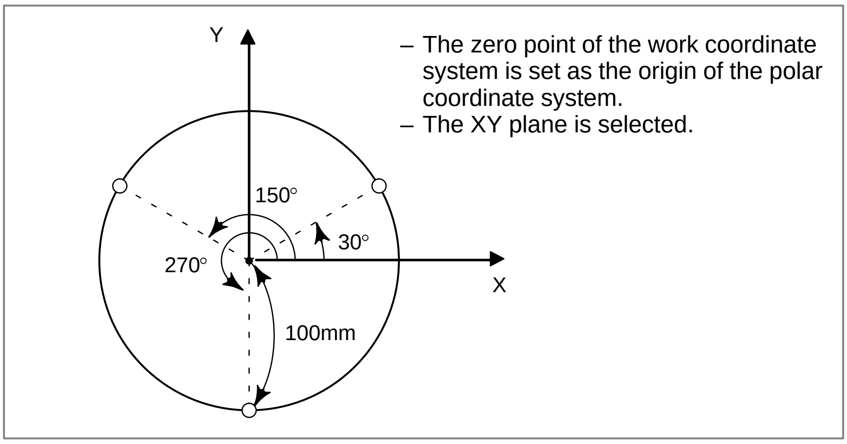
- **Setting the current position as the origin of the polar coordinate system**

Specify the radius (the distance between the current position and the point) to be programmed with an incremental command. The current position is set as the origin of the polar coordinate system.



Examples

Bolt hole circle



- **If the radius and the angle are specified by absolute values**

N1 G17 G90 G16 ;
Polar coordinate command, X–Y plane selection
The origin of the polar coordinate system is the same as that of the workpiece coordinate system.
N2 G00 X100.0 Y30.0;
Radius: 100 mm, angle: 30 deg.
N3 G65 P1000;
Drilling macro call
N4 G00 Y150.0;
Radius: 100 mm, angle: 150 deg.
N5 G65 P1000;
Drilling macro call
N6 G00 Y270.0;
Radius: 100 mm, angle: 270 deg.
N7 G65 P1000;
Drilling macro call
N8 G15;
Polar coordinate command cancel
N9 M30;

- **If the radius is specified by an absolute value and the angle by an incremental value**

N1 G17 G90 G16;
Polar coordinate command, X–Y plane selection]
The origin of the polar coordinate system is the same as that of the workpiece coordinate system.

N2 G00 X100.0 Y30.0;
Radius: 100 mm, angle: 30 deg.
N3 G65 P1000;
Drilling macro call
N4 G91 G00 Y120.0;
Radius: 100 mm, angle: +120 deg.
N5 G65 P1000;
Drilling macro call
N6 G91 G00 Y120.0;
Radius: 100 mm, angle: +120 deg.
N7 G65 P1000;
Drilling macro call
N8 G15;
Polar coordinate command cancel
N9 M30;

<u>Drilling macro</u>

O1000;
N1 G91 G15;
N2 G24;
Piercing
N3 G01 Y−5.0 F2000;
Bolt hole machining
N4 G03 J5.0;;
Bolt hole machining
N5 G90 G16;
N6 M99;

Limitations

- **Specifying a radius in the polar coordinate mode**

In the polar coordinate mode, specify a radius for circular interpolation or helical cutting (G02, G03) with R.

- **Axes that are not considered part of a polar coordinate command in the polar coordinate mode**

Axes specified for the following commands are not considered part of the polar coordinate command:

– Dwell (G04)
– Programmable data input (G10)
– Setting the local coordinate system (G52)
– Converting the workpiece coordinate system (G92)
– Selecting the machine coordinate system (G53)
– Stored stroke check (G22)
– Coordinate system rotation (G68)
– Scaling (G51)

- **Arbitrary angle chamfering and corner rounding**

Arbitrary angle chamfering and corner rounding cannot be specified in polar coordinate mode.

8.3 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. The unit of data input for degrees remains unchanged.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.

⚠ WARNING

G20 and G21 must not be switched during a program.

⚠ CAUTION

1 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.
2 For the first G28 command after switching inch input to metric input or vice versa, operation from the intermediate point is the same as that for manual reference position return. The tool moves from the intermediate point in the direction for reference position return, specified with bit 5 (ZMI) of parameter No. 1006.

NOTE

1 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.
2 The inch and metric input can also be switched using settings (Refer to III–11.4.2).

8.4 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 millimeters inch,or deg. 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

WARNING

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.

NOTE

1 Fractions less than the least input increment are truncated.
Examples:
X1.23456; 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.

2 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; P/S alarm (No.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 AUXILIARY FUNCTION

General

There are two types of auxiliary functions ; miscellaneous function (M code) for specifying program end and so on, and secondary auxiliary function (B code) for specifying index table positioning.

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.

9.1 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, M99,M198 or called subprogram(Parameter No.6071 to 6079), or called custom macro (Parameter No.6080 to 6089). 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) or bit 4 of parameter 3404 (M30) can be used to disable M02, 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 **(II– 10.3)** 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 **(II–10.3)** for details.

- **M198 (Calling a subprogram)**

This code is used to call a subprogram of a file in the external input/output function. See the description of the subprogram call function (III–4.7) for details.

> **NOTE**
> The block following M00, M01, M02, or M30 is not pre–read (buffered). 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.

9.2 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 ; : : : : :

9.3 M CODE GROUP CHECK FUNCTION

The M code group check function checks if a combination of multiple M codes (up to three M codes) contained in a block is correct.
This function has two purposes. One is to detect if any of the multiple M codes specified in a block include an M code that must be specified alone. The other purpose is to detect if any of the multiple M codes specified in a block include M codes that belong to the same group. In either of these cases, P/S alarm No. 5016 is issued.
For details on group data setting, refer to the manual available from the machine tool builder.

Explanations

- **M code setting**

Up to 500 M codes can be specified. In general, M0 to M99 are always specified. M codes from M100 and up are optional.

- **Group numbers**

Group numbers can be set from 0 to 127. Note, however, that 0 and 1 have special meanings. Group number 0 represents M codes that need not be checked. Group number 1 represents M codes that must be specified alone.

9.4 THE SECOND AUXILIARY FUNCTIONS (B CODES)

After a value that follows address B is issued, the code and strobe signals are output. The code is preserved until another B code is issued. The machine uses it to index the rotation axis. Each block can contain only one B code. Setting parameter No. 3460 enables using A, C, V, or W in place of address B. Note, however, that they cannot be used together with the control axis address.
Refer to the manual issued by the machine tool builder for details.

Explanations

- **Valid data range**

0 to 99999999

- **Specification**

1. To enable the use of a decimal point, set bit 0 (AUP) of parameter No.3450 to 1.

Command	Output value
B10.	10000
B10	10

2. Use bit 0 (DPI) of parameter No. 3401 to specify whether the magnification for B output will be $\times 1000$ or $\times 1$ when a decimal point is omitted.

	Command	Output value
DPI=1	B1	1000
DPI=0	B1	1

3. Use bit 0 (AUX) of parameter No. 3405 to specify whether the magnification for B output will be $\times 1000$ or $\times 10000$ when a decimal point is omitted for the inch Input system
(only when DPI=1).

	Command	Output value
AUX=1	B1	10000
AUX=0	B1	1000

Limitations

An address (B or one specified using parameter No. 3460) used with the second auxiliary function cannot be used as the name (parameter No. 1020) of the control axis.

10 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.

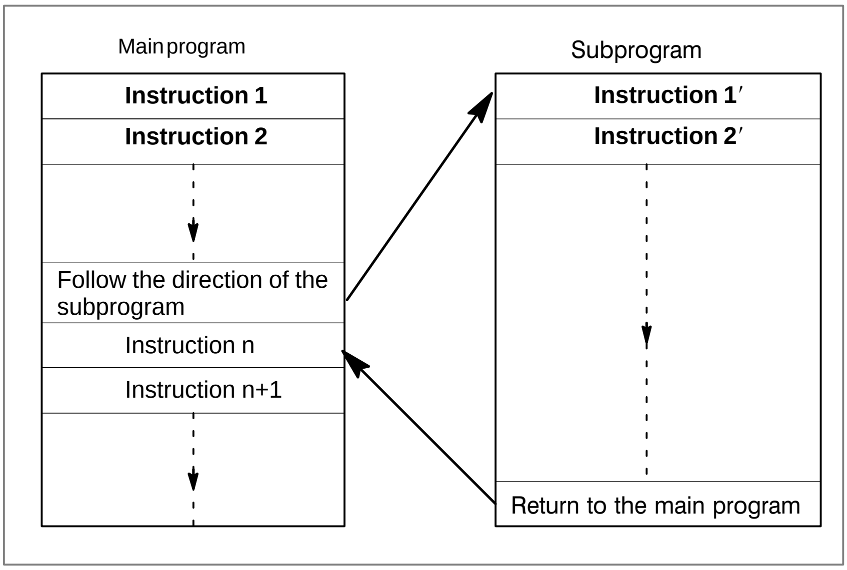


Fig.10 (a) Main program and Subprogram

The CNC memory can hold up to 400 main programs and subprograms (63 as standard). A main program can be selected from the stored main programs to operate the machine. See III–9.3 or III–10 in OPERATION for the methods of registering and selecting programs.

• Program components

A program consists of the following components:

Table 10 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

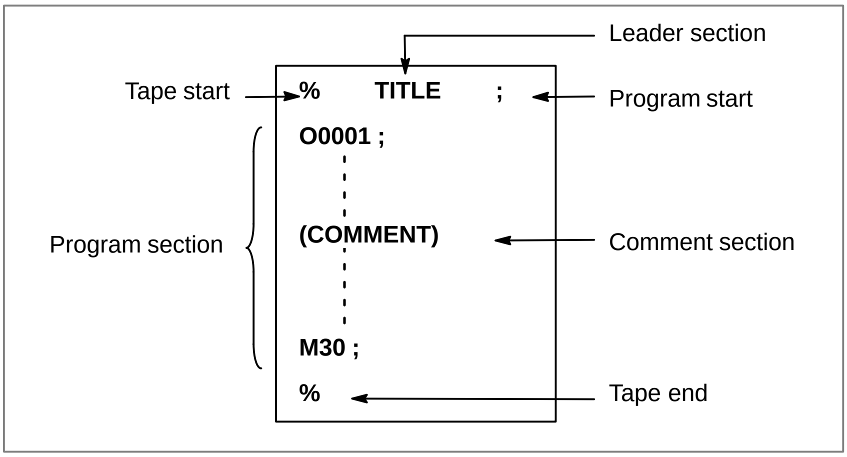


Fig.10 (b) Program configuration

• 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.

<u>**Program section configuration**</u>	<u>**Program section**</u>
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 assist gas on/off command.Specifying a slash (/) at the start of a block disables the execution of some blocks (see “optional block skip” in II–10.2).

10.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See II–10.2 for a program section.

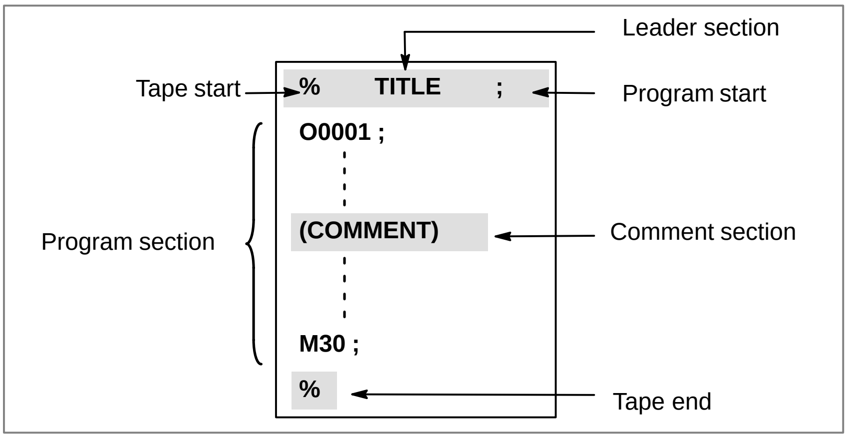


Fig.10.1 Program configuration

Explanations

- **Tape start**

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

Table 10.1 (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 10.1 (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.

Table 10.1 (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 data in memory is output on external I/O device(See III–8), the comment sections 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.
During memory operation or DNC operation, all comment sections are ignored.
The TV check function can be used for a comment section by setting parameter CTV (bit 1 of No. 0100).

⚠ **CAUTION**

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.

NOTE

1 If only a control–in code is read with no matching control–out code, the read control–in code is ignored.
2 The EOB code cannot be used in a comment.

- **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 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 P/S alarm (No. 5010) is occurred.

Table 10.1 (d) Code of a tape end

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

10.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See II–10.1 for program components other than program sections.

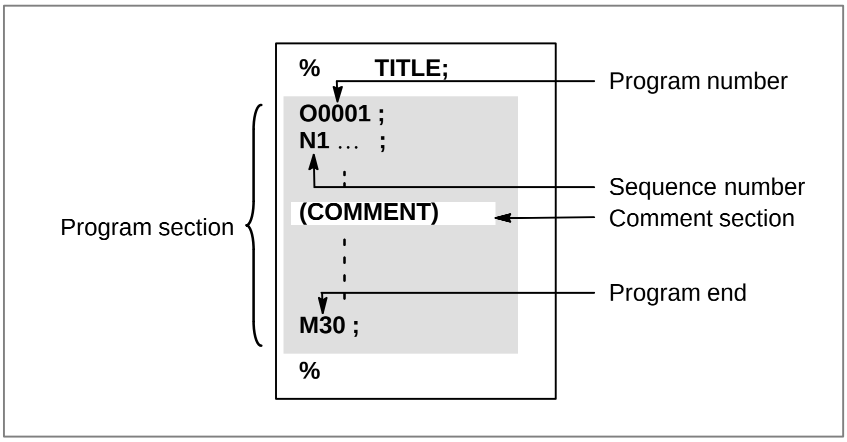


Fig.10.2 (a) Program configuration

- **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.
If there is no program number or sequence number at the start of a program, a program number must be specified using the MDI panel when the program is stored in memory (See III–8.4 or III–10.1)

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 EOB of end of block code.

Table 10.2 (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.

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

Fig.10.2 (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 P/S alarm (No.002) is output. No TV check is made only for those parts that are skipped by the label skip function. Bit 1 (CTV) of parameter No. 0100 is used to specify whether comments enclosed in parentheses are counted as characters during TV check. The TV check function can be enabled or disabled by setting on the MDI unit (See III–11.4.2.).

- **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 10.2 (b) indicates the usable addresses and their meanings.
The same address may have different meanings, depending on the preparatory function specification.

Table 10.2 (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,A,B,C, U,V,W	Coordinate axis move command
	R	Arc radius
	I, J, K	Coordinate values of arc center
Feed function	F	Specifies feedrate.
Laser function	S	Output power
	P	Pulse frequency
	Q	Pulse duty
	R	Piercing time
	E	Machining condition number
Miscellaneous function	M	Specifies on/off controll on machine tool.
	B	Table indeximg, etc.
Offset number	D,H	Specifies offset number.
Dwell	P,X	Specifies dwell time.
Program number designation	P	Specifies subprogram number.
Number of repetitions	P	Number of subprogram repetitions

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

N_	G_	X_ Y_	F_	S_	M_	;
Sequence number	Preparatory function	Dimension word	Feed–function	Laser function	Miscellaneous function	

Fig.10.2 (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 10.2 (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, U, V, W, A, B, C, I, J, K, R,	±99999.999mm	±9999.9999inch
Feed per minute	F	1–240000mm/min	0.01–9600.00 inch/min
Laser function	S	0 to 7000 W	0 to 7000 W
	P	5 to 2000 Hz	5 to 2000 Hz
	Q	0 to 100 %	0 to 100 %
	R	0 to 99999.999 seconds	0 to 99999.999 seconds
	E	1 to 30, 101 to 110	1 to 30, 101 to 110
Auxiliary function	M	0–99999999	0–99999999
	B	0–99999999	0–99999999
Offset number	H, D	0–400	0–400
Dwell	X, P	0–99999.999s	0–99999.999s
Designation of a program number	P	1–9999	1–9999
Number of repetitions	P	1–9999	1–9999

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

- **Optional block skip**

When a slash followed by a number (/n (n=1 to 9)) is specified at the head of a block, and optional block skip switch n on the machine operator panel is set to on, the information contained in the block for which /n corresponding to switch number n is specified is ignored in DNC operation or memory operation.

When optional block skip switch n is set to off, the information contained in the block for which /n is specified is valid. This means that the operator can determine whether to skip the block containing /n.

Number 1 for /1 can be omitted. However, when two or more optional block skip switches are used for one block, number 1 for /1 cannot be omitted.

Example)

```
(Incorrect)      (Correct)
//3 G00X10.0;    /1/3 G00X10.0;
```

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.

Depending on the machine tool, all optional block skip switches (1 to 9) may not be usable. Refer to manuals of the machine tool builder to find which switches are usable.

WARNING

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 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.

NOTE

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.

- **Program end**

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

Table 10.2 (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.

WARNING

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 "Optional block skip".)

10.3 SUBPROGRAM (M98, M99)

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○○○ ○○○○ ;

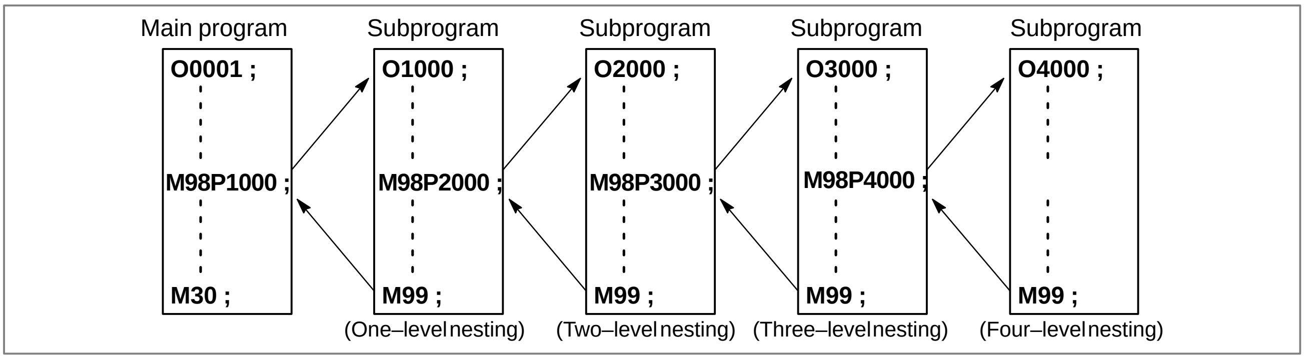
↑ **Number of times the subprogram is called repeatedly**

↑ **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 999 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 III–10 for the method of registering a subprogram.

> **NOTE**
> 1 The M98 and M99 code signal and strobe signal 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

Main program	1 2 3	Subprogram
N0010 0 ;		**O1010 0 ;**
N0020 0 ;		**N1020 0 ;**
N0030 M98 P21010 ;		**N1030 0 ;**
N0040 0 ;		**N1040 0 ;**
N0050 M98 P1010 ;		**N1050 0 ;**
N0060 0 ;		**N1060 0 M99 ;**

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 ... ;	O0010 ... ;
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/M99Pn ; 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 ... ;
N0040 ... ;
N0050 ... ;
/ N0060 M99 P0030 ;
N0070 ... ;
N0080 M02 ;
```

Optional block skip OFF

Optional block skip ON

- **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 III–9.3 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 M99Pn 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 ;          Optional block skip
  N1050 M99 P1020 ;    ON
```

10.4 8–DIGIT PROGRAM NUMBER

The 8–digit program number function enables specification of program numbers with eight digits following address O (O00000001 to O99999999).

Explanations

- **Disabling editing of programs**

Editing of subprograms O00008000 to O00008999, O00009000 to O00009999, O80000000 to O89999999, and O90000000 to O99999999 can be disabled.

Parameter	Program numbers for which editing is disabled
NE8(No.3202#0)	O00008000 to O00008999
NE9(No.3202#4)	O00009000 to O00009999
P8E(No.3204#3)	O80000000 to O89999999
P9E(No.3204#4)	O90000000 to O99999999

NOTE
When a wrong password has been specified for the password function (see III–9.9), the settings of NE9 (bit 3 of parameter No. 3202) and P9E (bit 4 of parameter No. 3204) cannot be changed.

- **File name**

For program punch by specifying a range, files are named as follows:
When punching by specifying O00000001 and O00123456:
"O00000001–G"
When punching by specifying O12345678 and O45678900:
"O12345678–G"

- **Special programs**

Special subprogram numbers can be changed by using bit 5 (SPR) of parameter No. 3204.

1) Macro call using G code

Parameter used to specify G code	Program number	
	When SPR = 0	When SPR = 1
No.6050	O00009010	O90009010
No.6051	O00009011	O90009011
No.6052	O00009012	O90009012
No.6053	O00009013	O90009013
No.6054	O00009014	O90009014
No.6055	O00009015	O90009015
No.6056	O00009016	O90009016
No.6057	O00009017	O90009017
No.6058	O00009018	O90009018
No.6059	O00009019	O90009019

2) Macro call using M code

Parameter used to specify M code	Program number	
	When SPR = 0	When SPR = 1
No.6080	O00009020	O90009020
No.6081	O00009021	O90009021
No.6082	O00009022	O90009022
No.6083	O00009023	O90009023
No.6084	O00009024	O90009024
No.6085	O00009025	O90009025
No.6086	O00009026	O90009026
No.6087	O00009027	O90009027
No.6088	O00009028	O90009028
No.6089	O00009029	O90009029

3) Subprogram call using M code

Parameter used to specify M code	Program number	
	When SPR = 0	When SPR = 1
No.6071	O00009001	O90009001
No.6072	O00009002	O90009002
No.6073	O00009003	O90009003
No.6074	O00009004	O90009004
No.6075	O00009005	O90009005
No.6076	O00009006	O90009006
No.6077	O00009007	O90009007
No.6078	O00009008	O90009008
No.6079	O00009009	O90009009

4) Macro call using T code

Parameter used to specify T code	Program number	
	When SPR = 0	When SPR = 1
TCS(No.6001#5)	O00009000	O90009000

5) Macro call using ASCII code

Parameter used to specify ASCII code	Program number	
	When SPR = 0	When SPR = 1
No.6090	O00009004	O90009004
No.6091	O00009005	O90009005

- **External program number search**

External input signals can be used to search for a program number. A program stored in CNC memory can be selected by externally inputting a program number, between 1 and 99999999, to the CNC. For details, refer to the appropriate manual supplied from the machine tool builder.

Limitations

- **Subprogram call**

This function disables subprogram call unless FS15 tape format (see II–15) is used. This restriction also applies to calling a program in external I/O devices (M198).

(Example)

M98 P12345678 ;

Subprogram number only. The repetition count is not included.

- **DNC**

The eight–digit program number cannot be used in DNC1, DNC2, Ethernet, data server, open CNC, and graphic conversation functions.

11 FUNCTIONS TO SIMPLIFY PROGRAMMING

General

This chapter explains the following items:

11.1 OPTIONAL ANGLE CHAMFERING AND CORNER ROUNDING

Chamfering and corner rounding blocks can be inserted automatically between the following:

·Between linear interpolation and linear interpolation blocks
·Between linear interpolation and circular interpolation blocks
·Between circular interpolation and linear interpolation blocks
·Between circular interpolation and circular interpolationblocks

Format

, C_	**Chamfering**
, R_	**Corner R**

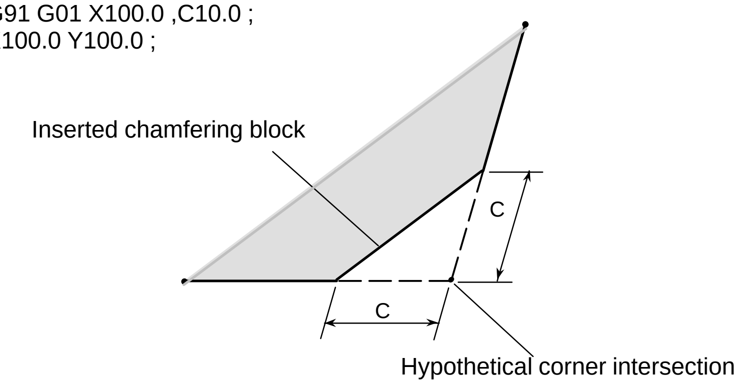
Explanations

When the above specification is added to the end of a block that specifies linear interpolation (G01) or circular interpolation (G02 or G03), a chamfering or corner rounding block is inserted.
Blocks specifying chamfering and corner rounding can be specified consecutively.

• Chamfering

After C, specify the distance from the virtual corner point to the start and end points. The virtual corner point is the corner point that would exist if chamfering were not performed.

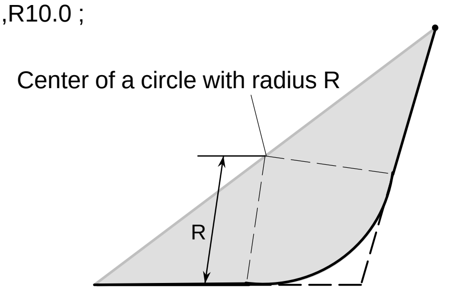
(1) G91 G01 X100.0 ,C10.0 ;
(2) X100.0 Y100.0 ;



• Corner R

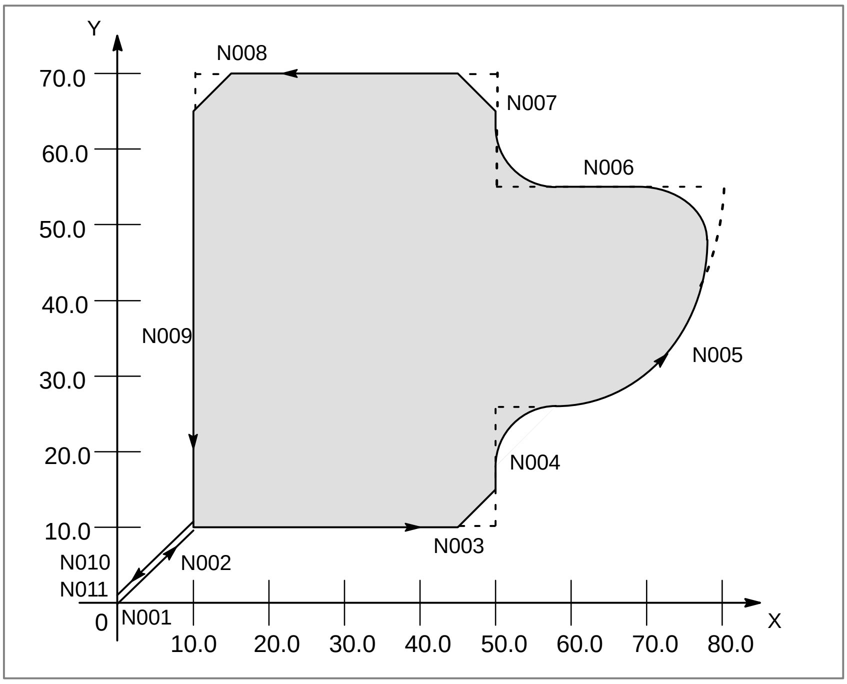
After R, specify the radius for corner rounding.

(1) G91 G01 X100.0 ,R10.0 ;
(2) X100.0 Y100.0 ;



Examples

```
N001 G92 G90 X0 Y0 ;
N002 G00 X10.0 Y10.0 ;
N003 G01 X50.0 F10.0 ,C5.0 ;
N004 Y25.0 ,R8.0 ;
N005 G03 X80.0 Y50.0 R30.0 ,R8.0 ;
N006 G01 X50.0 ,R8.0 ;
N007 Y70.0 ,C5.0 ;
N008 X10.0 ,C5.0 ;
N009 Y10.0 ;
N010 G00 X0 Y0 ;
N011 M0 ;
```



Restrictions

- **Plane selection**

Chamfering and corner rounding can be performed only in the plane specified by plane selection (G17, G18, or G19). These functions cannot be performed for parallel axes.

- **Next block**

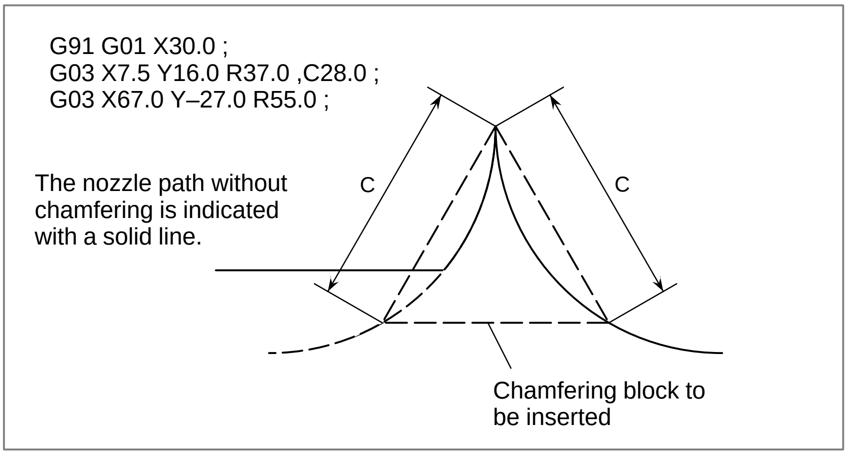
A block specifying chamfering or corner rounding must be followed by a block that specifies a move command using linear interpolation (G01) or circular interpolation (G02 or G03). If the next block does not contain these specifications, P/S alarm No. 052 is issued.

- **Plane switching**

A chamfering or corner rounding block can be inserted only for move commands which are performed in the same plane. In a block that comes immediately after plane switching (G17, G18, or G19 is specified), neither chamfering nor corner rounding can be specified.

- **Exceeding the move range**

If the inserted chamfering or corner rounding block causes the tool to go beyond the original interpolation move range, P/S alarm No.055 is issued.



If no arc end point is specified with a G02 or G03 command, the machine tool does not operate properly.

- **Coordinate system**

In a block that comes immediately after the coordinate system is changed (G92, or G52 to G59) or a return to the reference position (G28 to G30) is specified, neither chamfering nor corner rounding can be specified.

- **Travel distance 0**

When two linear interpolation operations are performed, the chamfering or corner rounding block is regarded as having a travel distance of zero if the angle between the two straight lines is within ±1°. When linear interpolation and circular interpolation operations are performed, the corner rounding block is regarded as having a travel distance of zero if the angle between the straight line and the tangent to the arc at the intersection is within ±1°. When two circular interpolation operations are performed, the corner rounding block is regarded as having a travel distance of zero if the angle between the tangents to the arcs at the intersection is within ±1°.

- **Unavailable G codes**

The following G codes cannot be used in a block that specifies chamfering or corner rounding. They also cannot be used between chamfering and corner rounding blocks that define a continuous figure.
·G codes of group 00 (except G04)
·G68 of group 16

- **DNC operation**

Chamfering and corner R are not used in the DNC operation.

11.2 FIGURE COPY (G72.1, G72.2)

Machining can be repeated after moving or rotating the figure using a subprogram.

Format

- **Rotational copy**

Xp–Yp plane (specified by G17) : G72.1 P_ L_ Xp_ Yp_ R_ ;
Zp–Xp plane (specified by G18) : G72.1 P_ L_ Zp_ Xp_ R_ ;
Yp–Zp plane (specified by G19) : G72.1 P_ L_ Yp_ Zp_ R ;_

P :Subprogram number
L :Number of times the operation is repeated
Xp :Center of rotation on the Xp axis
(Xp : X–axis or an axis parallel to the X–axis)
Yp :Center of rotation on the Yp axis
(Yp: Y–axis or an axis parallel to the Y–axis)
Zp :Center of rotation on the Zp axis
(Zp: Z–axis or an axis parallel to the Z–axis)
R :Angular displacement
(A positive value indicates a counterclockwise angular displacement. Specify an incremental value.)

Specify a plane selection command (G17, G18, or G19) to select the plane on which the rotational copy is made.

- **Linear copy**

Xp–Yp plane (specified by G17) : G72.2 P_ L_ I_ J_ ;
Zp–Xp plane (specified by G18) : G72.2 P_ L_ K_ I_ ;
Yp–Zp plane (specified by G19) : G72.2 P_ L_ J_ K_;

P :Subprogram number
L :Number of times the operation is repeated
I :Shift along the Xp axis
J :Shift along the Yp axis
K :Shift along the Zp axis

Specify a plane selection command (G17, G18, or G19) to select the plane on which the linear copy is made.

Explanations

- **First block of the subprogram**

Always specify a move command in the first block of a subprogram that performs a rotational or linear copy. If the first block contains only the program number such as O1234; and does not have a move command, movement may stop at the start point of the figure made by the n–th (n = 1, 2, 3, ...) copying.

<u>Specify the first move command in the absolute mode.</u>

(Example of an incorrect program)

```
O1234 ;
G00 G90 X100.0 Y200.0 ;
.... ;
.... ;
M99 ;
```

(Example of a correct program)

```
O1000 G00 G90 X100.0 Y200.0 ;
.... ;
.... ;
M99 ;
```

- **Combination of rotational and linear copying**

The linear copy command can be specified in a subprogram for a rotational copy. Also, the rotational copy command can be specified in a subprogram for a linear copy.

- **Subprogram calling**

In a subprogram for rotational or linear copying, M98 for calling another subprogram or G65 for calling a macro can be specified.

- **Specifying the center of rotation**

The center of rotation specified with G72.1 is processed as an absolute position even in the incremental mode.

- **Specifying address**

In a block with G72.1, addresses other than P, L, Xp, Yp, Zp, or R are ignored. The subprogram number (P), coordinates of the center of rotation (Xp, Yp, Zp), and angular displacement (R) must be specified.
In a block with G72.2, addresses other than P, L, I, J, or K are ignored. The subprogram number (P) and shift (I, J, K) must be specified.

- **Address P**

If the subprogram number specified with P is not found, P/S alarm No. 078 occurs. If P is not specified, P/S alarm No. 076 occurs.

- **Address L**

If L is omitted, the repetition count is assumed to be 1 and the sub–program is called only once.

- **Increment in angular displacement or shift**

In a block with G72.1, an increment in angular displacement is specified with address R. The angular displacement of the figure made by the n–th rotation is calculated as follows : $R \times (n - 1)$.
In a block with G72.2, an increment in shift is specified with addresses I, J, and K. The shift of the figure made by the n–th movement is calculated as follows : (Programmed shift) $\times$ $(n - 1)$.

- **Nesting level of a subprogram**

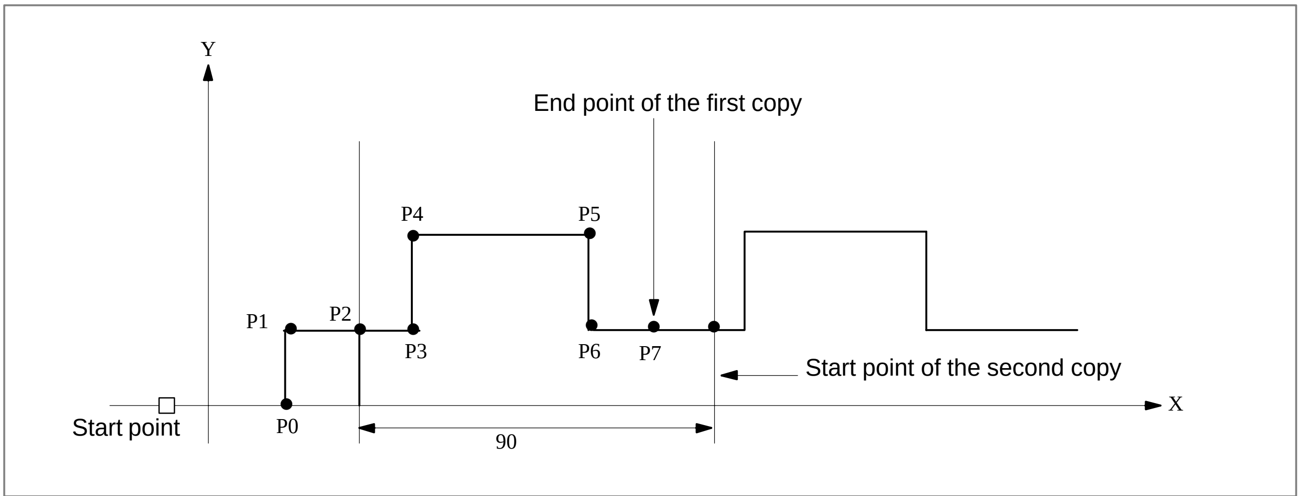
If a subprogram is called by G72.1 or G72.2, the nesting level is increased by one in the same manner as when M98 is specified.

- **Block end position**

The coordinates of a figure moved rotationally or linearly (block end position) can be read from #5001 and subsequent system variables of the custom macro of rotational or linear copy.

- **Disagreement between end point and start point**

If the end point of the figure made by the n–th copy does not agree with the start point of the figure to be made by the next (n + 1) copy, the figure is moved from the end point to the start point, then copying is started. (Generally, this disagreement occurs if an incorrect angular displacement or shift is specified.)



Main program

```
O1000 ;
N10 G92 X–20.0 Y0 ;
N20 G00 G90 X0 Y0 ;
N30 G01 G17 G41 X20. Y0 D01 F10 ;          (P0)
N40 Y20. ;                                 (P1)
N50 X30. ;                                 (P2)
N60 G72.2 P2000 L3 I90. J0 ;
```

Although a shift of 70 mm was required, I90.0 was specified instead of I70.0. Since an incorrect shift was specified, the end point of the figure made by the n–th copy disagrees with the start point of the figure to be made by the next (n + 1) copy.

Subprogram

```
O2000 G90 G01 X40. ;                       (P3)
N100 Y40. ;                                (P4)
N200 G01 X80. ;                            (P5)
N300 G01 Y20. ;                            (P6)
N400 X100. ;                               (P7)
N500 M99;
```

Limitations

- **Specifying two or more commands to copy a figure**

G72.1 cannot be specified more than once in a subprogram for making a rotational copy (If this is attempted, P/S alarm No.160 will occur). G72.2 cannot be specified more than once in a subprogram for making a linear copy (If this is attempted, P/S alarm No. 161 will occur).

- **Commands that must not be specified**

Within a program that performs a rotational or linear copy, the following must not be specified:
·Command for changing the selected plane (G17 to G19)
·Command for specifying polar coordinates
·Reference position return command
·Coordinate system rotation, scaling, programmable mirror image
The command for rotational or linear copying can be specified after a command for coordinate system rotation, scaling, or programmable mirror image is executed.

- **Modes that must not be selected**

The figure cannot be copied during chamfering, corner rounding, or tool offset.

- **Unit system**

The two axes of the plane for copying a figure must have an identical unit system.

- **Single block**

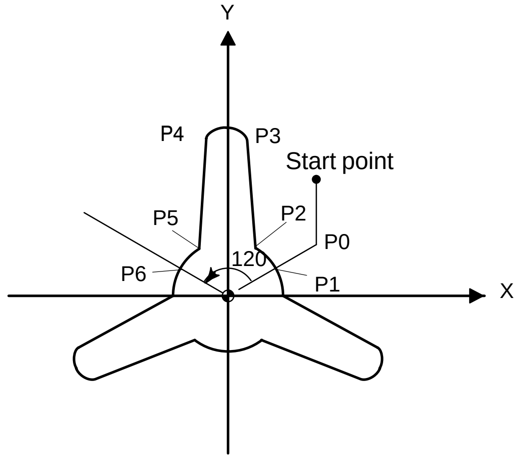
Single–block stops are not performed in a block with G721.1 or G72.2.

- **Specifying cutter compensation and the workpiece coordinate system**

In a subprogram for copying a figure, the G code for cutter compensation C or compensation amount (H or D code) cannot be changed. G92 and G54 to G59 cannot be changed either. Those codes must be specified before figure copying is started.

Examples

- **Rotational copy**



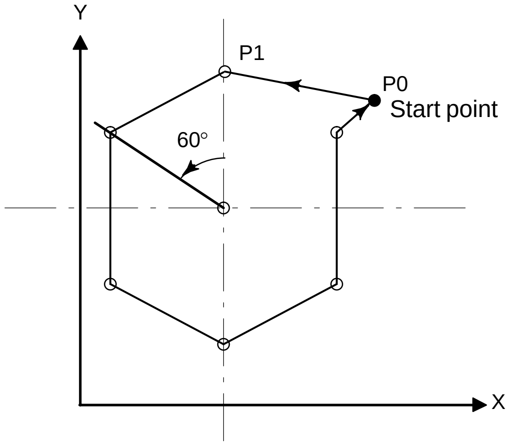
Main program

```
O1000 ;
N10 G92 X40.0 Y50.0 ;
N20 G00 G90 X_ Y_ ;                     (P0)
N30 G01 G17 G41 X_ Y_ D01 F10 ;         (P1)
N40 G72.1 P2000 L3 X0 Y0 R120.0 ;
N50 G40 G01 X_ Y_ I_ J_ ;               (P0)
N60 G00 X40.0 Y50.0 ;
N70 M30 ;
```

Sub program

```
O2000 G03 X_ Y_ R30.0 ;                 (P2)
N100 G01 X_ Y_ ;                        (P3)
N200 G03 X_ Y_ R10.0 ;                  (P4)
N300 G01 X_ Y_ ;                        (P5)
N400 G03 X_ Y_ R30.0 ;                  (P6)
N500 M99;
```

- **Rotational copy (spot boring)**



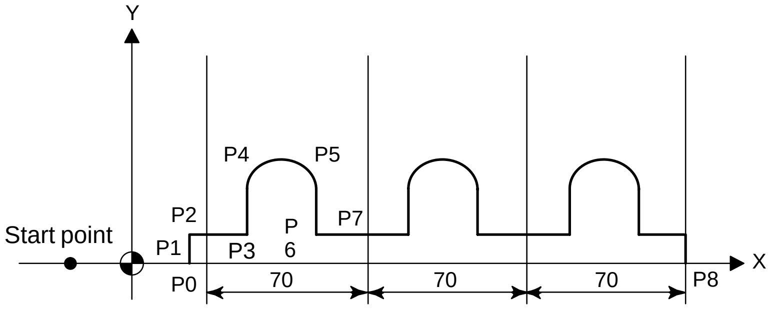
Main program

```
O3000 ;
N10 G92 G17 X80.0 Y50.0 ;                    (P0)
N20 G72.1 P4000 L6 X0 Y0 R60.0 ;
N30 G80 G00 X80.0 Y50.0 ;                    (P0)
N40 M30 ;
```

Subprogram

```
O4000 N100 G90 G81 X_ Y_ R_ Z_ F_ ;          (P1)
N200 M99 ;
```

- **Linear copy**



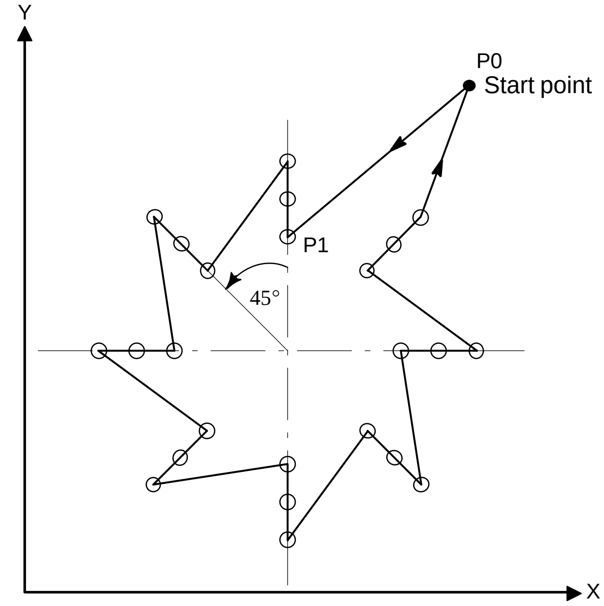
Main program

```
O1000 ;
N10 G92 X–20.0 Y0 ;
N20 G00 G90 X0 Y0 ;
N30 G01 G17 G41 X_ Y_ D01 F10 ;            (P0)
N40 Y_ ;                                   (P1)
N50 X_ ;                                   (P2)
N60 G72.2 P2000 L3 I70.0 J0 ;
N70 X_ Y_ ;                                (P8)
N80 X0 ;
N90 G00 G40 X–20.0 Y0 ;
N100 M30 ;
```

Subprogram

```
O2000 G90 G01 X_ ;                         (P3)
N100 Y_ ;                                  (P4)
N200 G02 X_ I_ ;                           (P5)
N300 G01 Y_ ;                              (P6)
N400 X_ ;                                  (P7)
N500 M99 ;
```

- **Combination of rotational copying and linear copying (bolt hole circle)**



Main program

```
O1000 ;
N10 G92 G17 X100.0 Y80.0 ;                    (P0)
N20 G72.1 P2000 X0 Y0 L8 R45.0 ;
N30 G80 G00 X100.0 Y80.0 ;                    (P0)
N40 M30 ;
```

Subprogram (rotational copy)

```
O2000 N100 G72.2 P3000 I0 J_ L3 ;
N200 M99 ;
```

Subprogram (linear copy)

```
O3000 N110 G90 G81 X_ Y_ R_ Z_ F_ ;           (P1)
N210 M99 ;
```

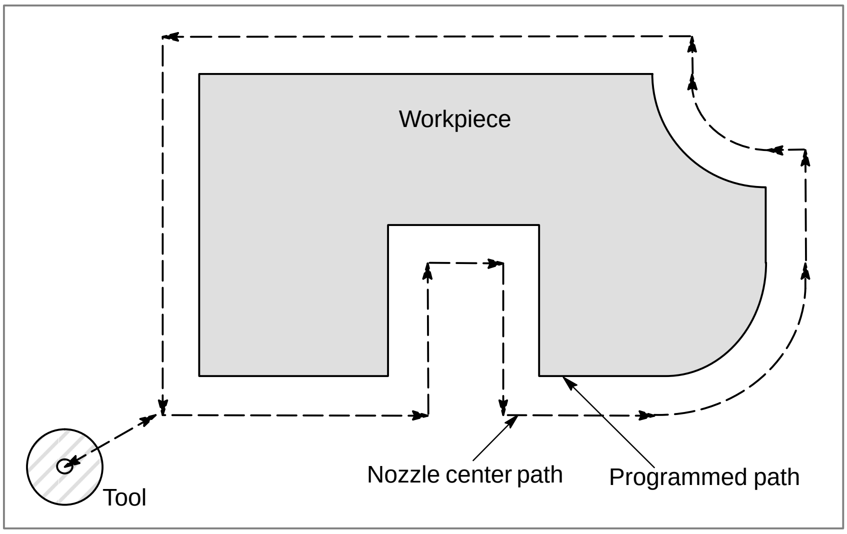
12 COMPENSATION FUNCTION

General

This chapter describes the following compensation functions:

12.1 TOOL OFFSET (G45–G48)

The programmed travel distance of the nozzle can be increased or decreased by a specified tool offset value or by twice the offset value.
The tool offset function can also be applied to an additional axis.



Format

G45 IP_D_ ;	Increase the travel distance by the tool offset value
G46 IP_D_ ;	Decrease the travel distance by the tool offset value
G47 IP_D_ ;	Increase the travel distance by twice the tool offset value
G48 IP_D_ ;	Decrease the travel distance by twice the tool offset value

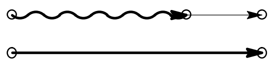
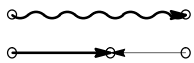
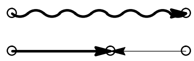
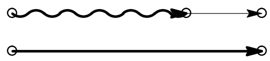
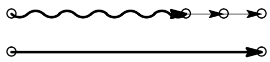
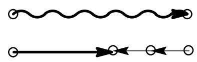
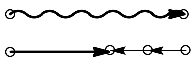
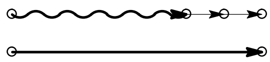
G45 to G48 : One–shot G code for increasing or decreasing the travel distance
IP : Command for moving the tool
D : Code for specifying the tool offset value

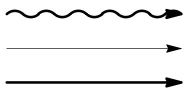
Explanations

• Increase and decrease

As shown in Table 12.1 (a), the travel distance of the nozzle is increased or decreased by the specified tool offset value.
In the absolute mode, the travel distance is increased or decreased as the nozzle is moved from the end position of the previous block to the position specified by the block containing G45 to G48.

Table 12.1 (a) Increase and decrease of the travel distance

G code	When a positive tool offset value is specified	When a negative tool offset value is specified
G45	 Start position End position	 Start position End position
G46	 Start position End position	 Start position End position
G47	 Start position End position	 Start position End position
G48	 Start position End position	 Start position End position



If a move command with a travel distance of zero is specified in the incremental command (G91) mode, the nozzle is moved by the distance corresponding to the specified tool offset value.
If a move command with a travel distance of zero is specified in the absolute command (G90) mode, the nozzle is not moved.

• Tool offset value

Once selected by D code, the tool offset value remains unchanged until another tool offset value is selected.
Tool offset values can be set within the following range:

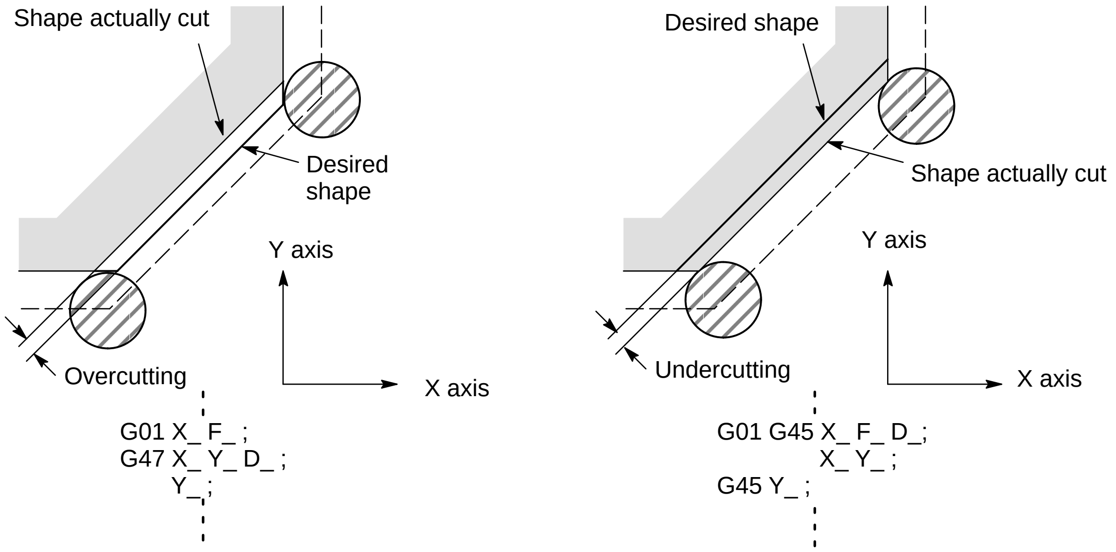
Table 12.1 (b) Range of tool offset values

	Metric input	**inch input**
Tool offset value	0 to ±999.999mm	0 to ±99.9999inch
	0 to ±999.999deg	0 to ±999.999deg

D0 always indicates a tool offset value of zero.

WARNING

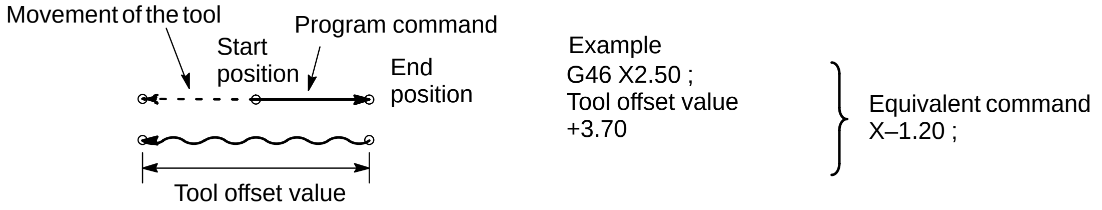
1 When G45 to G48 is specified to n axes (n=1–6) simultaneously in a motion block, offset is applied to all n axes.
When the cutter is offset only for cutter radius or diameter in taper cutting, overcutting or undercutting occurs.
Therefore, use cutter compensation (G40 or G42) shown in II–12.2.



2 G45 to G48 (tool offset) must not be used in the G41 or G42 (cutter compensation) mode.

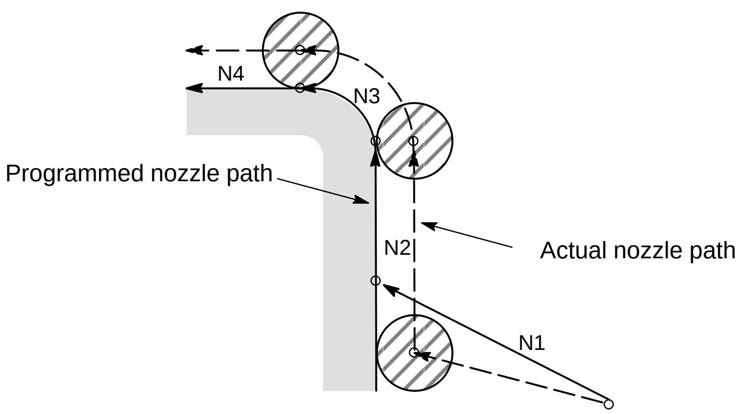
NOTE

1 When the specified direction is reversed by decrease as shown in the figure below, the tool moves in the opposite direction.



2 Tool offset can be applied to circular interpolation (G02, G03) with the G45 to G48 commands only for 1/4 and 3/4 circles using addresses I, J and K by the parameter setting, providing that the coordinate rotation be not specified at the same time. This function is provided for compatibility with the conventional CNC tape without any cutter compensation. The function should not be used when a new CNC program is prepared.

Tool offset for circular interpolation



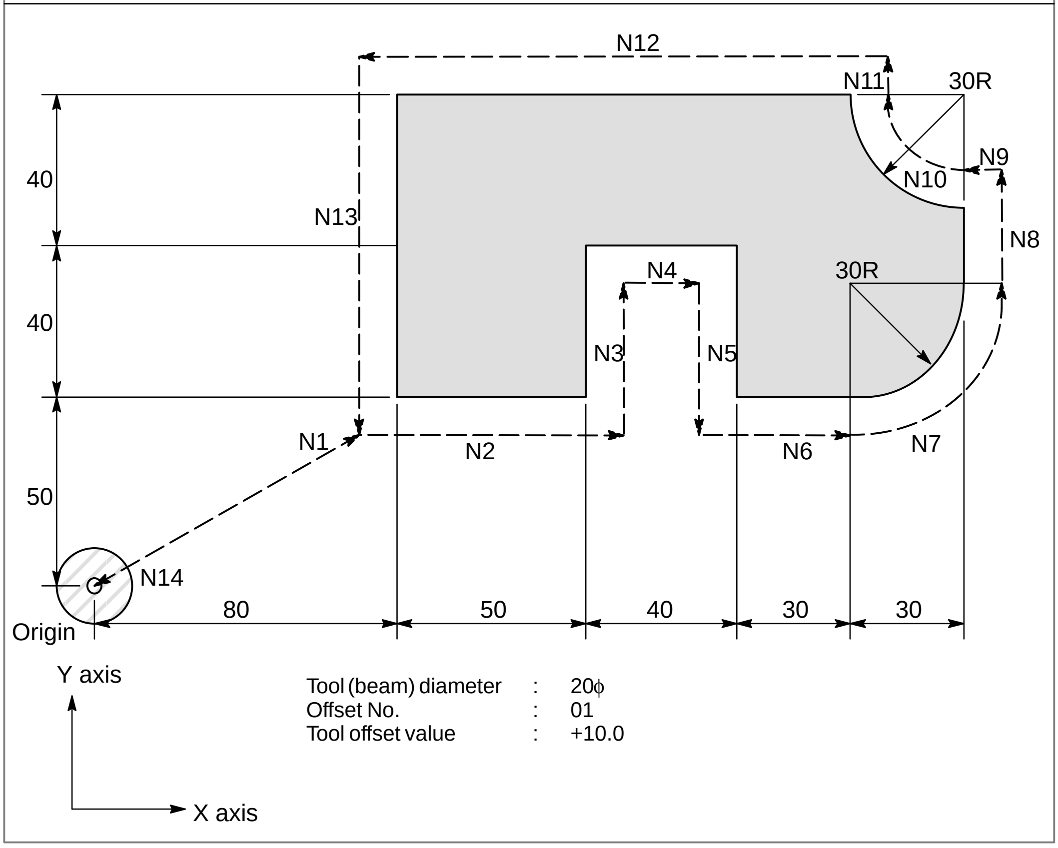
Program

N1 G46 G00 X_ Y_ D_ ;
N2 G45 G01 Y_ F_ ;
N3 G45 G03 X_ Y_ I_ ;
N4 G01 X_ ;

3 D code should be used in tool offset mode (G45 to G48). However, H code can be used by setting the parameter TPH (No. 5001#5) because of compatibility with conventional CNC tape format. The H code must be used under length offset cancel (G49).

Examples

Program using tool offset



Program

N1 G91 G46 G00 X80.0 Y50.0 D01 ;
N2 G47 G01 X50.0 F120.0 ;
N3 Y40.0 ;
N4 G48 X40.0 ;
N5 Y–40.0 ;
N6 G45 X30.0 ;
N7 G45 G03 X30.0 Y30.0 J30.0 ;
N8 G45 G01 Y20.0 ;
N9 G46 X0 ; Decreases toward the positive direction for movement amount “0”. The beam moves in the –X direction by theoffset value.
N10 G46 G02 X–30.0 Y30.0 J30.0 ;
N11 G45 G01 Y0 ; Increase toward the positive direction for movement amount “0”. The beam moves in the +Y direction by the offset value.
N12 G47 X–120.0 ;
N13 G47 Y–80.0 ;
N14 G46 G00 X80.0 Y–50.0 ;

12.2 OVERVIEW OF CUTTER COMPENSATION C (G40–G42)

When the beam is moved, the nozzle path can be shifted by the radius of the nozzle (Fig.12.2 (a)).
To make an offset as large as the radius of the beam, CNC first creates an offset vector with a length equal to the radius of the nozzle (start–up). The offset vector is perpendicular to the nozzle path. The tail of the vector is on the workpiece side and the head positions to the center of the beam.
If a linear interpolation or circular interpolation command is specified after start–up, the nozzle path can be shifted by the length of the offset vector during machining.
To return the nozzle to the start position at the end of machining, cancel the cutter compensation mode.

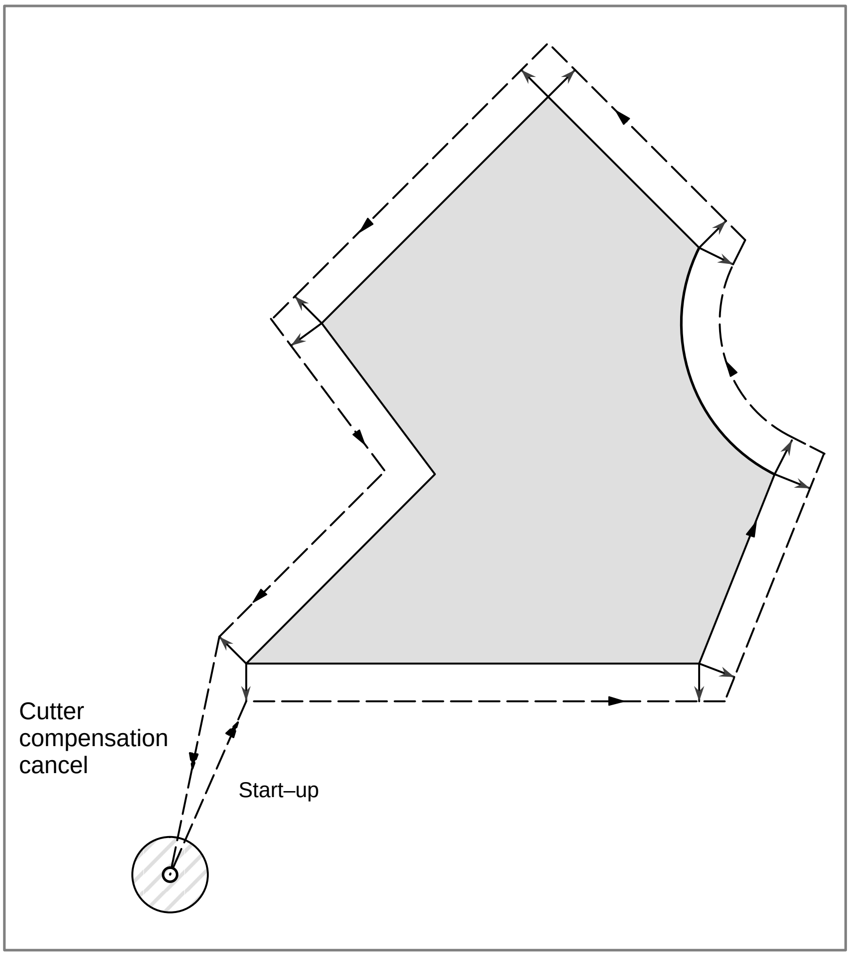


Fig.12.2 (a) Outline of Cutter Compensation C

Format

- **Start up (Cutter 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(1–3digits) (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_
XpYp	**G17 ;**	Xp_Yp_
ZpXp	**G18 ;**	Xp_Zp_
YpZp	**G19 ;**	Yp_Zp_

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 beam 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 nozzle with this command is called start–up.
Specify positioning (G00) or linear interpolation (G01) for start–up. If circular interpolation (G02, G03) is specified, P/S alarm 34 occurs.
When processing the start–up block and subsequent blocks, the CNC prereads two blocks.

- **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 nozzle (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, P/S alarm 37 occurs and the nozzle is stopped.

• Offset mode cancel

In the offset mode, when a block which satisfies any one of the following conditions is executed, the CNC 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 P/S alarm (No. 034) is generated and the nozzle 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.

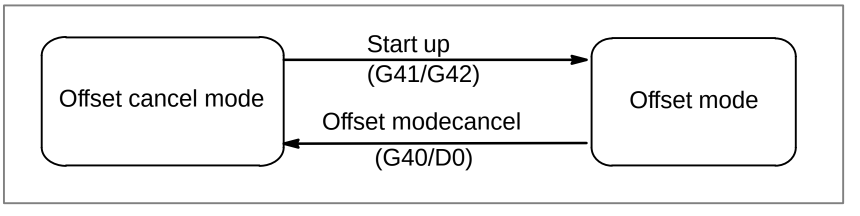


Fig.12.2 (b) Changing the offset mode

• 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.

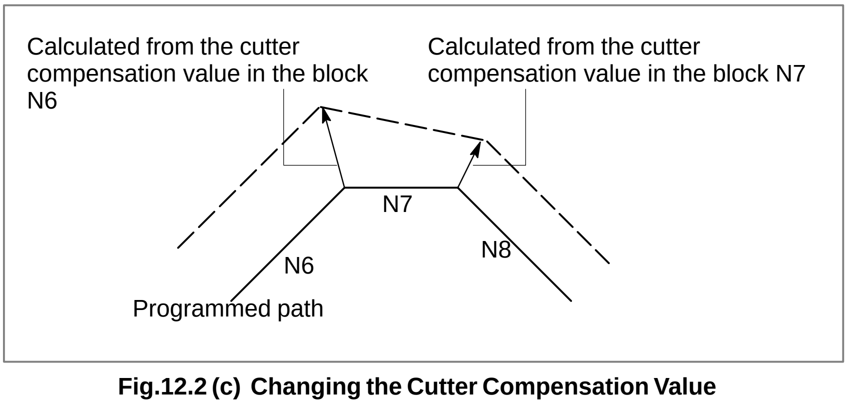


Fig.12.2 (c) Changing the Cutter Compensation Value

• 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 nozzle 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 nozzle 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 II– 12.3.2 and 12.3.4)

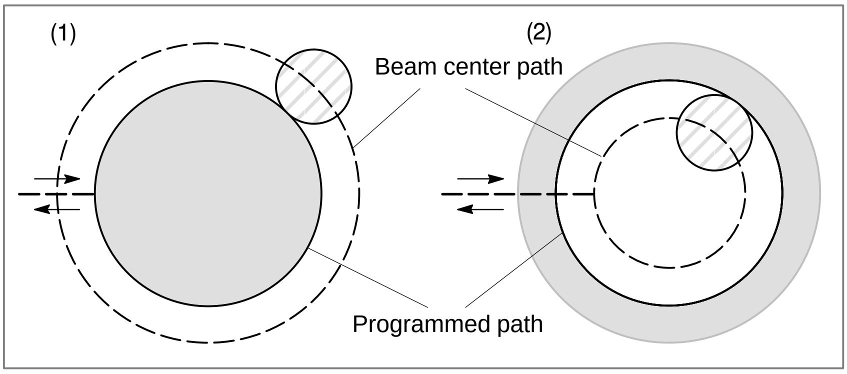


Fig.12.2 (d) Beam Center Paths when Positive and Negative Cutter Compensation Values are Specified

• Cutter compensation value setting

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

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

NOTE
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 3 digits after address D (D code). The D code is valid until another D code is specified. The D code is used to specify the tool offset value as well as the cutter compensation value.

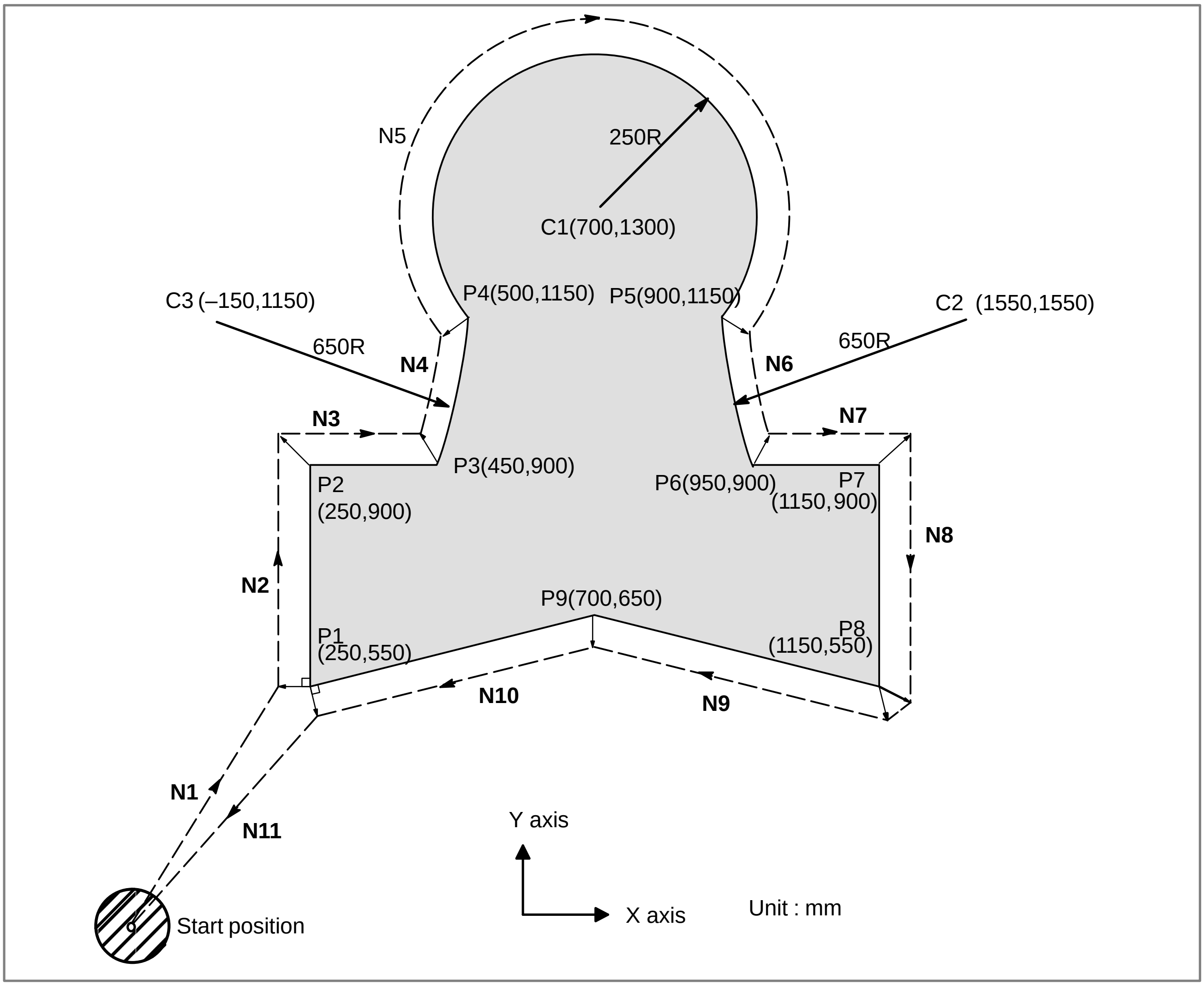
- **Plane selection and vector**

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 beam 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, a P/S alarm (No. 37) is displayed and the machine is stopped.

Limitations

In cutter compensation mode, spatial circular interpolation, spatial corner R insertion, and three–dimensional conversion cannot be performed.

Examples



G92 X0 Y0 Z0 ;	Specifies absolute coordinates. The nozzle 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 nozzle is shifted to the left of the programmed path by the distance specified in D07. In other words the beam path is shifted by the radius of the beam (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 nozzle is returned to the start position (X0, Y0, Z0).

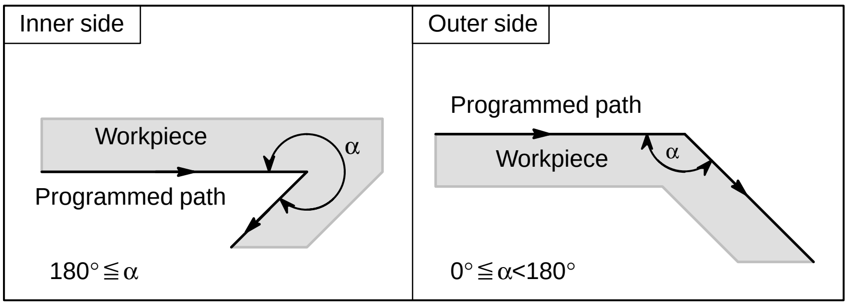
12.3 DETAILS OF CUTTER COMPENSATION C

This section provides a detailed explanation of the movement of the beam for cutter compensation C outlined in Section 12.2.
This section consists of the following subsections:

12.3.1 General

• Inner side and outer side

When an angle of intersection created by beam 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

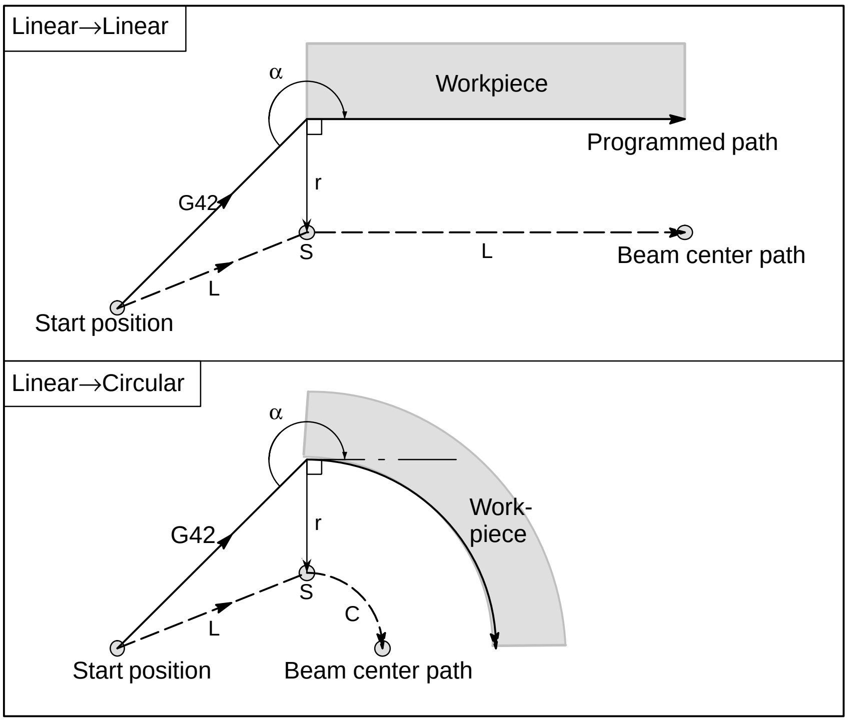
The following symbols are used in subsequent figures:
– ***S*** indicates a position at which a single block is executed once.
– ***SS*** indicates a position at which a single block is executed twice.
– ***SSS*** indicates a position at which a single block is executed three times.
– ***L*** indicates that the nozzle moves along a straight line.
– ***C*** indicates that the nozzle 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 beam.

12.3.2 Nozzle Movement in Start–up

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

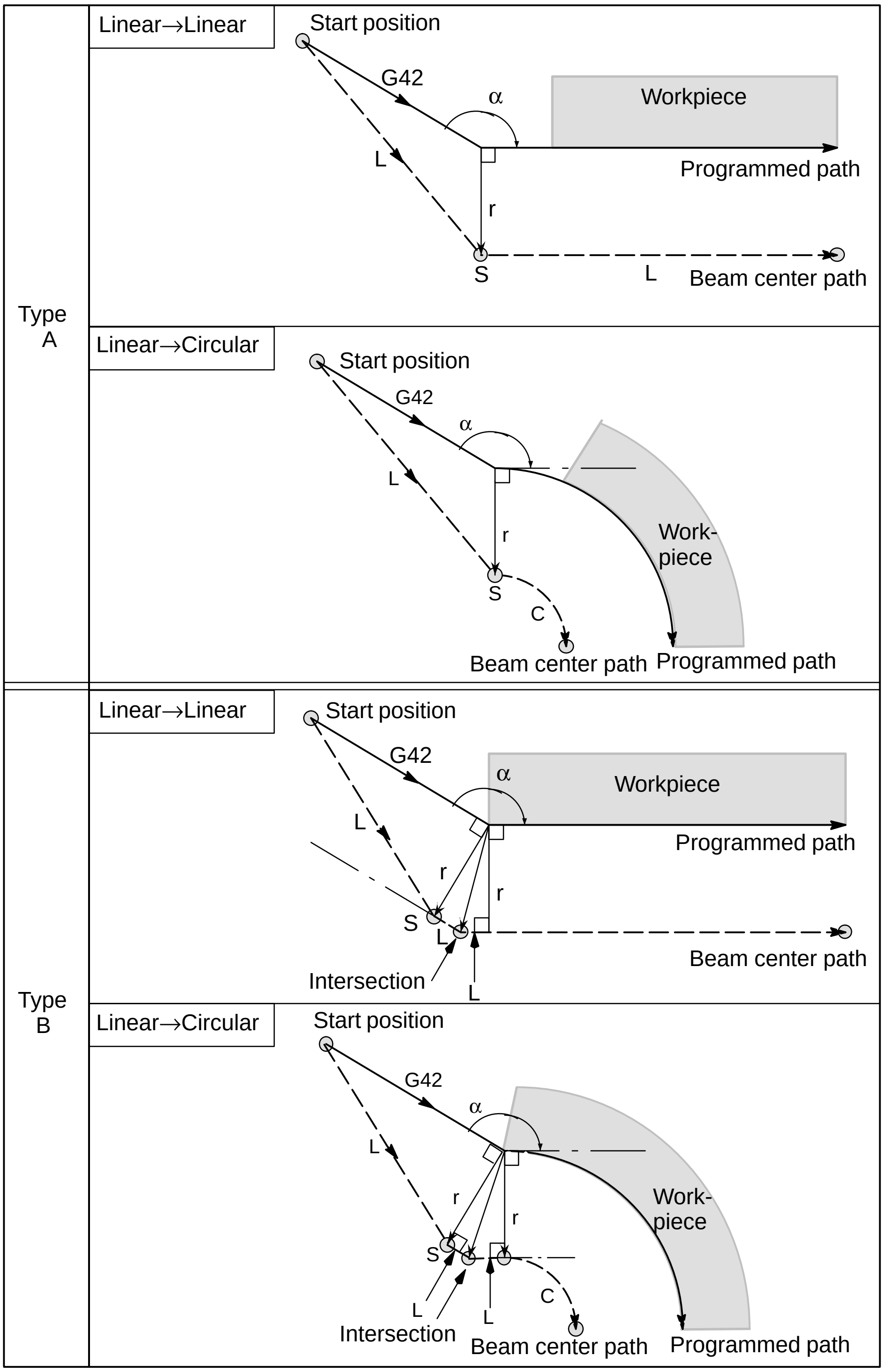
Explanations

- **Nozzle movement around an inner side of a corner (180°≦α)**



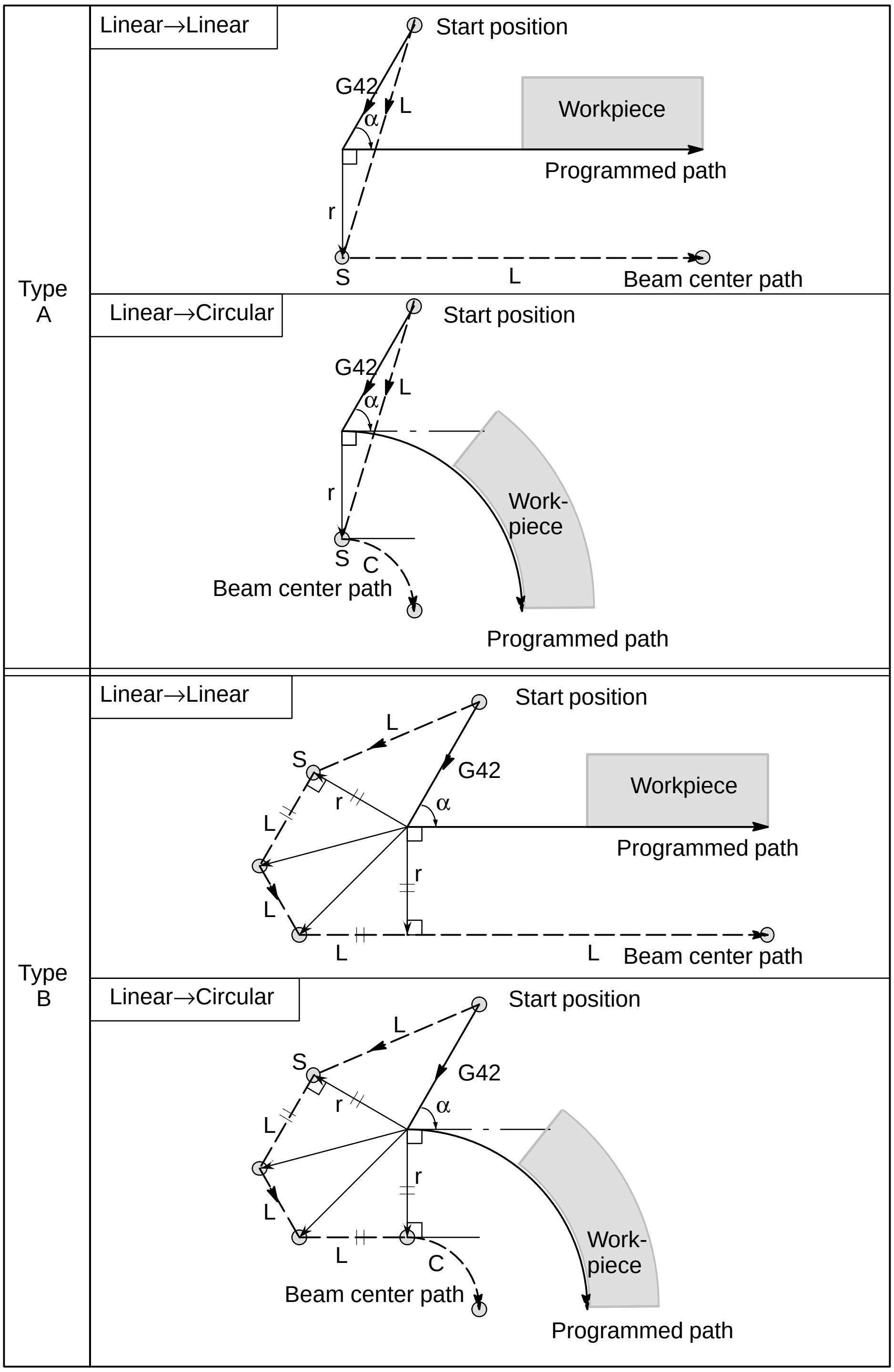
- **Nozzle movement around the outside of a corner at an obtuse angle (90°≦α<180°)**

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

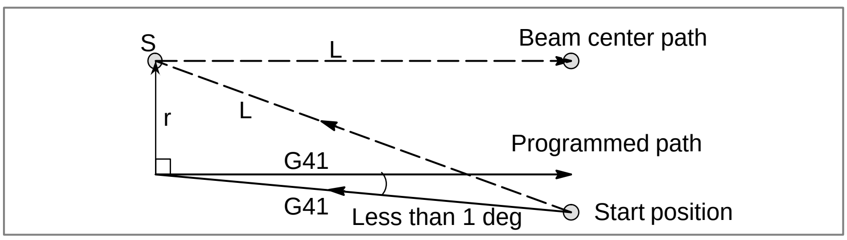


- **Nozzle movement around the outside of an acute angle (α<90°)**

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



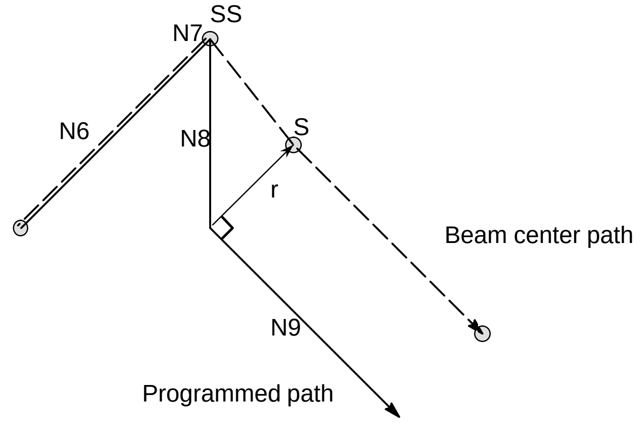
- **Beam movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



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

If the command is specified 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 ;
```



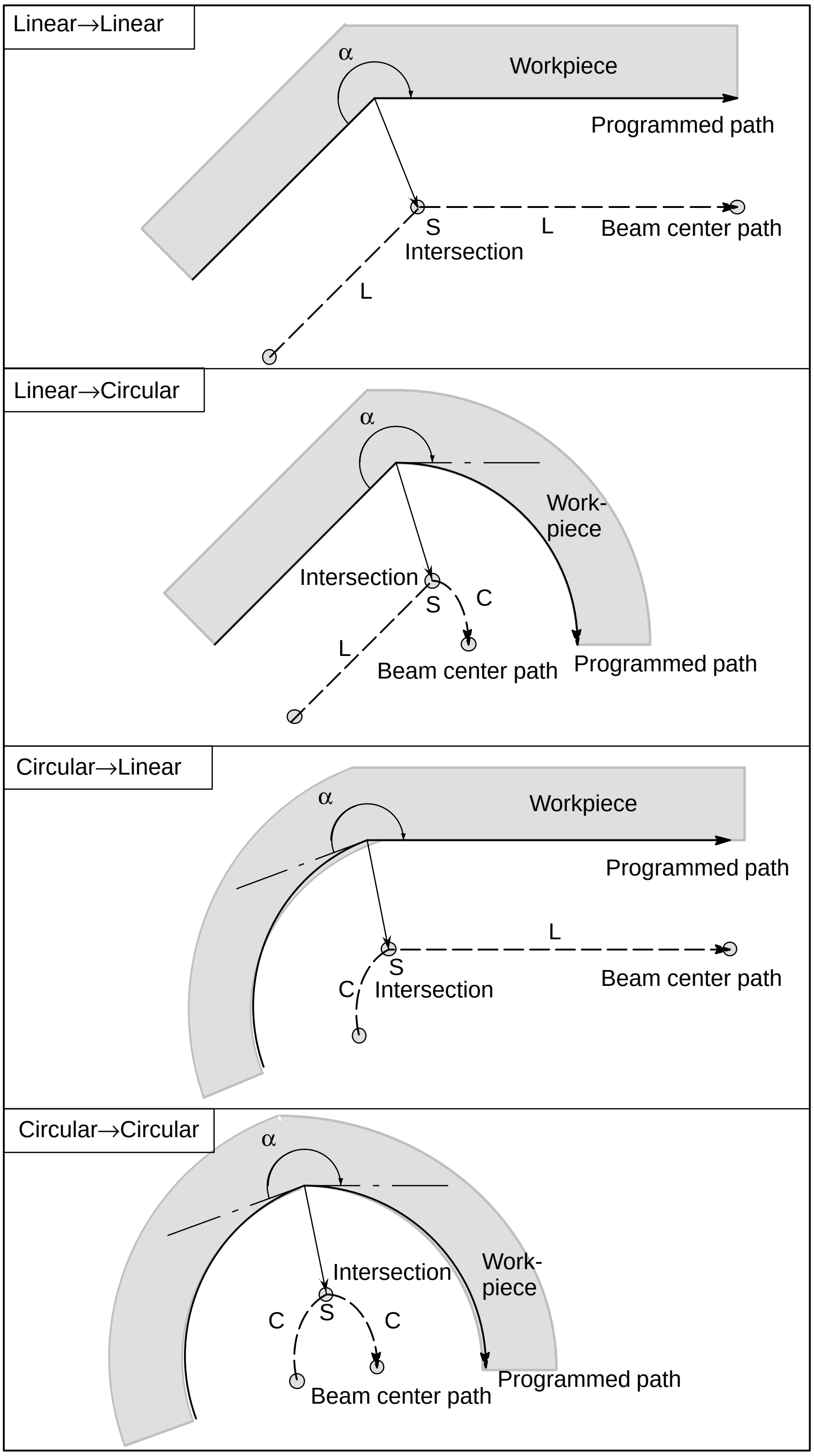
> **NOTE**
> For the definition of blocks that do not move the nozzle, see II–12.3.3.

12.3.3 Nozzle Movement in Offset Mode

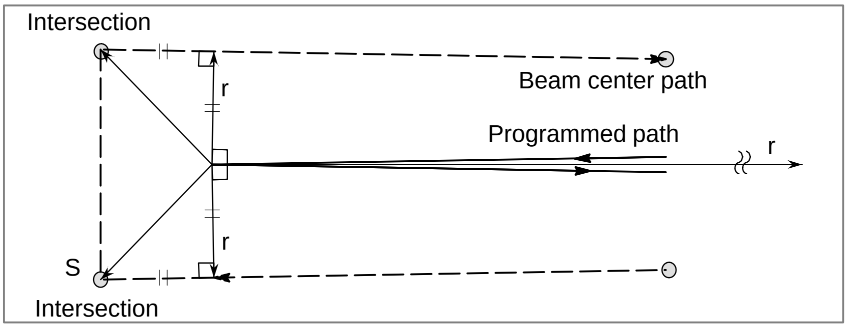
In the offset mode, the nozzle moves as illustrated below:

Explanations

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

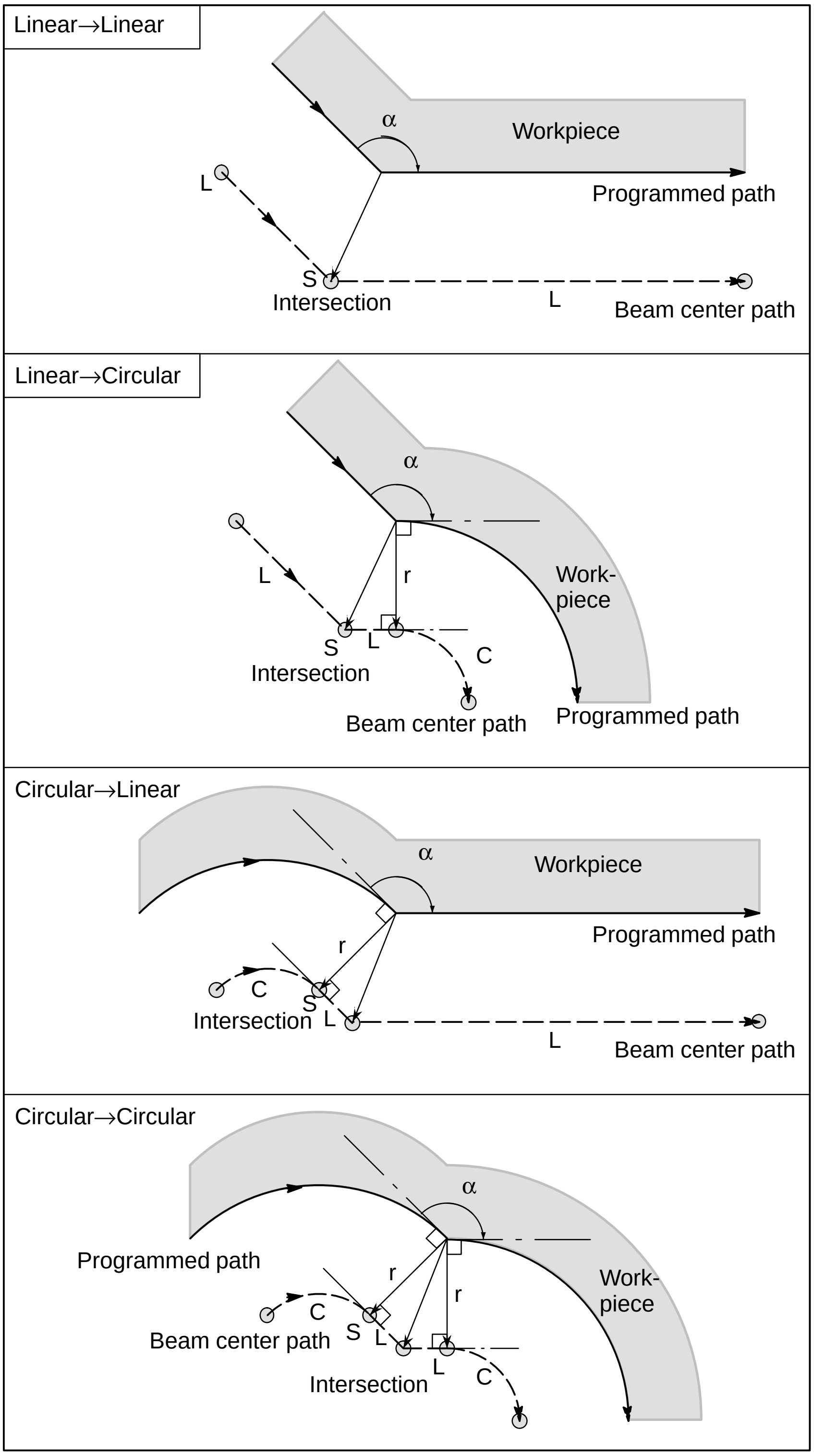


- **Nozzle movement around the inside (α<1°) with an abnormally long vector, linear → linear**

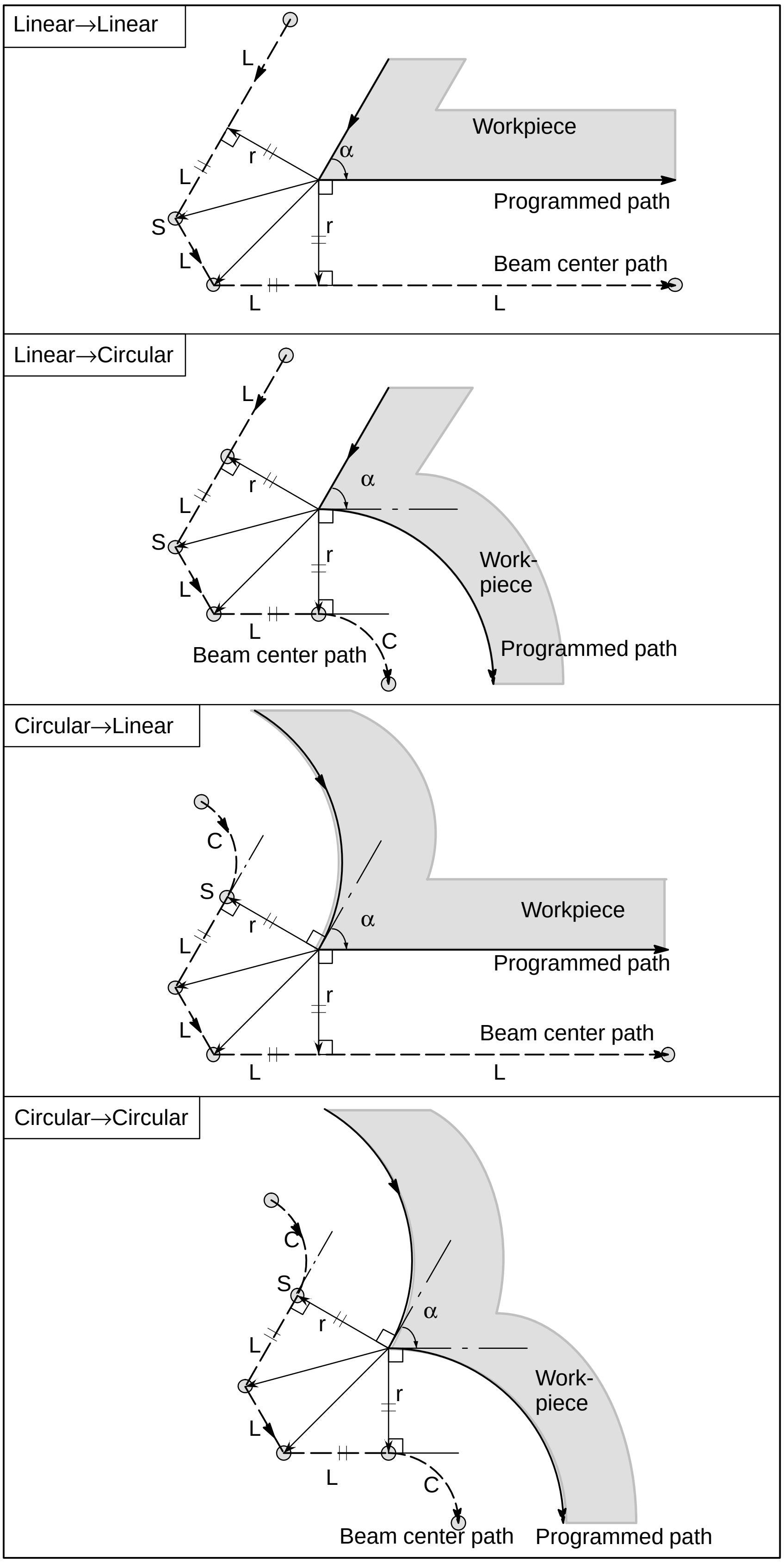


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

- **Nozzle movement around the outside corner at an obtuse angle ($90° \leqq \alpha < 180°$)**



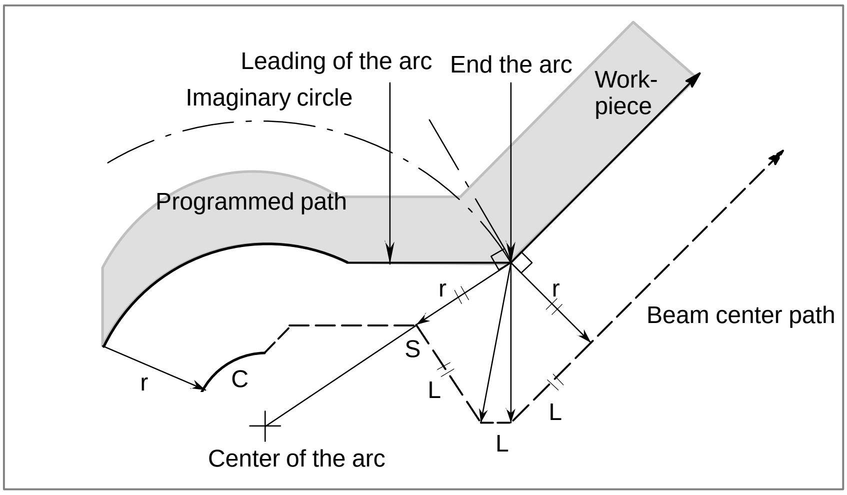
- **Nozzle movement around the outside corner at an acute angle (α<90°)**



- **When it is exceptional**

End position for the arc is not on the arc

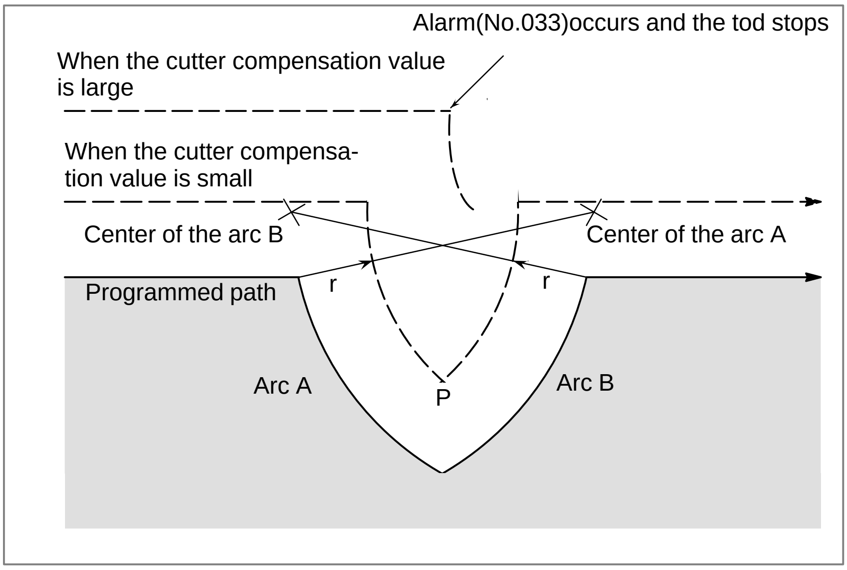
If the end of a line leading to an arc is programmed as the end of the arc by mistake as illustrated below, the system assumes that cutter compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption, the system creates a vector and carries out compensation. The resulting beam 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 beam movement between two circular paths.

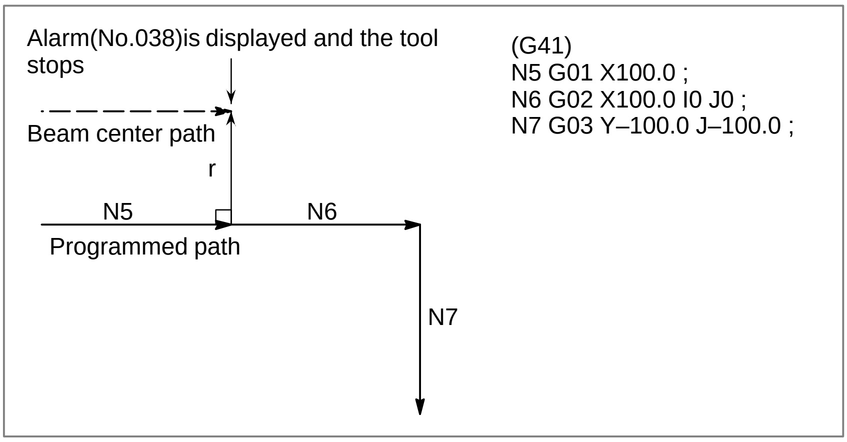
There is no inner intersection

If the cutter compensation value is sufficiently small, the two circular beam 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, P/S alarm No.033 occurs at the end of the previous block and the beam is stopped. In the example shown below, beam 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, P/S alarm (No. 038) is displayed, and the nozzle will stop at the end position of the preceding block.



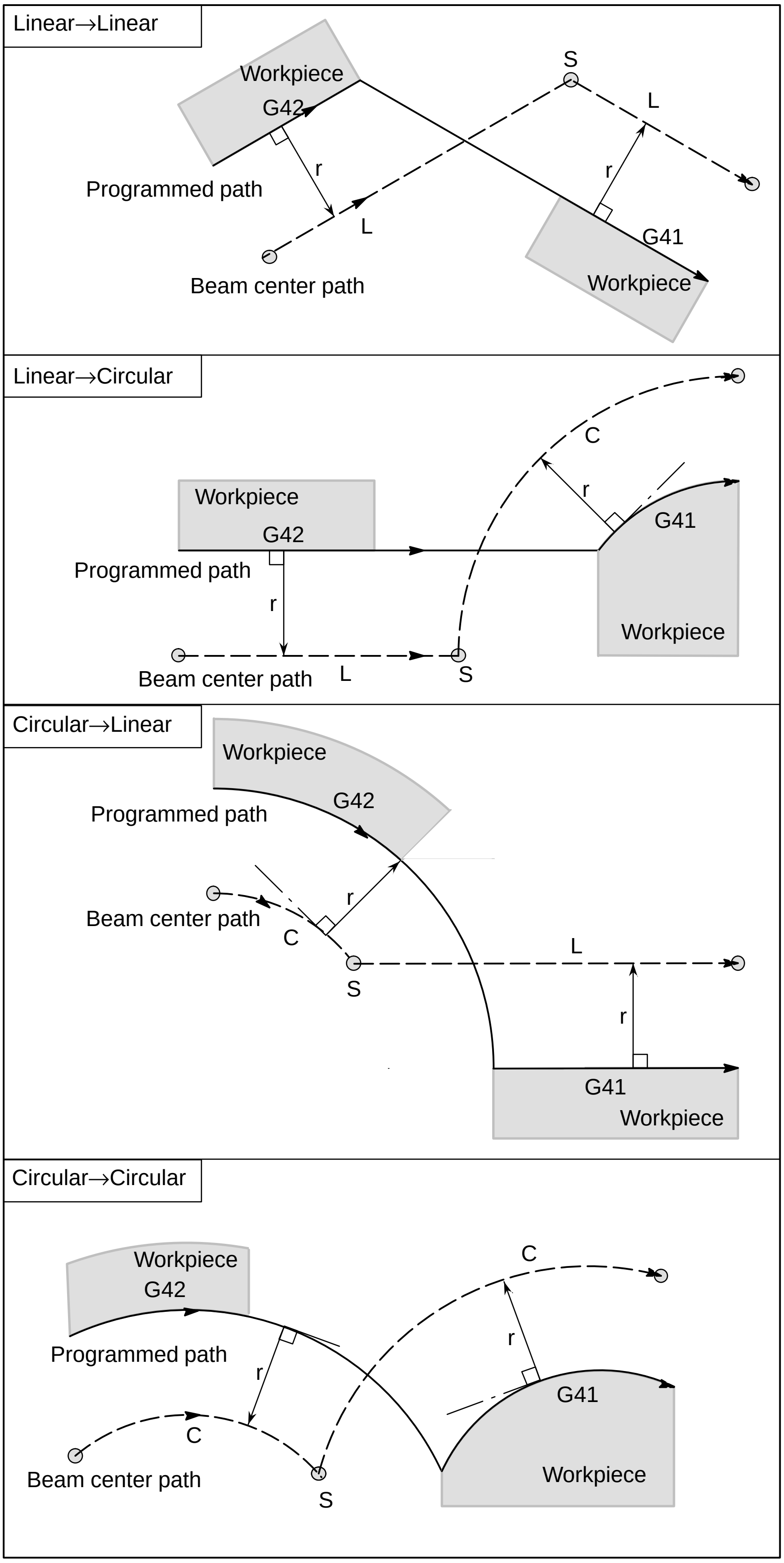
- **Change in the offset direction in the offset mode**

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

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

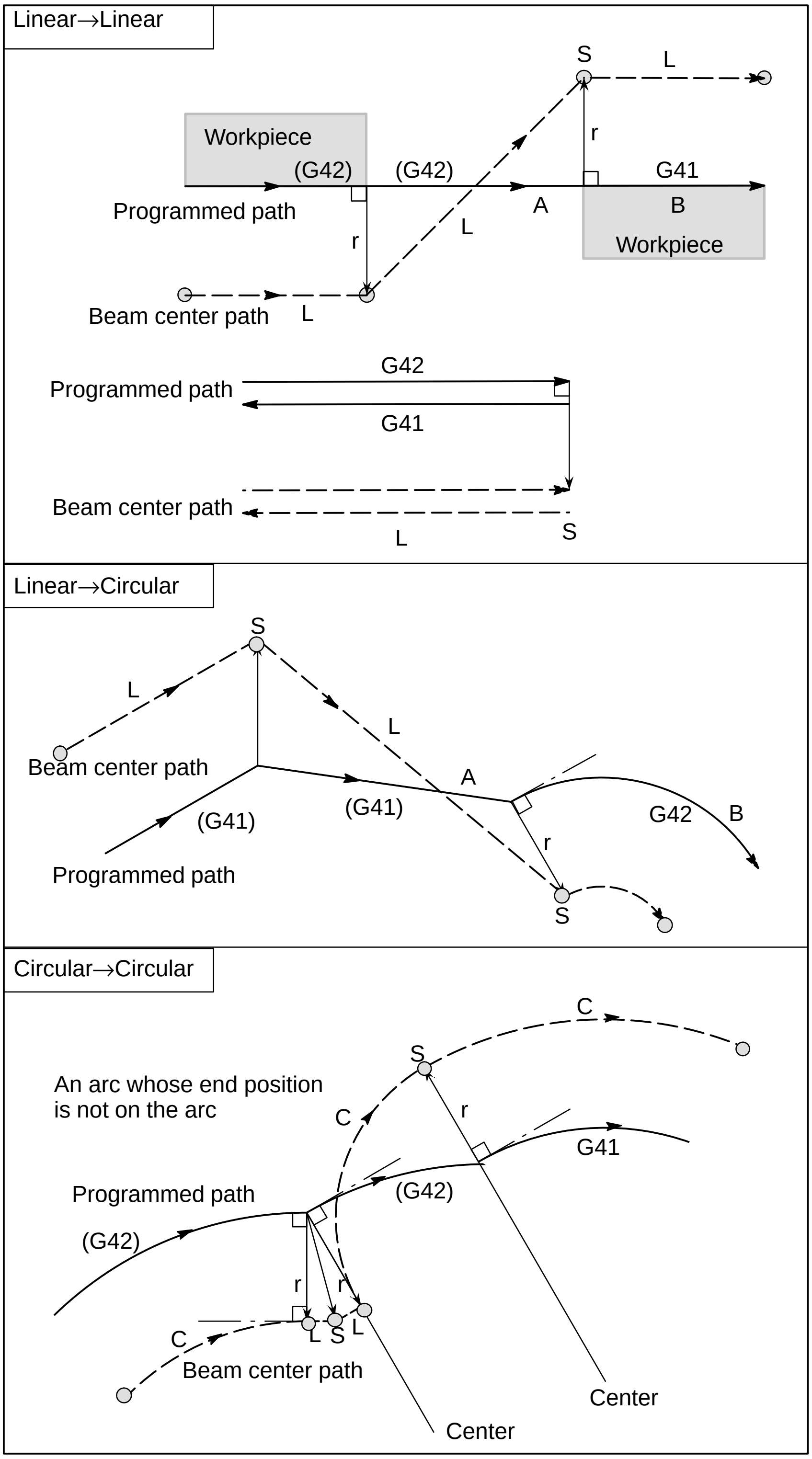
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the beam center path of that block and the beam center path of a preceding block. However, the change is not available in the start–up block and the block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

Beam center path with an intersection



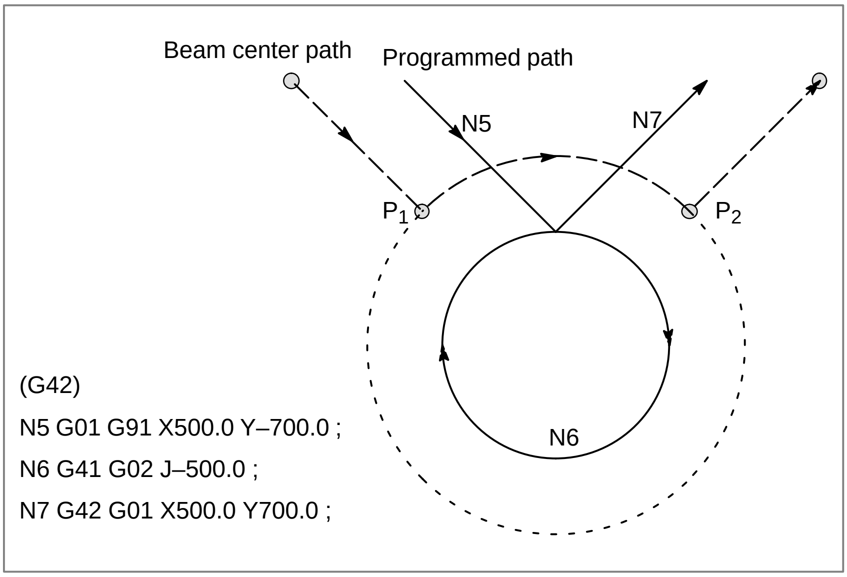
Beam center path without an intersection

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 beam 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.

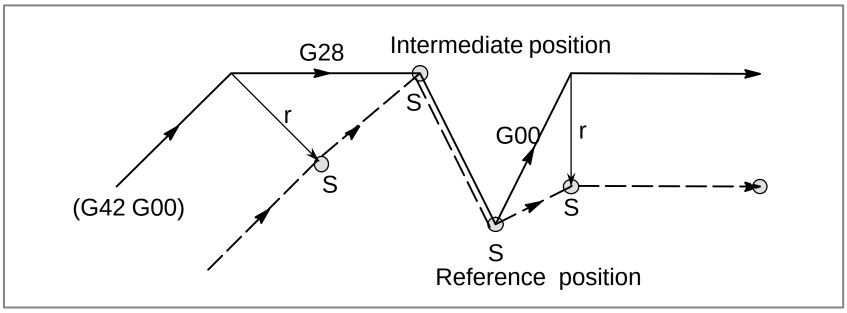


- **Temporary cutter compensation cancel**

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 II–12.3.2 and 12.3.4.

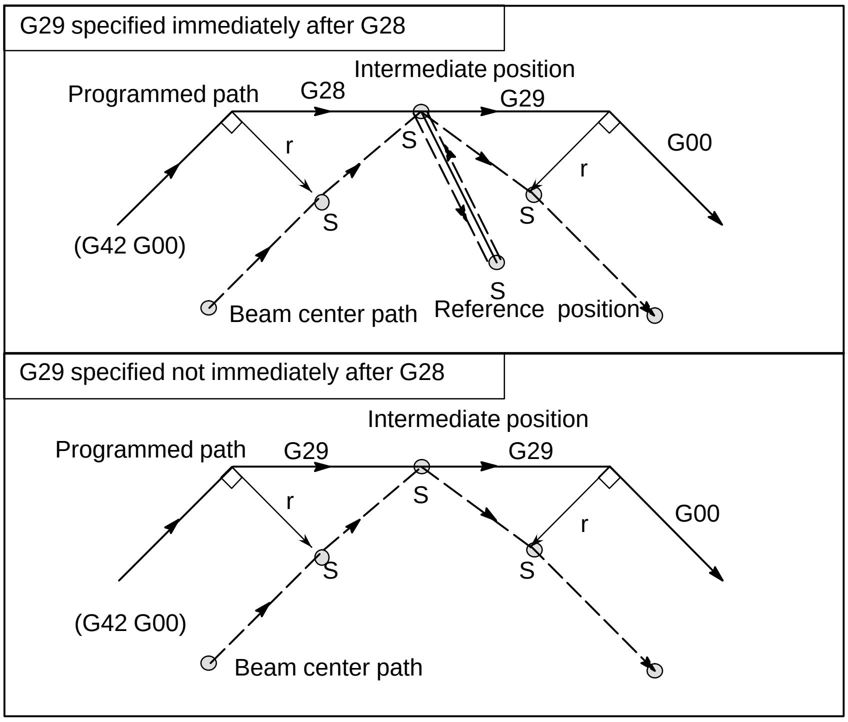
Specifying G28 (automatic return to the reference position) in the offset mode

If G28 is specified in the offset mode, the offset mode is canceled at an intermediate position. If the vector still remains after the beam 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.



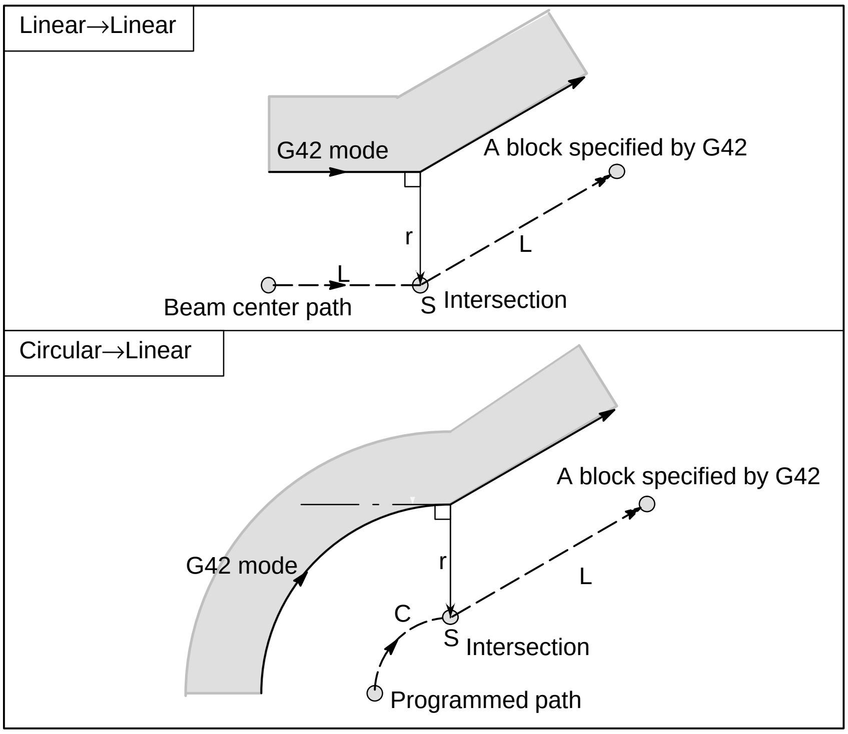
Specifying G29 (automatic return from the reference position) in the offset mode

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.



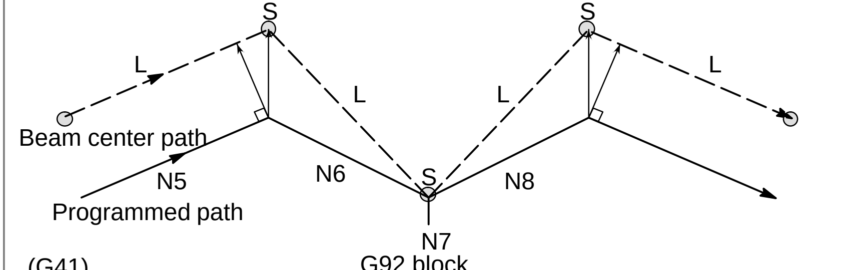
- **Cutter compensation G code in the offset mode**

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. “Changing in the offset direction in the offset mode”.



- **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 beam moves directly from the intersecting point to the commanded point where offset vector is canceled. Also when restored to offset mode, the beam 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 beam movement**

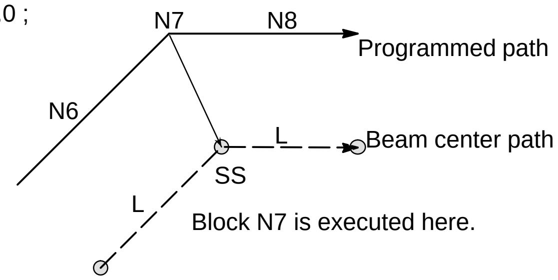
The following blocks have no beam movement. In these blocks, the beam will not move even if cutter compensation is effected.

M05 ;	M code output	Commands (1) to (6) are of no movement.
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.	

A block without beam movement specified in offset mode

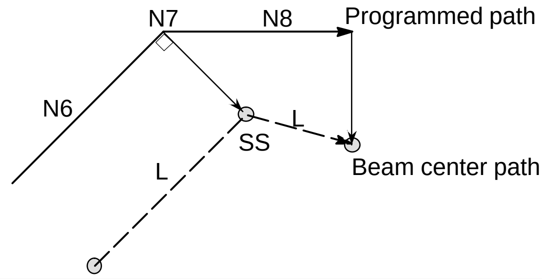
When a single block without baem movement is commanded in the offset mode, the vector and beam center path are the same as those when the block is not commanded. This block is executed at the single block stop point.

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



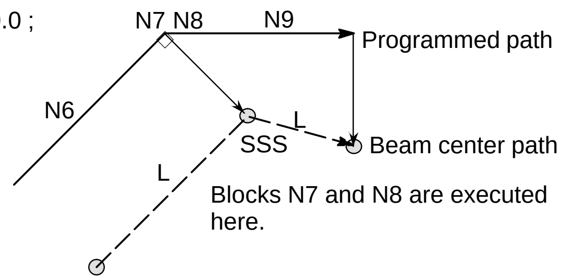
However, when the move distance is zero, even if the block is commanded singly, beam motion becomes the same as that when more than one block of without beam movement are commanded, which will be described subsequently.

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



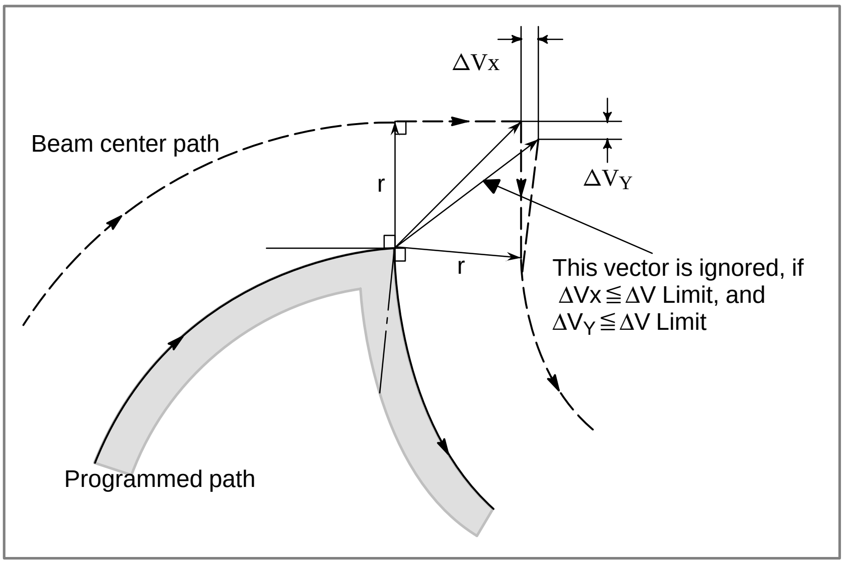
Two blocks without beam movement should not be commanded consecutively. If commanded, a vector whose length is equal to the offset value is produced in a normal direction to beam motion in earlier block, so overcutting may result.

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

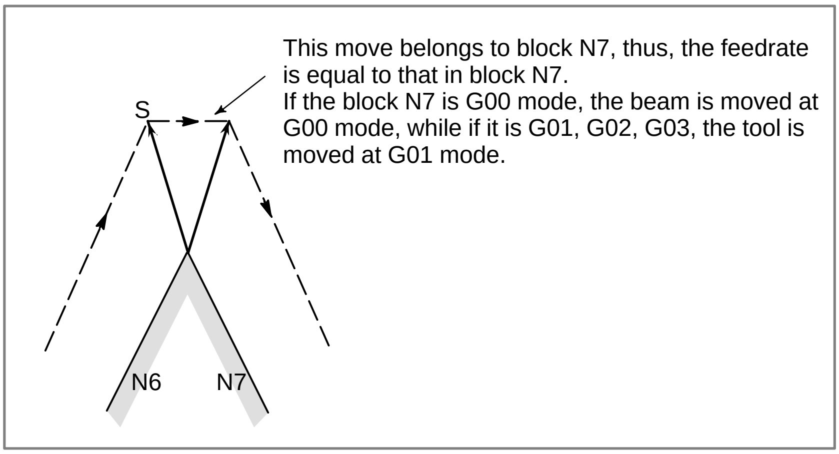


- **Corner movement**

When two or more vectors are produced at the end of a block, the beam 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.

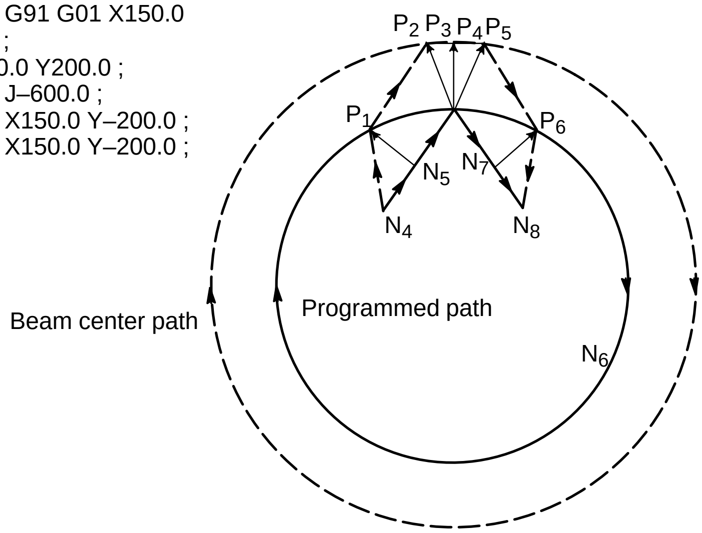


If $\Delta Vx \leqq \Delta V$limit and $\Delta Vy \leqq \Delta V$limit, the latter vector is ignored. The ΔVlimit is set in advance by parameter (No. 5010).
If these vectors do not coincide, a move is generated to turn around the corner. This move belongs to the latter block.



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 beam 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 beam path is as follows:
$P_2 \rightarrow P_4$
Namely, circle cutting by the block N6 is ignored.

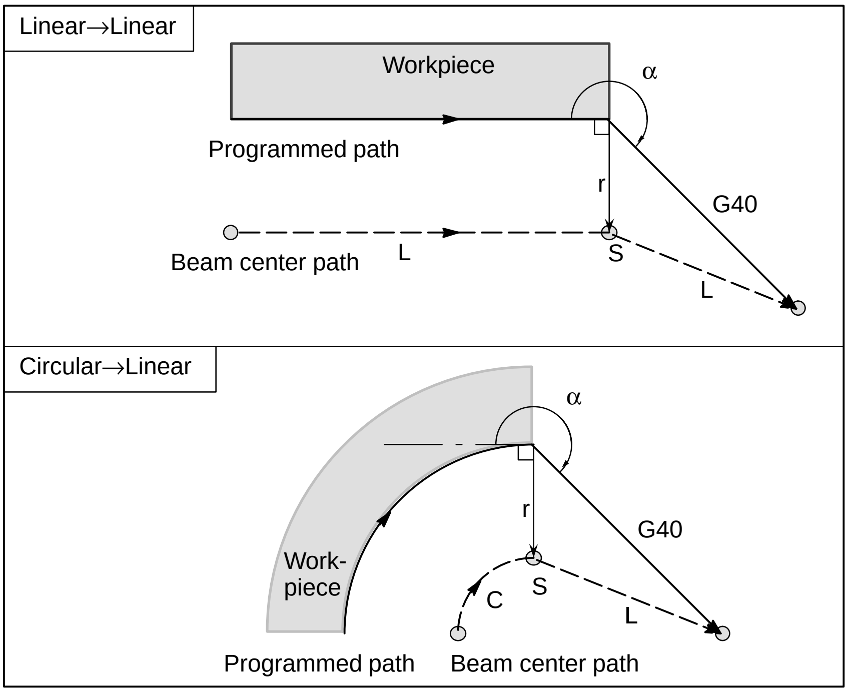
- **Interruption of manual operation**

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

12.3.4 Nozzle Movement in Offset Mode Cancel

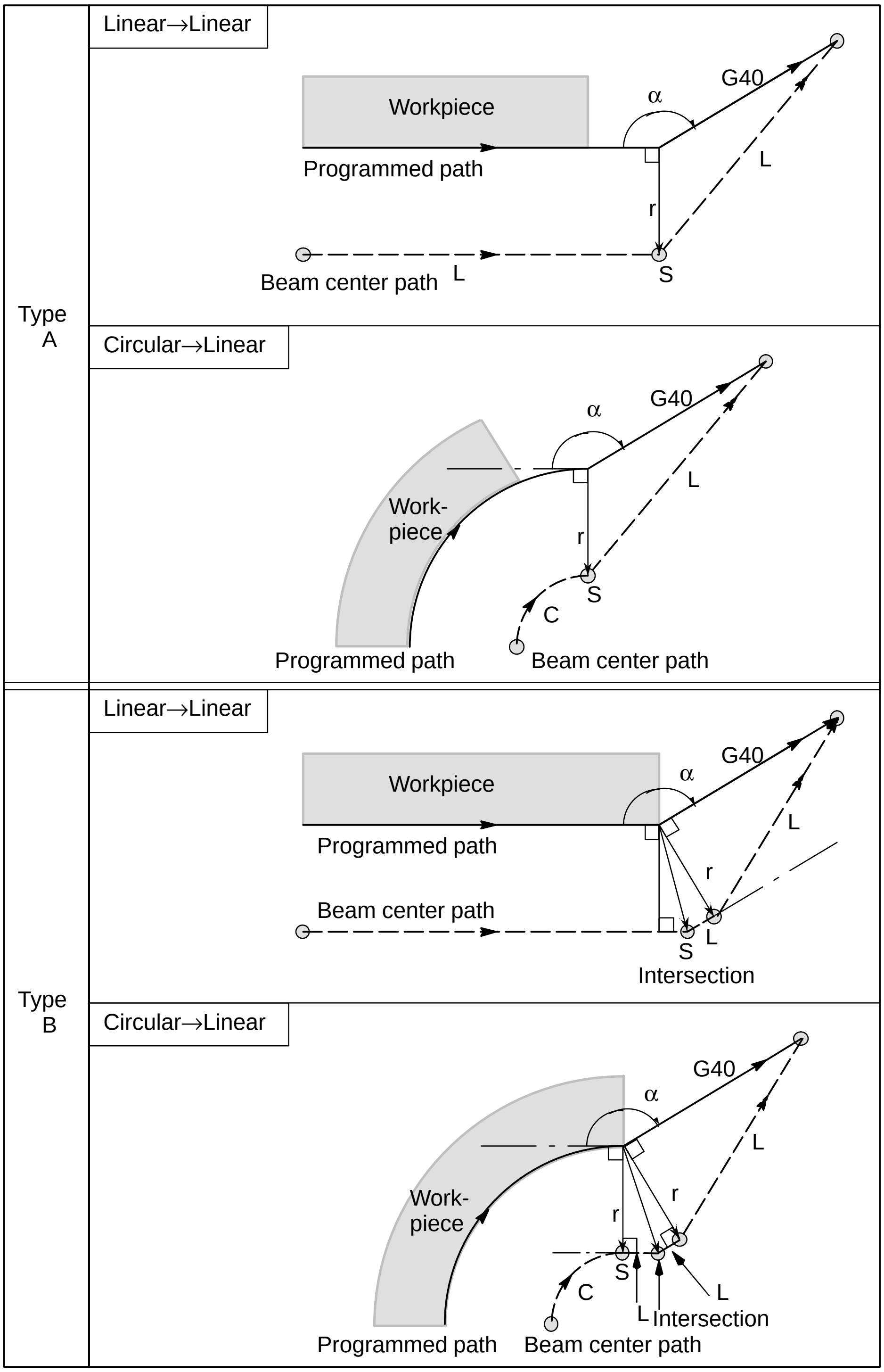
Explanations

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



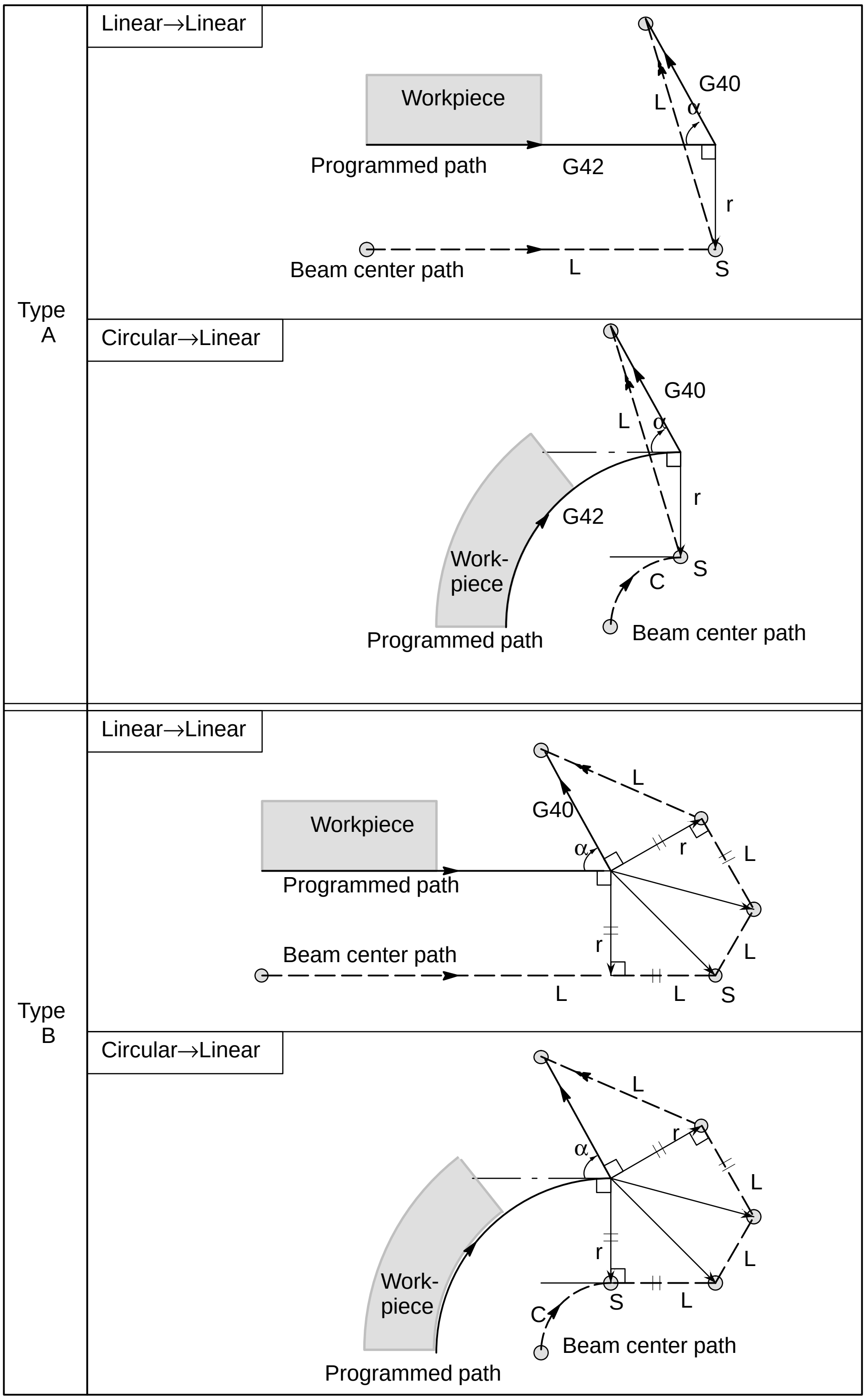
- **Nozzle movement around an outside corner at an obtuse angle (90°≦α<180°)**

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

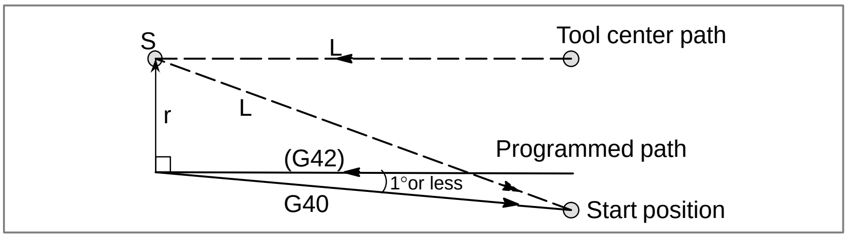


- **Nozzle movement around an outside corner at an acute angle (α<90°)**

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

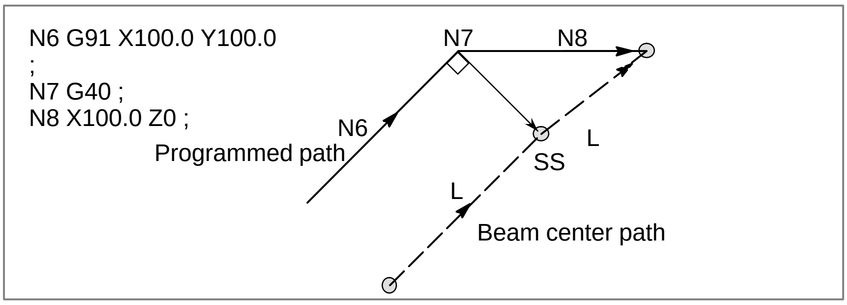


- **Nozzle movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



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

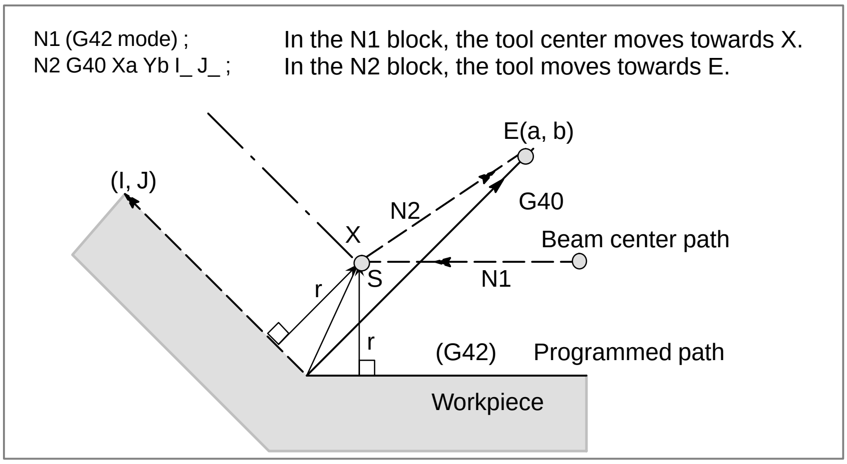
When a block without beam movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to beam motion in the earlier block, the vector is cancelled in the next move command.



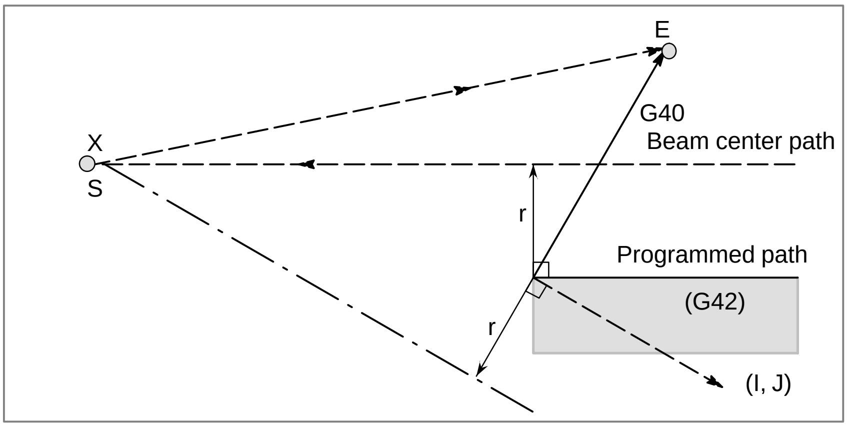
• Block containing G40 and I_J_K_

The previous block contains G41 or G42

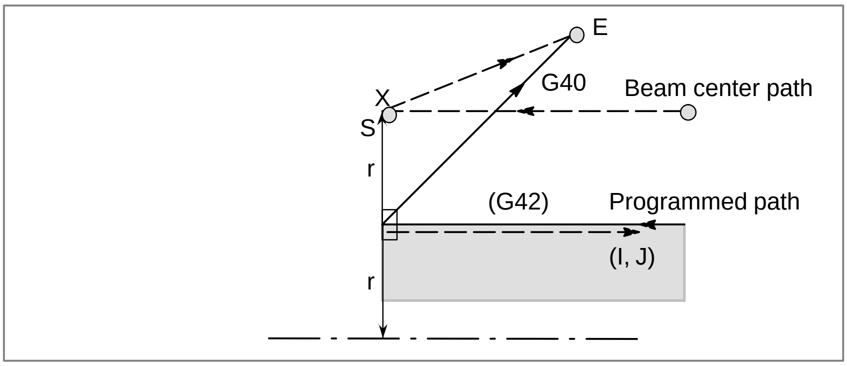
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



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

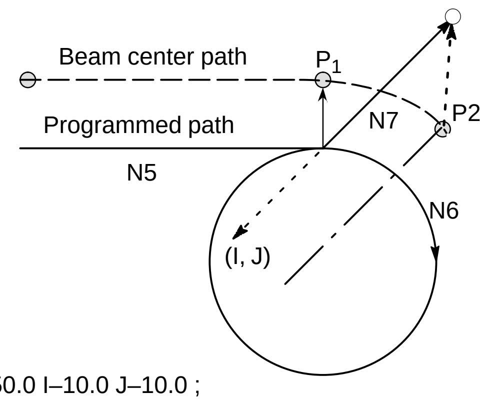


When an intersection is not obtainable, the beam comes to the normal position to the previous block at the end of the previous block.



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

In the example shown below, the beam does not trace the circle more than once. It moves along the arc from P1 to P2. The interference check function described in II–12.3.5 may raise an alarm.



(G41)
N5 G01 G91 X100.0 ;
N6 G02 J–60.0 ;
N7 G40 G01 X50.0 Y50.0 I–10.0 J–10.0 ;

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

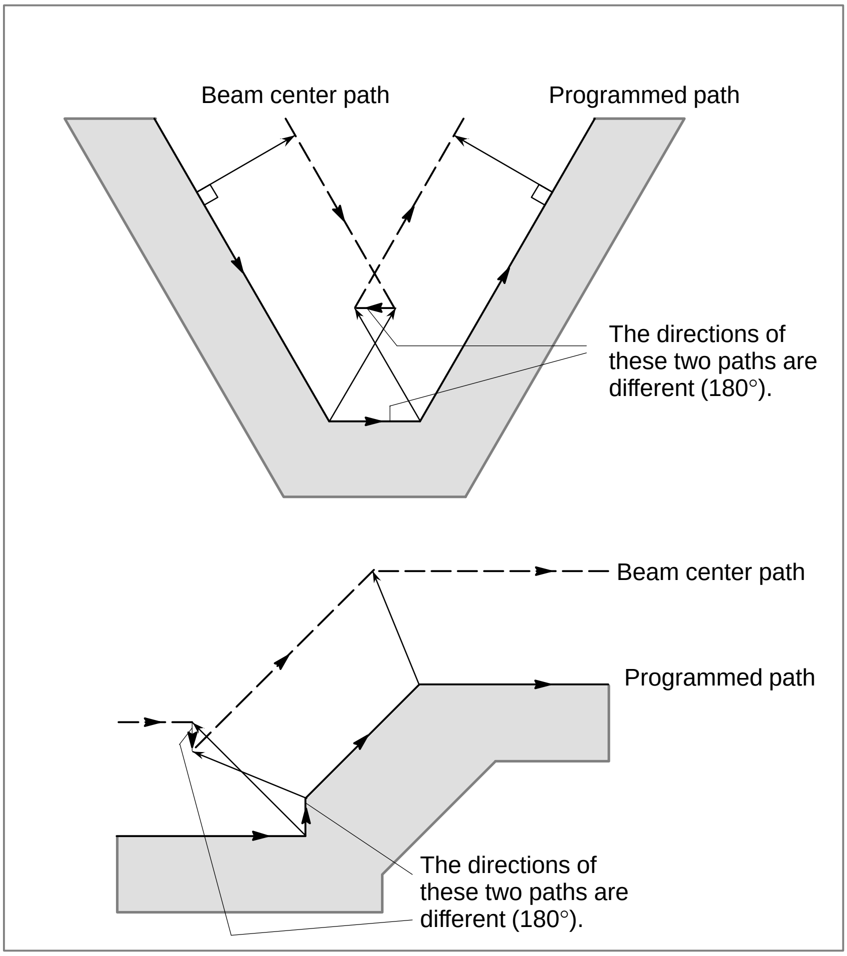
12.3.5 Interference Check

Beam 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.

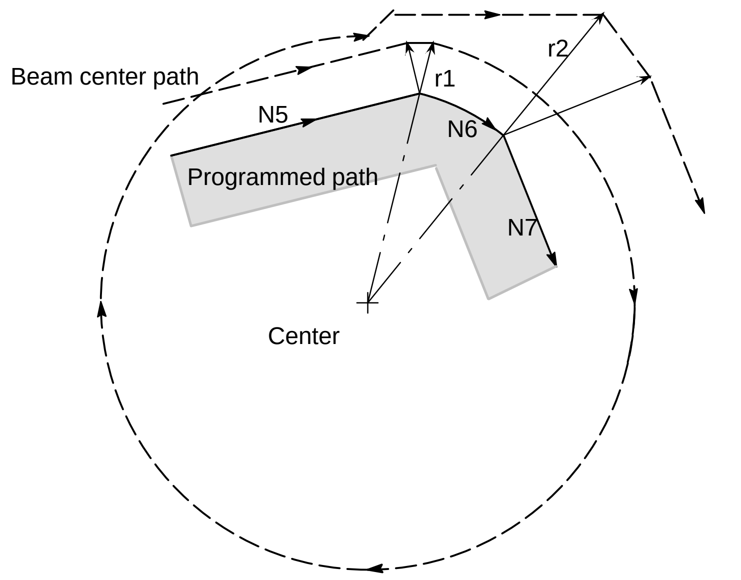
Explanations

- **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 beam 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 ;
(Cutter compensation value corresponding to D1 : r_1 = 200.0)
(Cutter 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

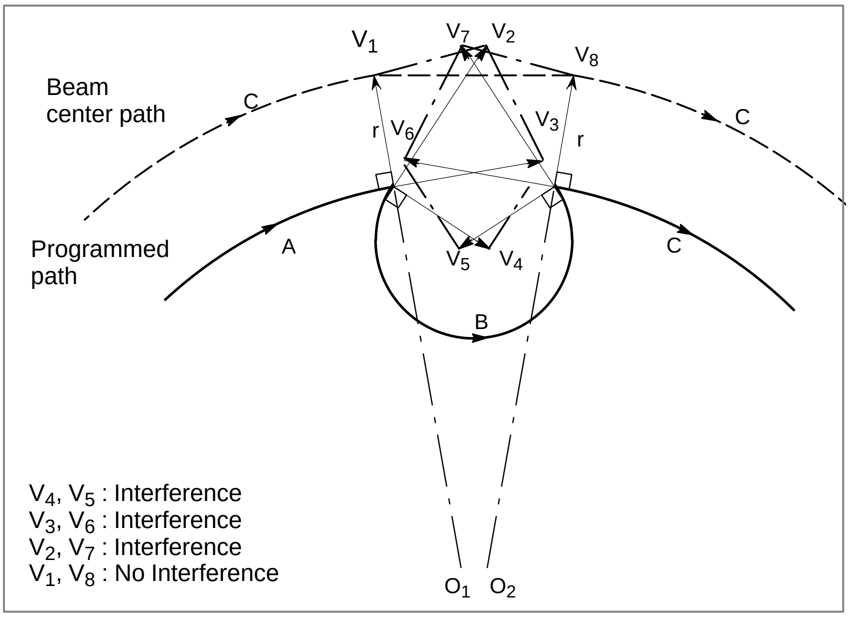
Interference —– V_2 and V_7 are Ignored

Check between V_1 and V_8

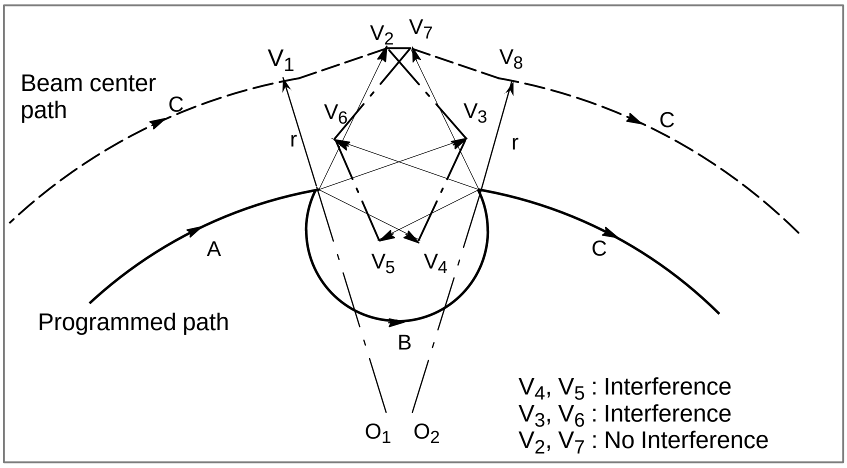
Interference ——– V_1 and V_8 are cannot be ignored

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.

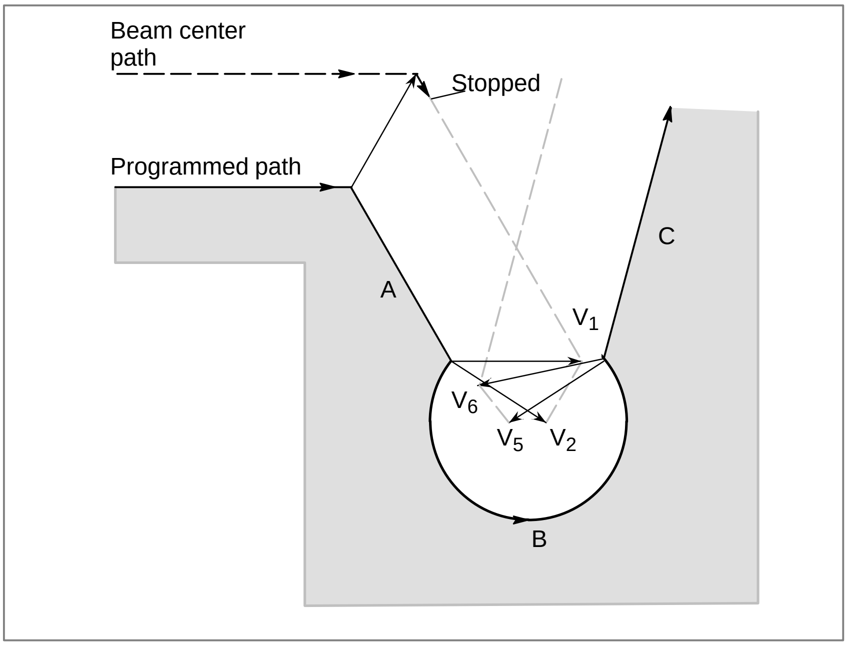
(Example 1) The beam moves linearly from V_1 to V_8



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



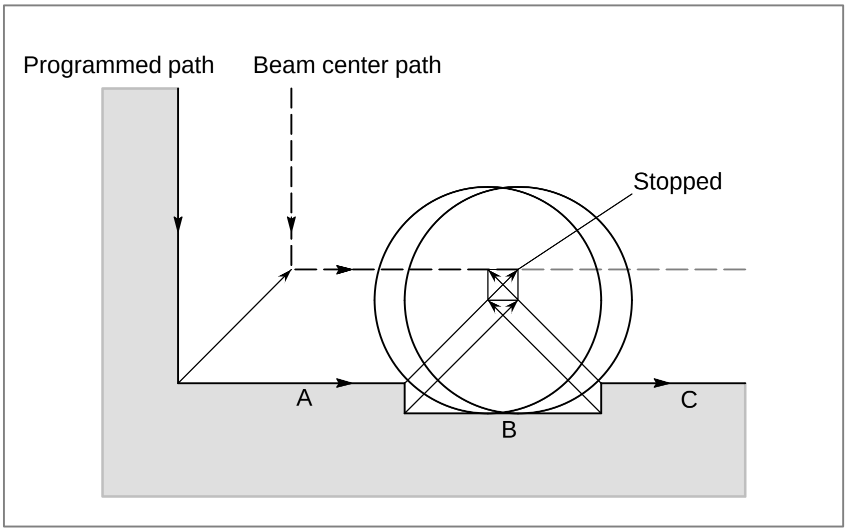
(2) If the interference occurs after correction (1), the beam is stopped with an alarm.
If the interference occurs after correction (1) or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the P/S alarm (No.41) is displayed and the beam is stopped immediately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

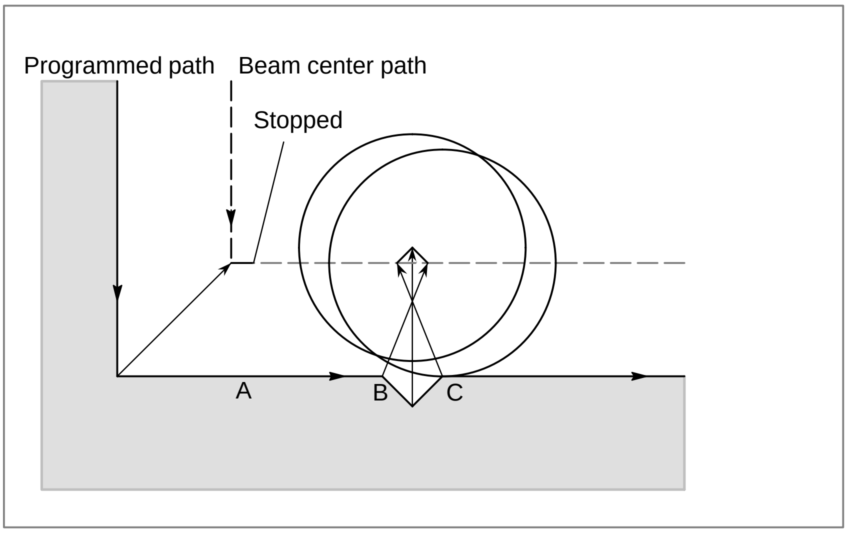
- **When interference is assumed although actual interference does not occur**

(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 beam stops and an alarm is displayed.

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



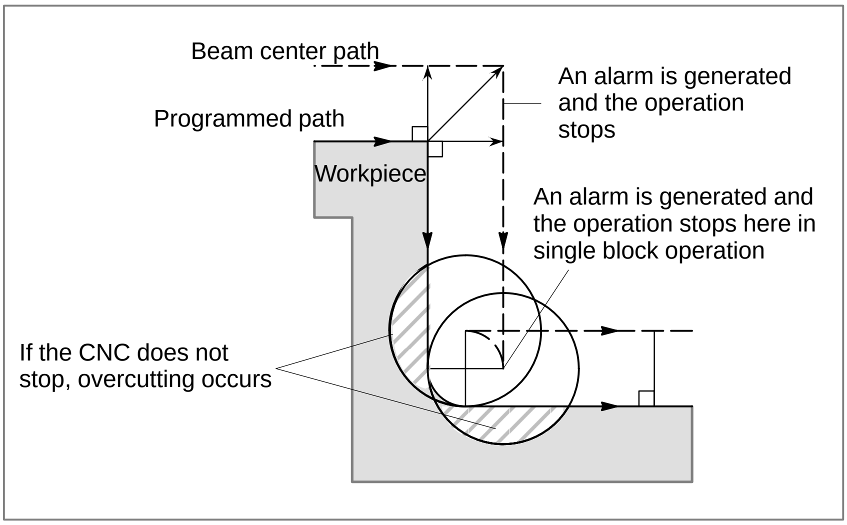
Like (1), P/S alarm is displayed because of the interference as the direction is reverse in block B.

12.3.6 Overcutting by Cutter Compensation

Explanations

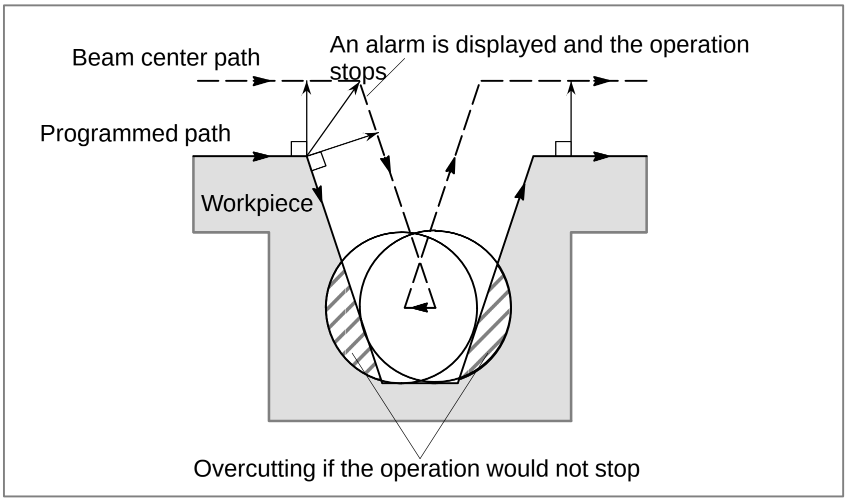
- **Machining an inside corner at a radius smaller than the beam radius**

When the radius of a corner is smaller than the beam 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 beam is stopped after the block execution.



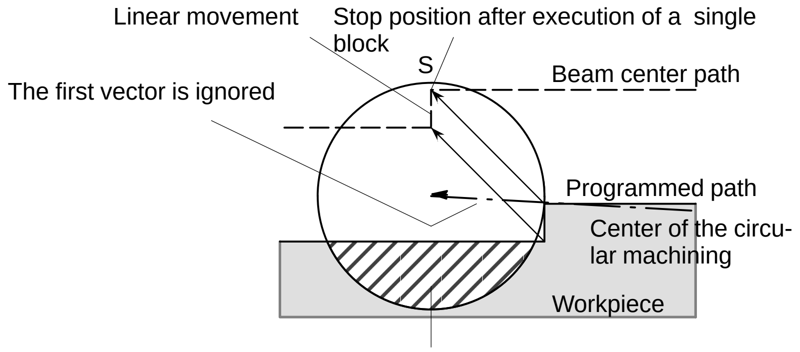
- **Machining a groove smaller than the beam radius**

Since the cutter compensation forces the path of the center of the nozzle 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**

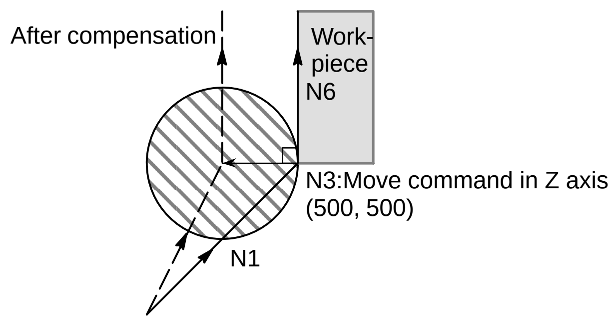
When machining of the step is commanded by circular machining in the case of a program containing a step smaller than the beam radius, the path of the center of beam with the ordinary offset becomes reverse to the programmed direction. In this case, the first vector is ignored, and the beam moves linearly 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 beam 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.

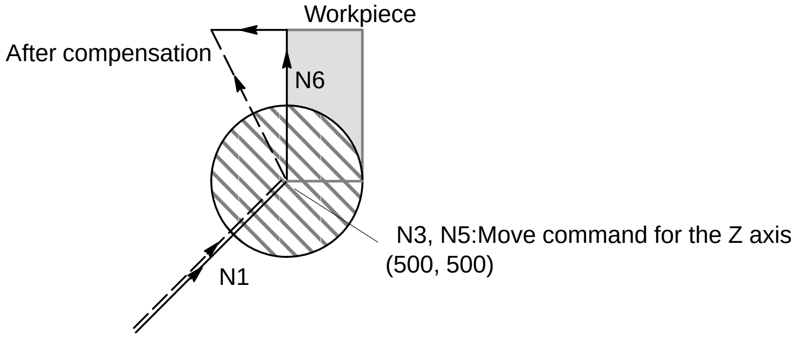
```
N1 G91 G00 G41 X500.0 Y500.0 D1 ;
N3 G01 Z–300.0 F100 ;
N6 Y1000.0 F200 ;
```



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 beam 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.

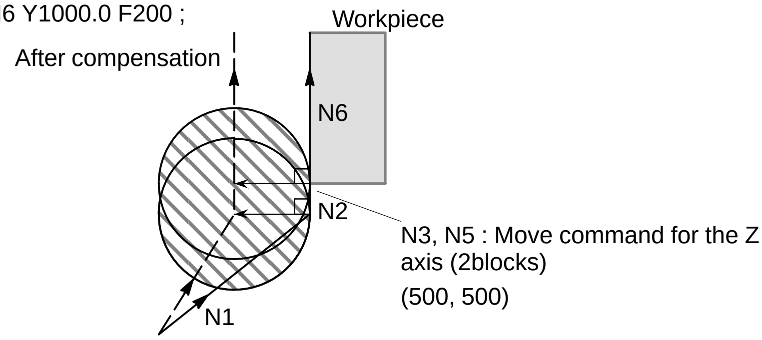
The above example should be modified as follows:

```
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 ;
```



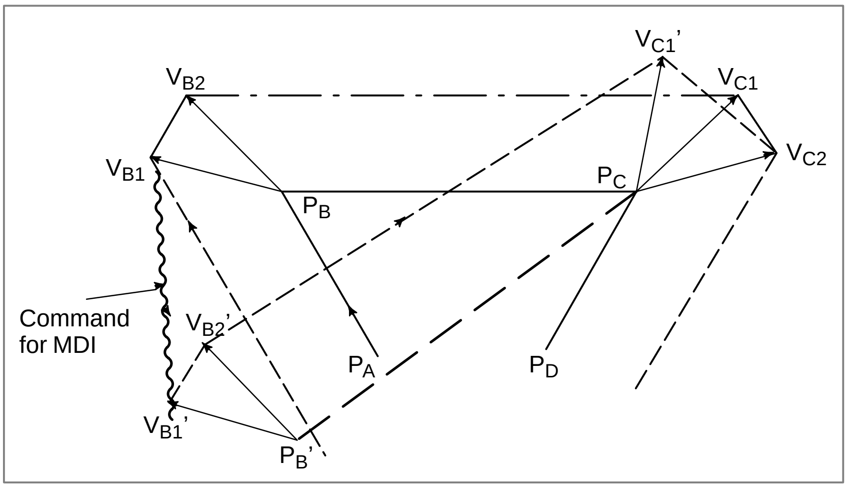
As the block with sequence No. N2 has the move command in the same direction as that of the block with sequence No. N6, the correct compensation is performed.

12.3.7 Input Command from MDI

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

However, when automatic operation using the absolute commands is temporarily stopped by the single block function, MDI operation is performed, then automatic operation starts again, the beam 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, beam is stopped by the single block function after executing the block from P_A to P_B and the beam 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.

12.3.8 G53, G28, G30, G30.1 and G29 Commands in Cutter Compensation C Mode

A function has been added which performs positioning by automatically canceling a cutter compensation vector when G53 is specified in cutter compensation C mode, then automatically restoring that cutter compensation vector with the execution of the next move command.
The cutter compensation vector restoration mode is of FS16 type when CCN (bit 2 of parameter No. 5003) is set to 0; it is of FS15 type when CCN is set to 1.

When G28, G30, or G30.1 is specified in cutter compensation C mode, automatic reference position return is performed by automatically canceling a cutter compensation vector, that cutter compensation vector automatically being restored with the execution of the next move command. In this case, the timing and format of cutter compensation vector cancellation/restoration, performed when CCN (bit 2 of parameter No. 5003) is set to 1, are changed to FS15 type.
When CCN (bit 2 of parameter No. 5003) is set to 0, the conventional specification remains applicable.

When G29 is specified in cutter compensation C mode, the cutter compensation vector is automatically canceled/restored. In this case, the timing and format of cutter compensation vector cancellation/restoration, performed when CCN (bit 2 of parameter No. 5003) is set to 1, are changed to FS15 type.
When CCN (bit 2 of parameter No. 5003) is set to 0, the conventional specification remains applicable.

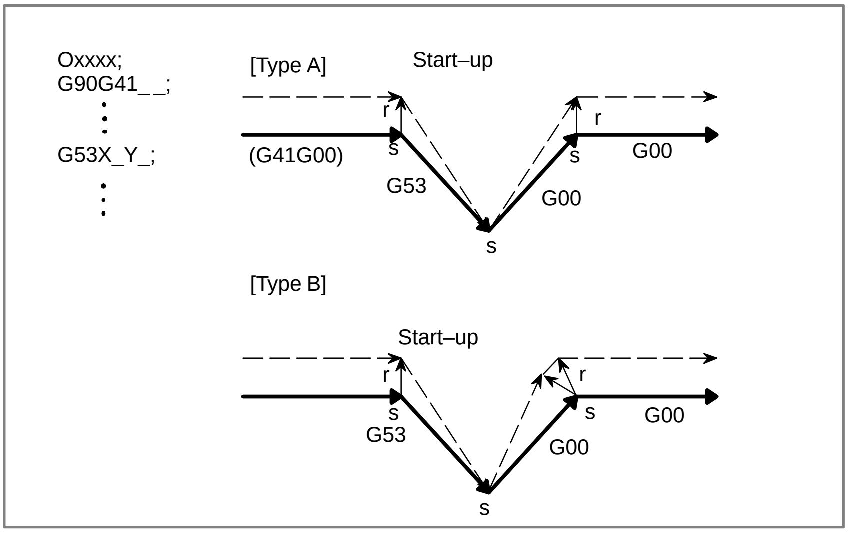
Explanations

- **G53 command in cutter compensation C mode**

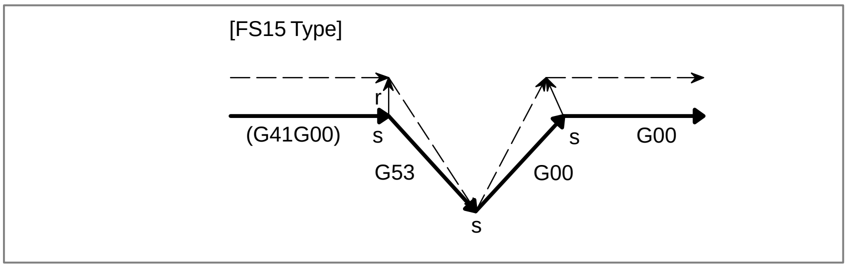
When G53 is specified in cutter compensation C mode, the previous block generates a vector that is perpendicular to the move direction and which has the same magnitude as the offset value. Then, the offset vector is canceled when movement to a specified position is performed in the machine coordinate system. In the next block, offset mode is automatically resumed.

Note that cutter compensation vector restoration is started when CCN (bit 2 of parameter No. 5003) is set to 0; when CCN is set to 1, an intersection vector is generated (FS15 type).

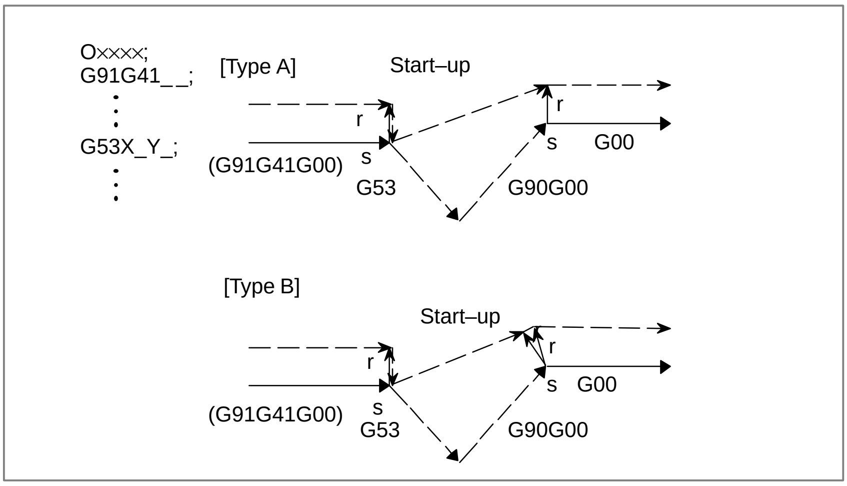
(1) G53 specified in offset mode
When CCN (bit 2 of parameter No.5003)=0



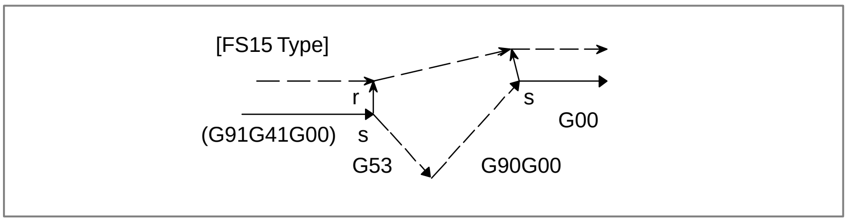
When CCN (bit 2 of parameter No.5003)=1



(2) Incremental G53 specified in offset mode
When CCN (bit 2 of parameter No.5003)=0

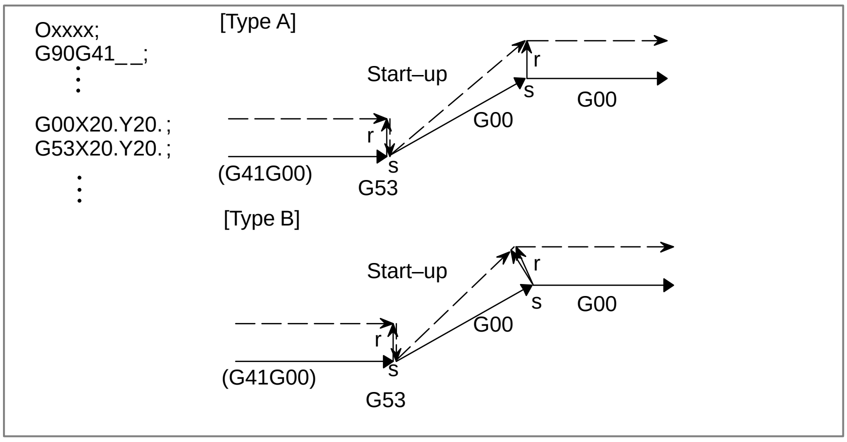


When CCN (bit2 of parameter No.5003)=1

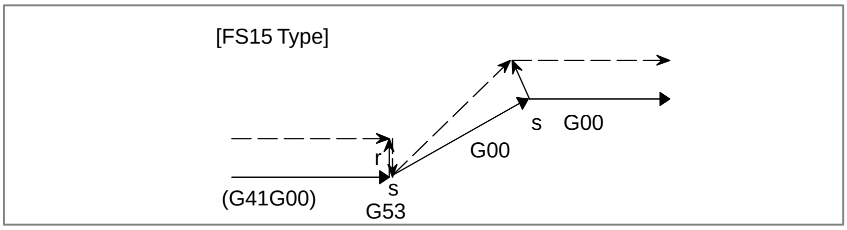


(3) G53 specified in offset mode with no movement specified

When CCN (bit2 of parameter No.5003)=0



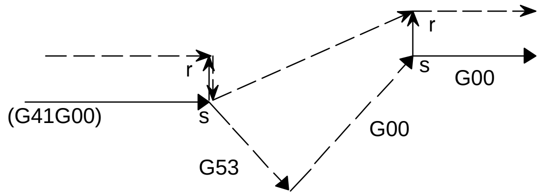
When CCN (bit2 of parameter No.5003)=1



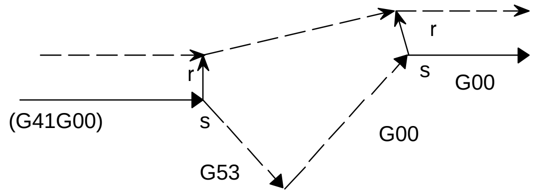
WARNING

1 When cutter compensation C mode is set and all–axis machine lock is applied, the G53 command does not perform positioning along the axes to which machine lock is applied. The vector, however, is preserved. When CCN (bit 2 of parameter No. 5003) is set to 0, the vector is canceled. (Note that even if the FS15 type is used, the vector is canceled when each–axis machine lock is applied.)

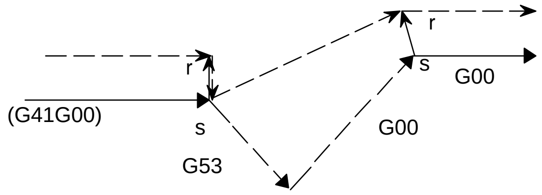
Example 1: When CCN (bit 2 of parameter No. 5003) = 0, type A is used, and all–axis machine lock is applied



Example 2: When CCN (bit 2 of parameter No. 5003) = 1 and all–axis machine lock is applied [FS15 type]

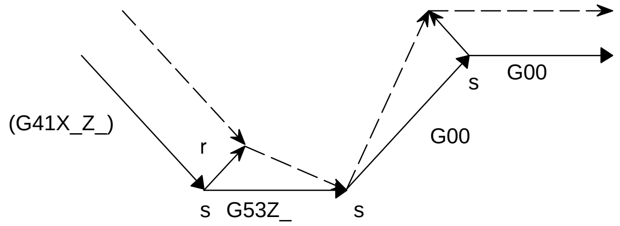


Example 3: When CCN (bit 2 of parameter No. 5003) = 1 and specified–axis machine lock is applied [FS15 type]



2 When G53 is specified for a compensation axis in cutter compensation mode, the vectors along the other axes are also canceled. (This also applies when CCN (bit 2 of parameter No.5003) is set to 1. When the FS15 type is used, only the vector along a specified axis is canceled. Note that the FS15 type cancellation differs from the actual FS15 specification in this point.)

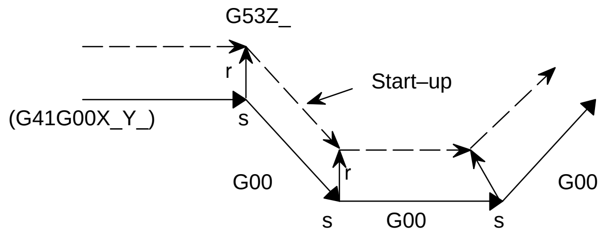
Example: When CCN (bit 2 of parameter No.5003)=1[FS 15 type]



NOTE

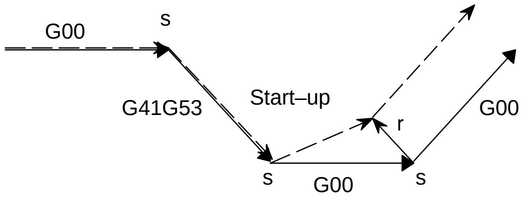
1 When a G53 command specifies an axis that is not in the cutter compensation C plane, a perpendicular vector is generated at the end point of the previous block, and the beam does not move. In the next block, offset mode is automatically resumed (in the same way as when two or more continuous blocks do not specify any move commands).

Example: When CCN (bit 2 of parameter No. 5003) = 0, and type A is used



2 When a G53 block is specified to become a start–up block, the next block actually becomes the start–up block. When CCN (bit 2 of parameter No. 5003) is set to 1, an intersection vector is generated.

Example: When CCN (bit 2 of parameter No. 5003) = 0 and type A is used



- **G28, G30, or G30.1 command in cutter compensation C mode**

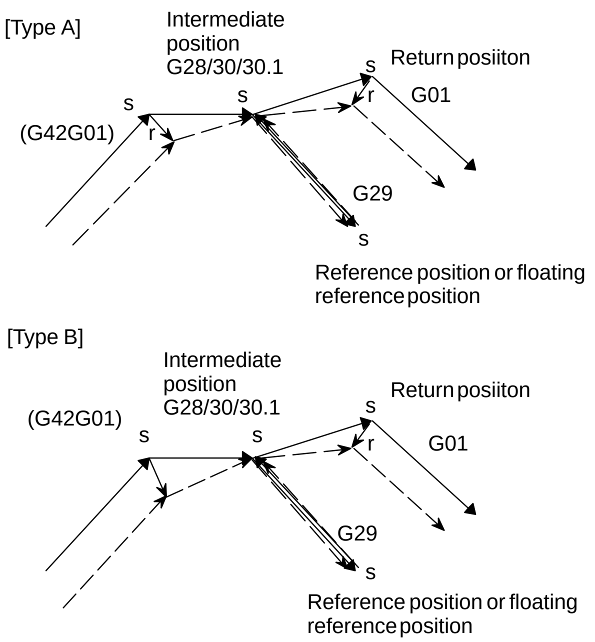
When G28, G30, or G30.1 is specified in cutter compensation C mode, an operation of FS15 type is performed if CCN (bit 2 of parameter No. 5003) is set to 1.

This means that an intersection vector is generated in the previous block, and a perpendicular vector is generated at an intermediate position. Offset vector cancellation is performed when movement is made from the intermediate position to the reference position. As part of restoration, an intersection vector is generated between a block and the next block.

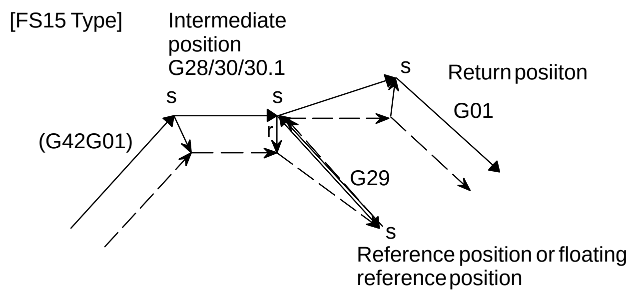
(1) G28, G30, or G30.1, specified in offset mode (with movement to both an intermediate position and reference position performed)

(a) For return by G29
When CCN (bit 2 of parameter No. 5003) = 0

```
Oxxxx;
G91G41_ _ _;
   :
G28X40.Y0 ;
G29X45.Y15. ;
   :
```

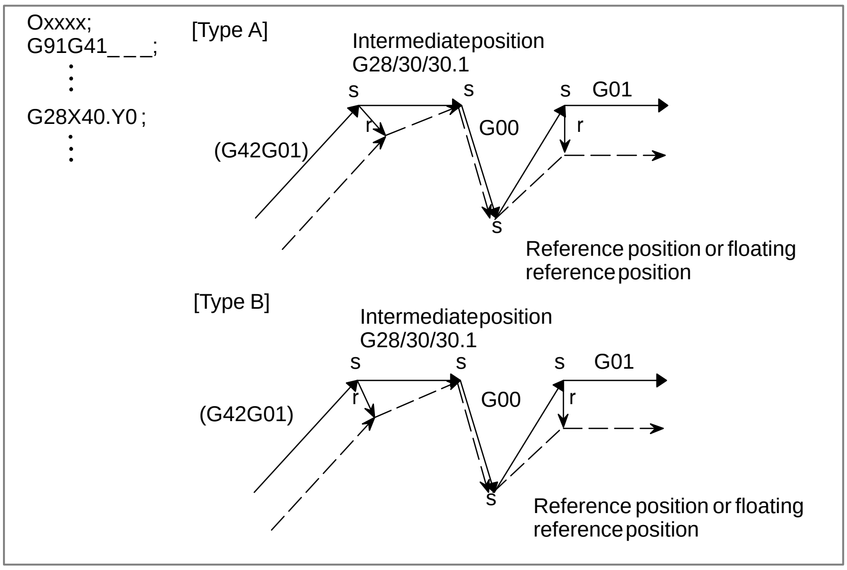


When CCN (bit 2 of parameter No. 5003) = 1

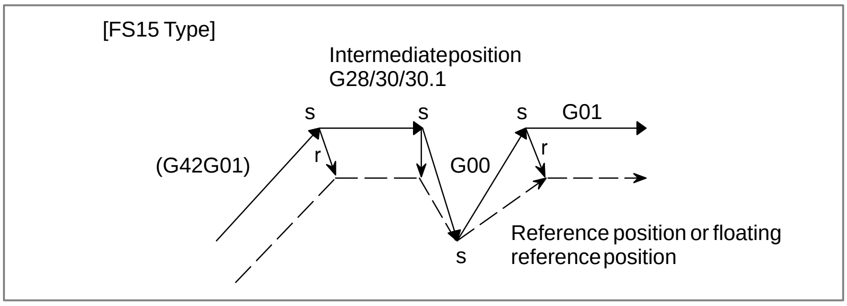


(b) For return by G00

When CCN (bit 2 of parameter No. 5503) = 0



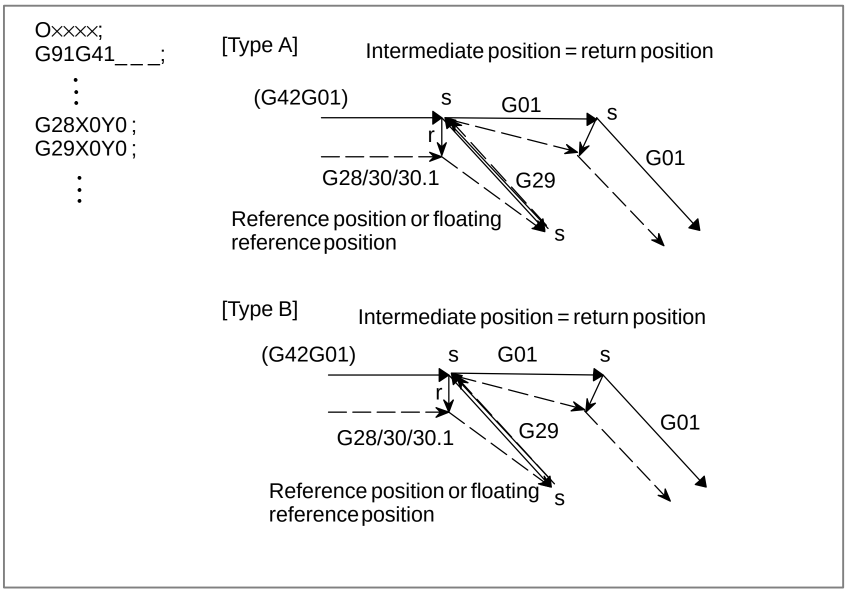
When CCN (bit 2 of parameter No. 5503) = 1



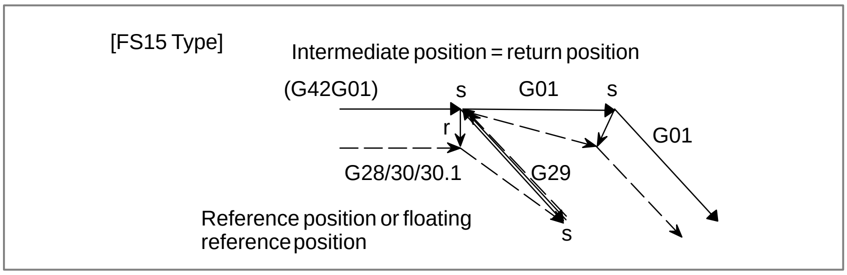
(2) G28, G30, or G30.1, specified in offset mode (with movement to an intermediate position not performed)

(a) For return by G29

When CCN (bit 2 of parameter No. 5503) = 0

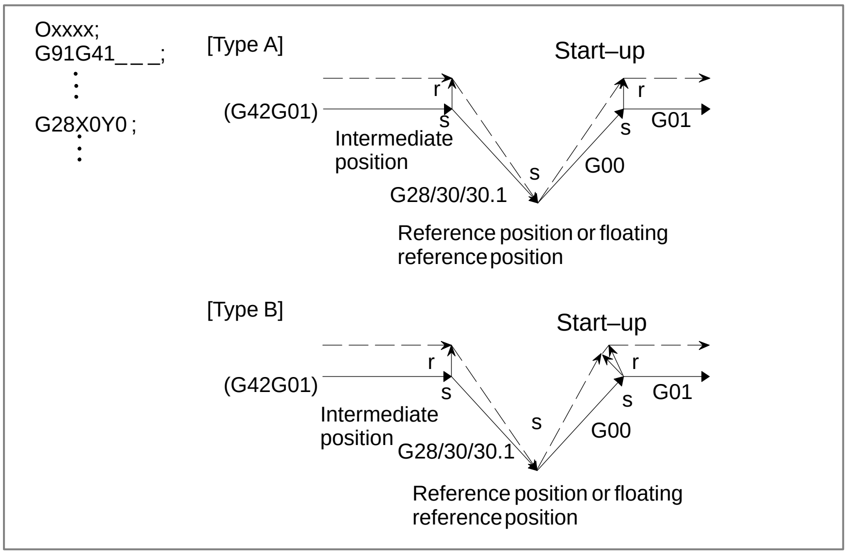


When CCN (bit 2 of parameter No. 5503) = 1

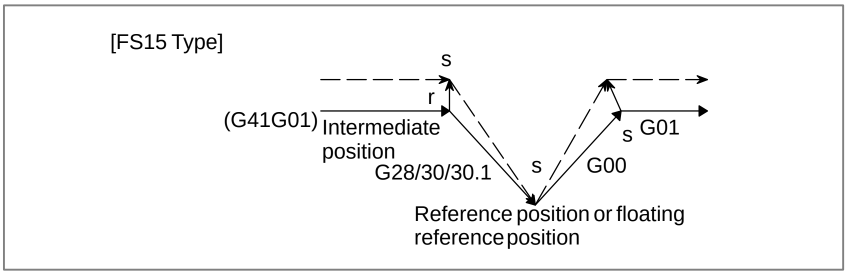


(b) For return by G00

When CCN (bit 2 of parameter No.5503)=0



When CCN (bit 2 of parameter No.5503)=1

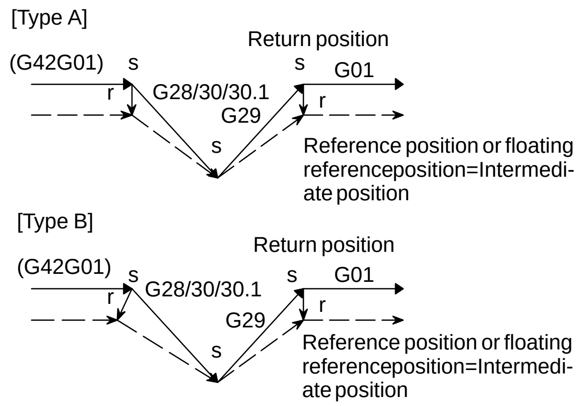


(3) G28, G30, or G30.1, specified in offset mode
(with movement to a reference position not performed)

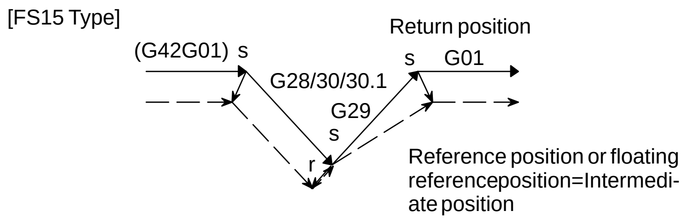
(a) For return by G29

When CCN (bit 2 of parameter No.5503)=0

```
Oxxxx;
G91G41_ _ _;
   :
G28X40.Y–40.;
G29X40.Y40.;
   :
```



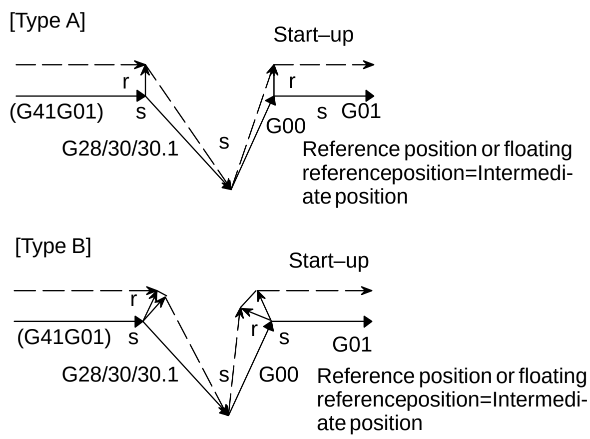
When CCN (bit 2 of parameter No.5503)=1



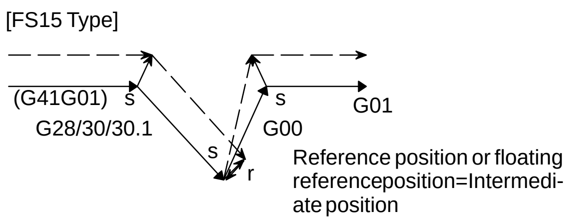
(b) For return by G00

When CCN (bit 2 of parameter No.5503)=0

```
Oxxxx;
G91G41_ _ _;
   :
G28X40.Y–40.;
   :
```



When CCN (bit 2 of parameter No.5503)=1

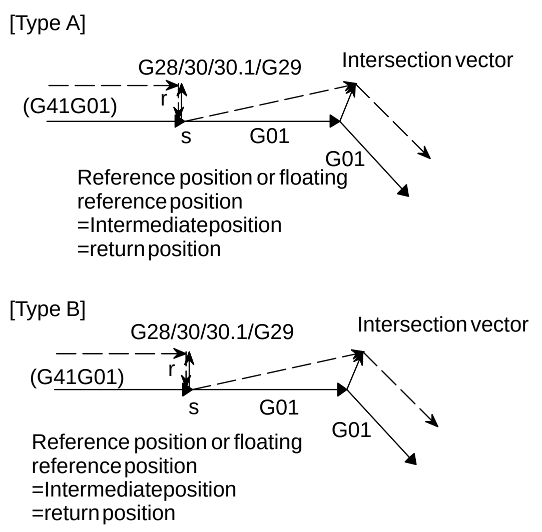


(4) G28, G30, or G30.1 specified in offset mode (with no movement performed)

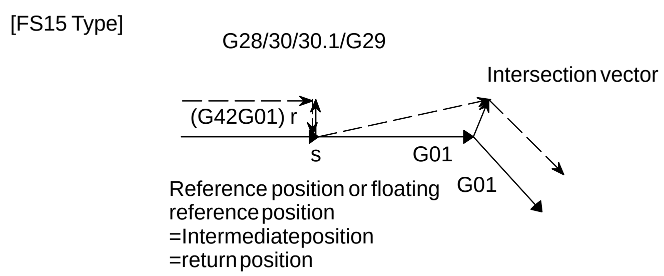
(a) For return by G29

When CCN (bit 2 of parameter No.5503)=0

```
O××××;
G91G41_ _ _;
  :
G28X0Y0;
G29X0Y0;
  :
```



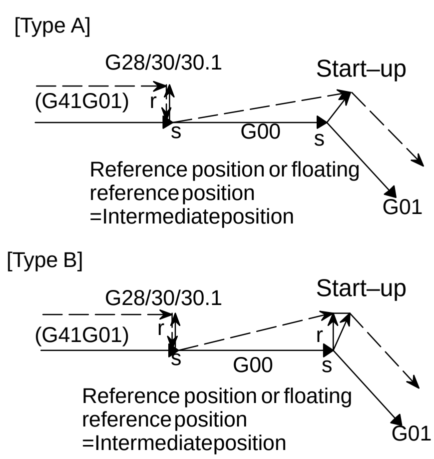
When CCN (bit 2 of parameter No.5503)=1



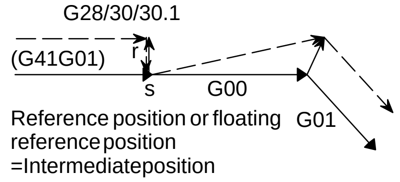
(b) For return by G00

When CCN (bit 2 of parameter No.5503)=0

```
O××××;
G91G41_ _ _;
  :
G28X0Y0;
  :
```



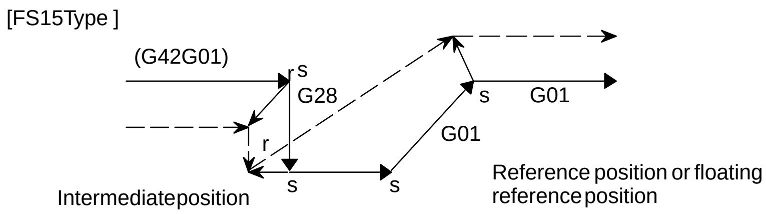
When CCN (bit 2 of parameter No.5503)=1



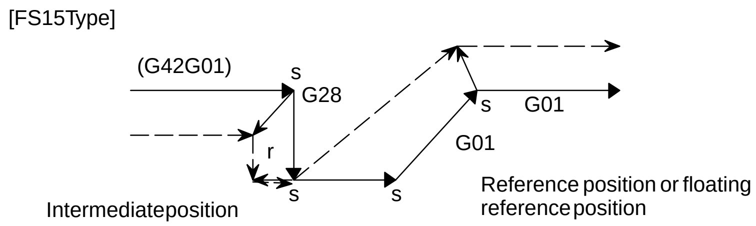
WARNING

1 When a G28, G30, or G30.1 command is specified during all–axis machine lock, a perpendicular offset vector is applied at the intermediate position, and movement to the reference position is not performed; the vector is preserved. Note, however, that even if the FS15 type is used, the vector is canceled only when each–axis machine lock is applied. (The FS15 type preserves the vector even when each–axis machine lock is applied.)

Example1: When CCN (bit 2 of parameter No.5003)=1 and all–axis machine lock is applied

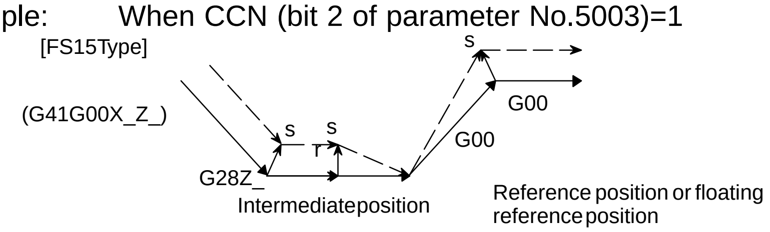


Example2: When CCN (bit 2 of parameter No.5003)=1 and each–axis machine lock is applied



2 When G28, G30, or G30.1 is specified for a compensation axis in cutter compensation mode, the vectors along the other axes are also canceled. (This also applies when CCN (bit 2 of parameter No. 5003) is set to 1. When the FS15 type is used, only the vector along a specified axis is canceled. Note that the FS15 type cancellation differs from the actual FS15 specification in this point.)

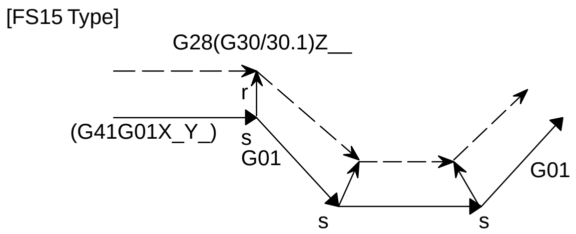
Example: When CCN (bit 2 of parameter No.5003)=1



NOTE

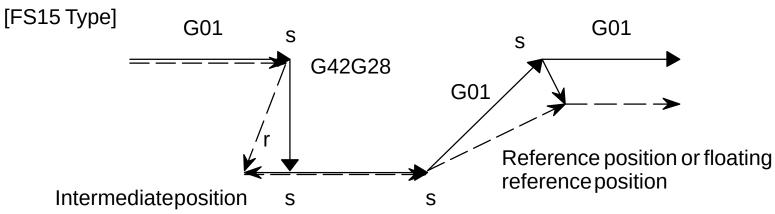
1 When a G28, G30, or G30.1 command specifies an axis that is not in the cutter compensation C plane, a perpendicular vector is generated at the end point of the previous block, and the tool does not move. In the next block, offset mode is automatically resumed (in the same way as when two or more continuous blocks do not specify any move commands).

Example: When CCN (bit 2 of parameter No. 5003) = 1



2 When a G28, G30, or G30.1 block is specified such that the block becomes a start–up block, a vector perpendicular to the move direction is generated at an intermediate position, then subsequently canceled at the reference position. In the next block, an intersection vector is generated.

Example: When CCN (bit 2 of parameter No.5003)=1



- **G29 command in cutter compensation C mode**

When G29 is specified in cutter compensation C mode, an operation of FS15 type is performed if CCN (bit 2 of parameter No. 5003) is set to 1.

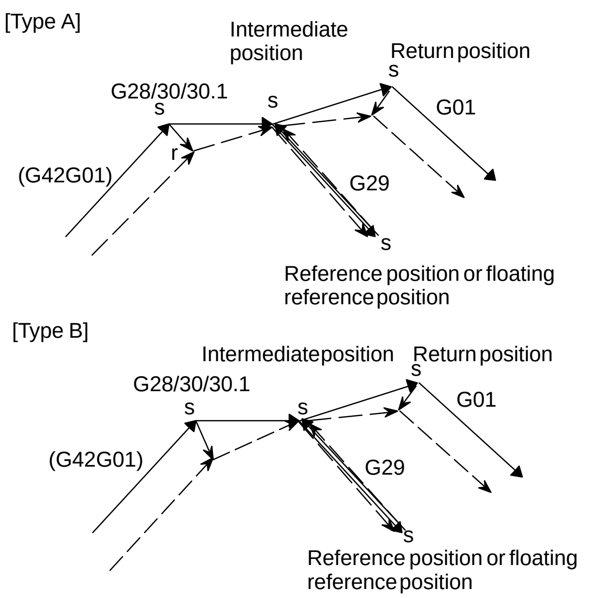
This means that an intersection vector is generated in the previous block, and vector cancellation is performed when a movement to an intermediate position is performed. When movement from the intermediate position to the return position is performed, the vector is restored; an intersection vector is generated between the block and the next block.

(1) G29 specified in offset mode (with movement to both an intermediate position and reference position performed)

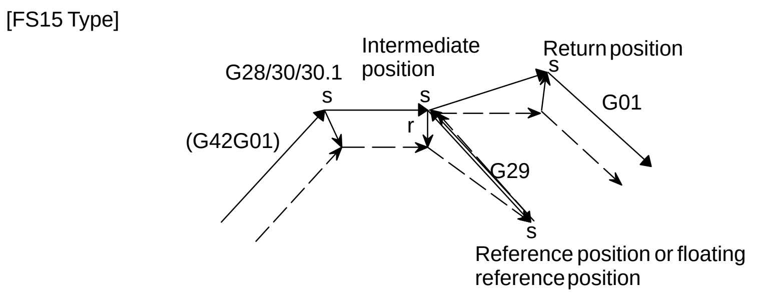
(a) For specification made immediately after automatic reference position return

When CCN (bit 2 of parameter No.5003)=0

```
O××××;
G91G41_ _ _;
   :
G28X40.Y0;
G29X45.Y15.;
   :
```

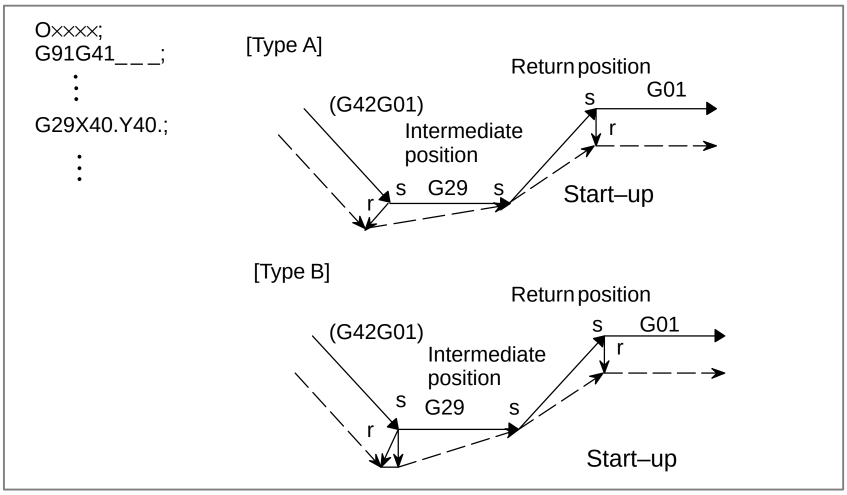


When CCN (bit 2 of parameter No.5003)=1

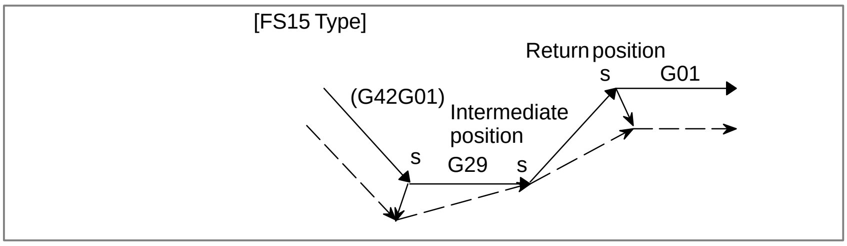


(b) For specification made other than immediately after automatic reference position return

When CCN (bit 2 of parameter No.5003)=0



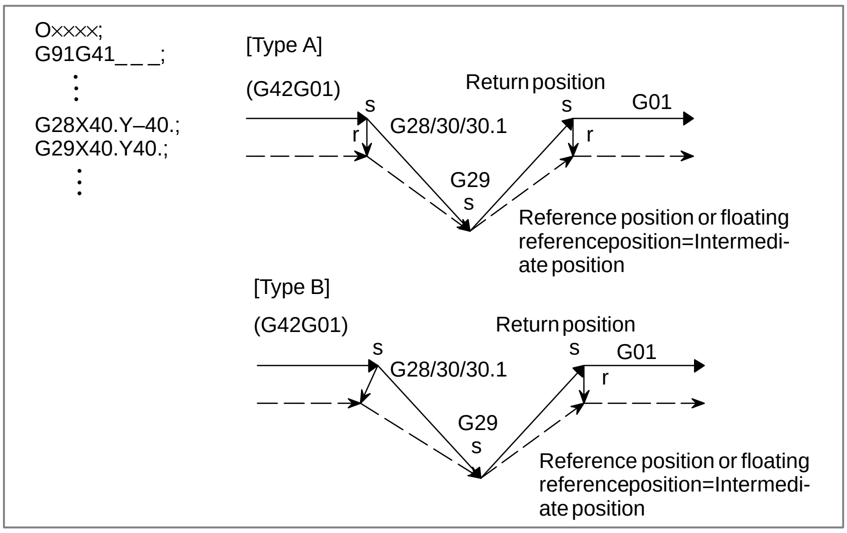
When CCN (bit 2 of parameter No.5003)=1



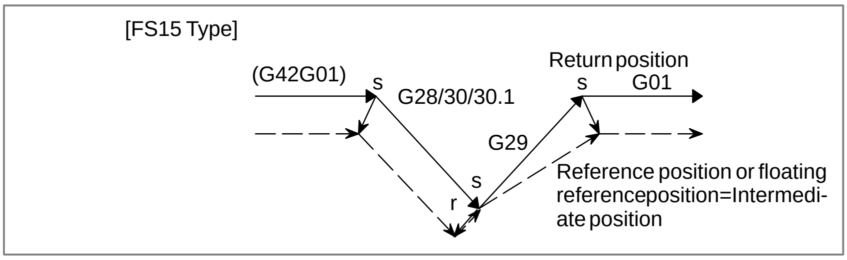
(2) G29 specified in offset mode (with movement to an intermediate position not performed)

(a) For specification made immediately after automatic reference position return

When CCN (bit 2 of parameter No.5003)=0

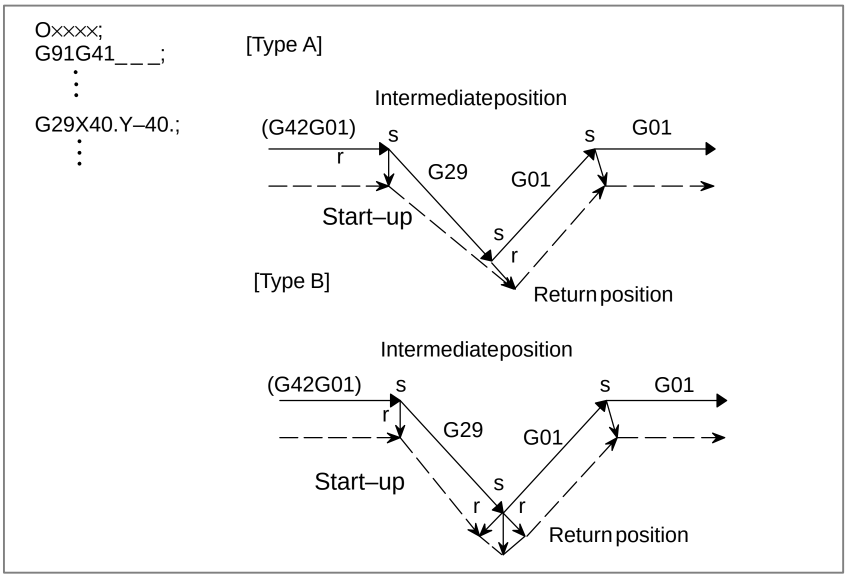


When CCN (bit 2 of parameter No.5003)=1

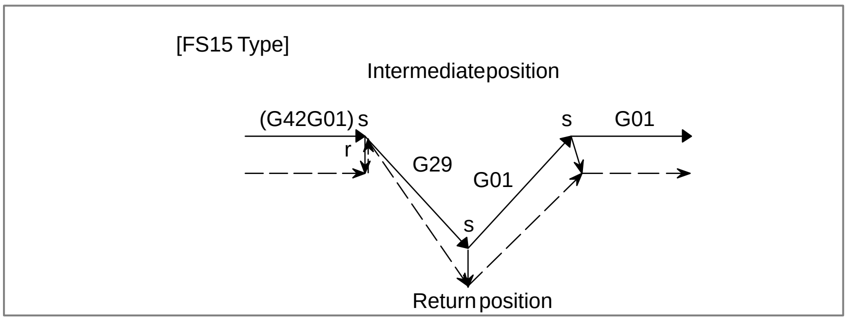


(b) For specification made other than immediately after automatic reference position return

When CCN (bit 2 of parameter No.5003)=0



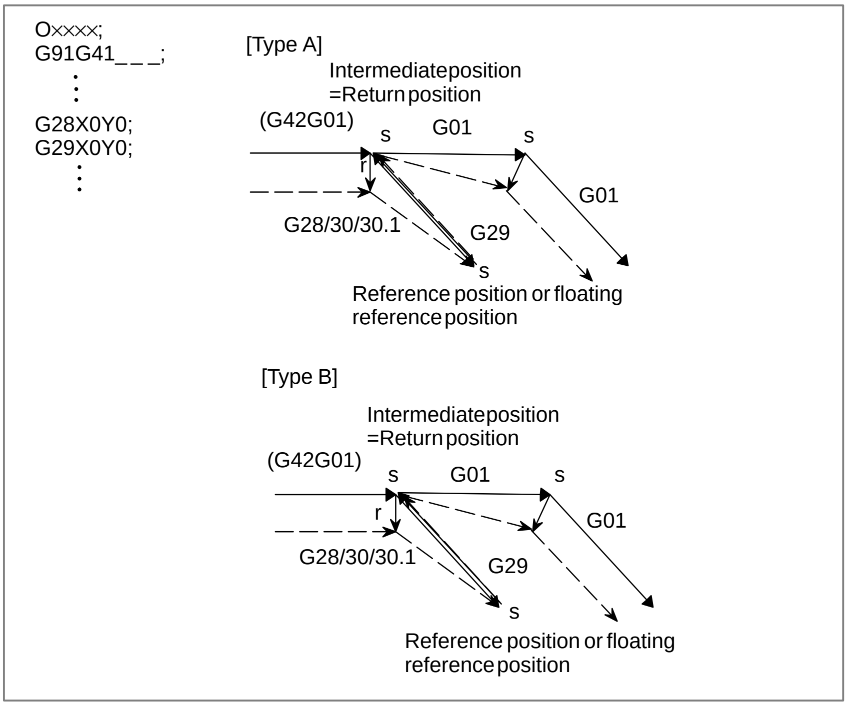
When CCN (bit 2 of parameter No.5003)=1



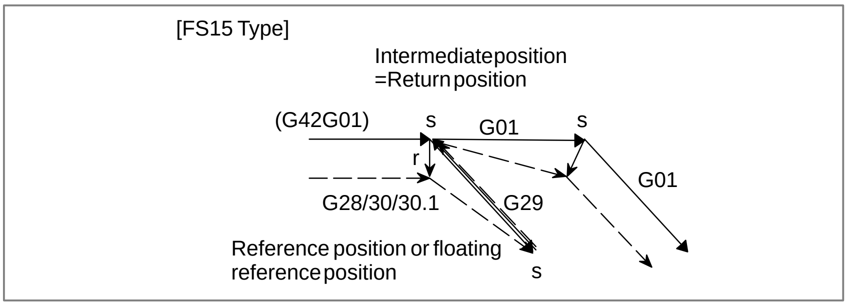
(3) G29 specified in offset mode (with movement to a reference position not performed)

(a) For specification made immediately after automatic reference position return

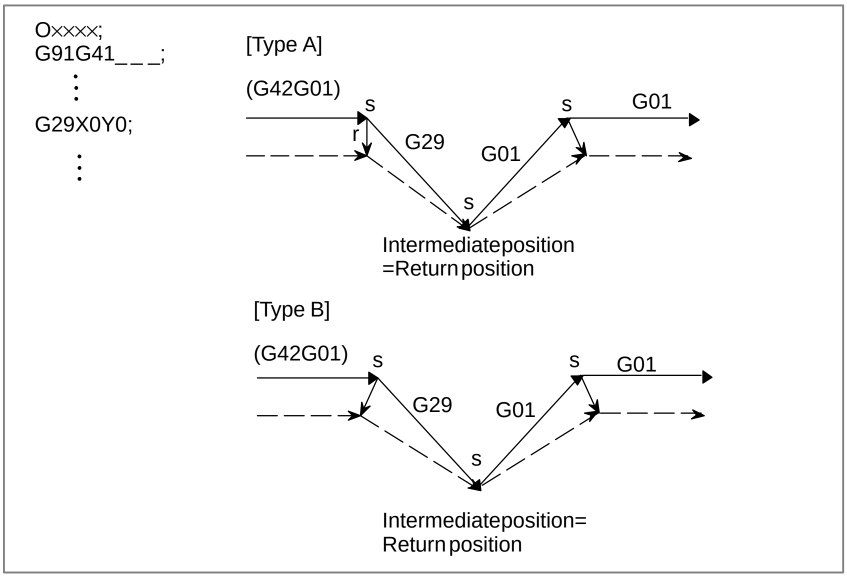
When CCN (bit 2 of parameter No.5003)=0



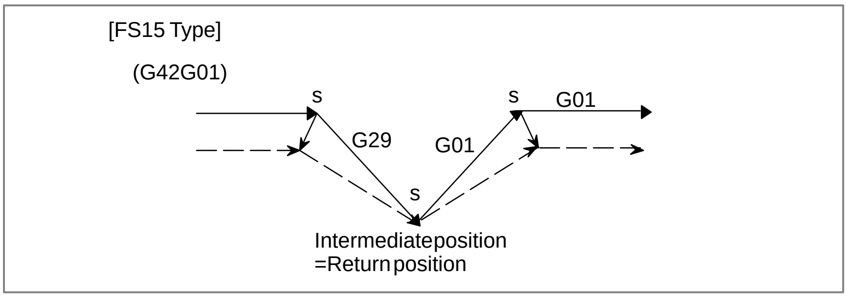
When CCN (bit 2 of parameter No.5003)=1



(b) For specification made other than immediately after automatic reference position return



When CCN (bit 2 of parameter No.5003)=1



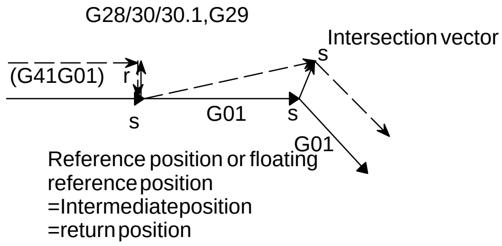
(4) G29 specified in offset mode (with movement to an intermediate position and reference position not performed)

(a) For specification made immediately after automatic reference position return

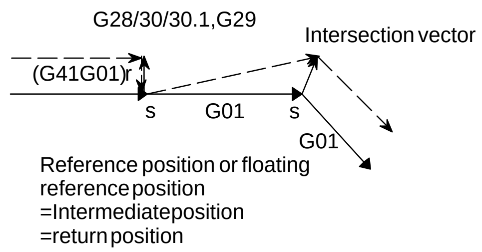
When CCN (bit 2 of parameter No.5003)=0

```
O××××;
G91G41_ _ _;
   :
G28X0Y0;
G29X0Y0;
   :
```

[Type A]

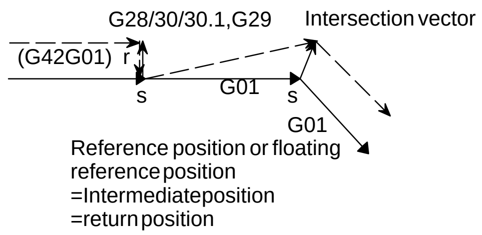


[Type B]



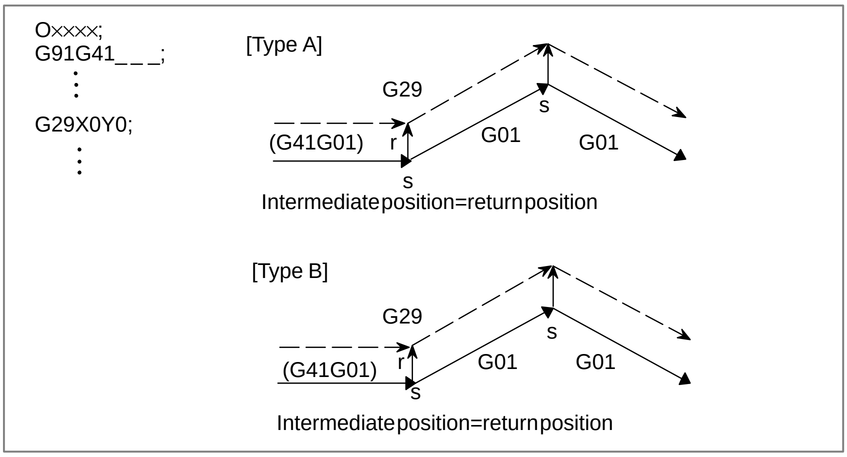
When CCN (bit 2 of parameter No.5003)=1

[FS15 Type]

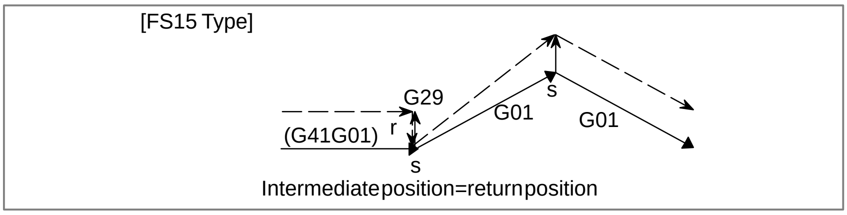


(b) For specification made other than immediately after automatic reference position return

When CCN (bit 2 of parameter No.5003)=0

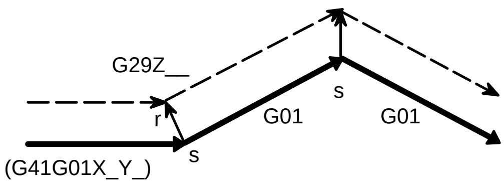


When CCN (bit 2 of parameter No.5003)=1



NOTE

When a G29 command is specified for an axis that is not in the cutter compensation C plane in cutter compensation C mode, a perpendicular vector is generated at the end point of the previous block, and the tool does not move. In the next block, an intersection vector is generated (in the same way as when two or more continuous blocks do not specify any move commands).



12.3.9 Corner Circular Interpolation (G39)

By specifying G39 in offset mode during cutter compensation C, corner circular interpolation can be performed. The radius of the corner circular interpolation equals the compensation value.

Format

```
In offset mode
  G39 ;
or
        ⎧ I_J_ ⎫
  G39 ⎨ I_K_ ⎬ ;
        ⎩ J_K_ ⎭
```

Explanations

- **Corner circular interpolation**

When the command indicated above is specified, corner circular interpolation in which the radius equals compensation value can be performed. G41 or G42 preceding the command determines whether the arc is clockwise or counterclockwise. G39 is a one–shot G code.

- **G39 without I, J, or K**

When G39; is programmed, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the start point of the next block.

- **G39 with I, J, and K**

When G39 is specified with I, J, and K, the arc at the corner is formed so that the vector at the end point of the arc is perpendicular to the vector defined by the I, J, and K values.

Limitations

- **Move command**

In a block containing G39, no move command can be specified.

- **Non–move command**

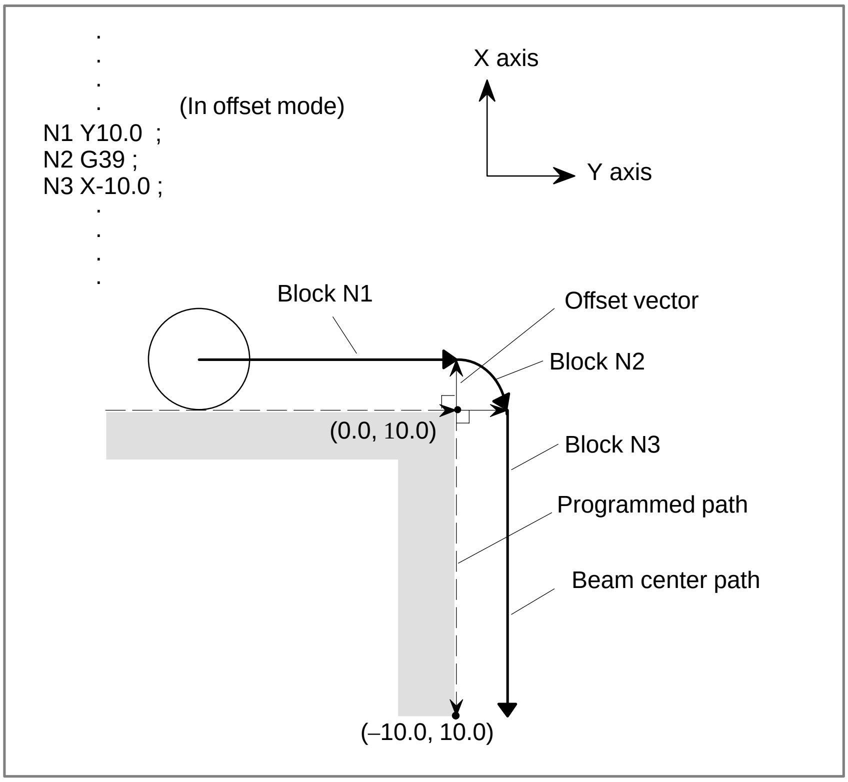
Two or more consecutive non–move blocks must not be specified after a block containing G39 without I, J, or K. (A single block specifying a travel distance of zero is assumed to be two or more consecutive non–move blocks.) If the non–move blocks are specified, the offset vector is temporarily lost. Then, offset mode is automatically restored.

- **Use with edge machining or startup machining**

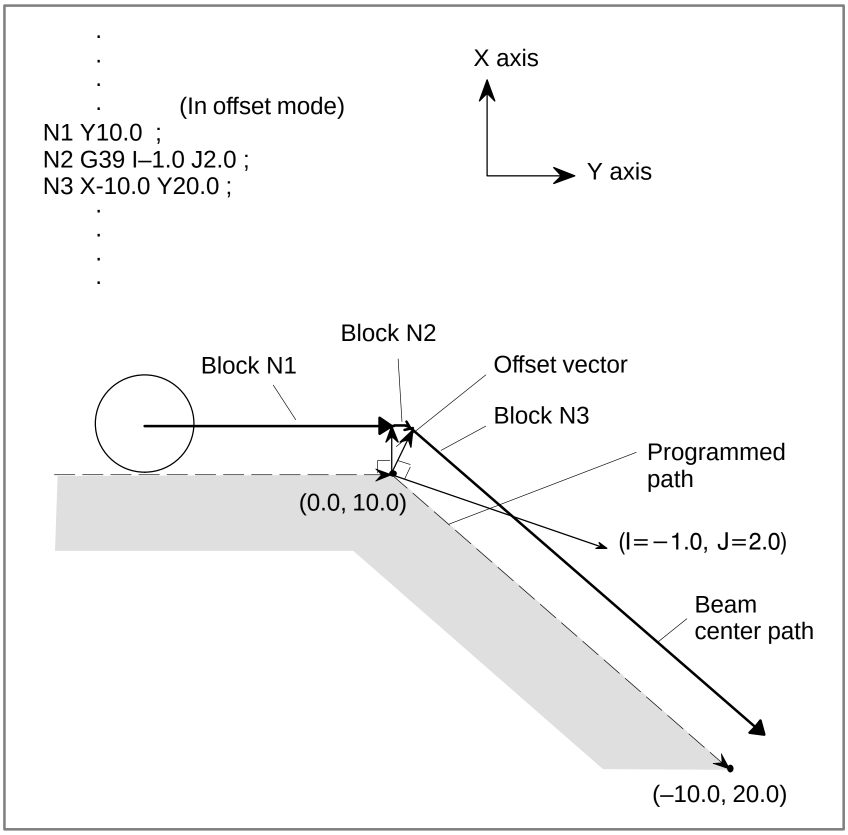
Use with edge machining or startup machining is not supported.

Examples

- **G39 without I, J, or K**



- **G39 with I, J, and K**



12.4 CUTTER COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

Cutter compensation values include beam geometry compensation values and beam wear compensation (Fig.12.4 (a)).

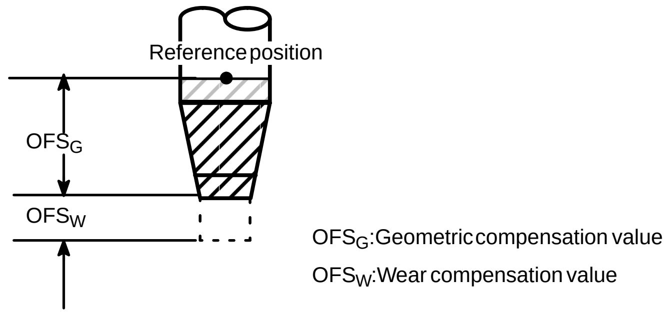


Fig.12.4 (a) Geometric compensation and wear compensation

Cutter compensation values can be entered into CNC memory from the CRT/MDI panel (see section III–11.4.1) or from a program.
A cutter 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 beam length compensation, cutter compensation, or the tool offset.

Explanations

- **Valid range of cutter compensation values**

Table 12.4 (a) shows the valid input range of cutter compensation values.

Table 12.4 (a) The valid input range of cutter compensation value

Incre- ment system	Geometric compensation value		Wear compensation value	
	Metric input	Inch input	Metric input	Inch input
IS–B	±999.999 mm	±99.9999inch	±99.999 mm	±9.9999 inch

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

The memory can hold 32, 64, 99, 200, 400, 499, or 999 tool compensation values (option).
Address D or H is used in the program. The address used depends on which of the following functions is used: tool offset (see II–12.1), or cutter compensation C (see II–12.3).
The range of the number that comes after the address (D or H) depens on the number of cutter compensation values : 0 to 32, 0 to 64, 0 to 99, 0 to 200, 0 to 400, 0 to 499, or 0 to 999.

- **Cutter compensation memory and the cutter compensation value to be entered**

Cutter compensation memory A, B, or C can be used.
The cutter compensation memory determines the cutter compensation values that are entered (set) (Table 12.4 (b)).

Table 12.4 (b) Setting contents cutter compensation memory and cutter compensation value

Cutter compensation value	Cutter compensation memory A	Cutter compensation memory B	Cutter compensation memory C
Beam geometry compensation value for address D	Set beam geometry + beam wear compensation values for addresses D and H (values can be specified with either address).	Set beam geometry compensation values for addresses D and H (values can be specified with either address).	set
Beam geometry compensation value for address H			set
Beam wear compensation for value address D		Set beam wear compensation values for addresses D and H (values can be specified with either address).	set
Beam wear compensation value for address H			set

Format

The programming format depends on which cutter compensation memory is used.

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

Table 12.4 (c) Setting range of Tool compensation memory and Tool compensation value

Variety of cutter compensation memory		Format
A	Cutter compensation value (geometry compensation value+wear compensation value)	G10L11P_R_;
B	Geometry compensation value	G10L10P_R_;
	Wear compensation value	G10L11P_R_;
C	Geometry compensation value for H code	G10L10P_R_;
	Geometry compensation value for D code	G10L12P_R_;
	Wear compensation value for H code	G10L11P_R_;
	Wear compensation value for D code	G10L13P_R_;

P : Number of cutter compensation
R : Cutter compensation value in the absolute command(G90) mode
Value to be added to the specified cutter compensation value in the incremental command(G91) mode (the sum is also a beam compensation value.)

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

12.5 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.

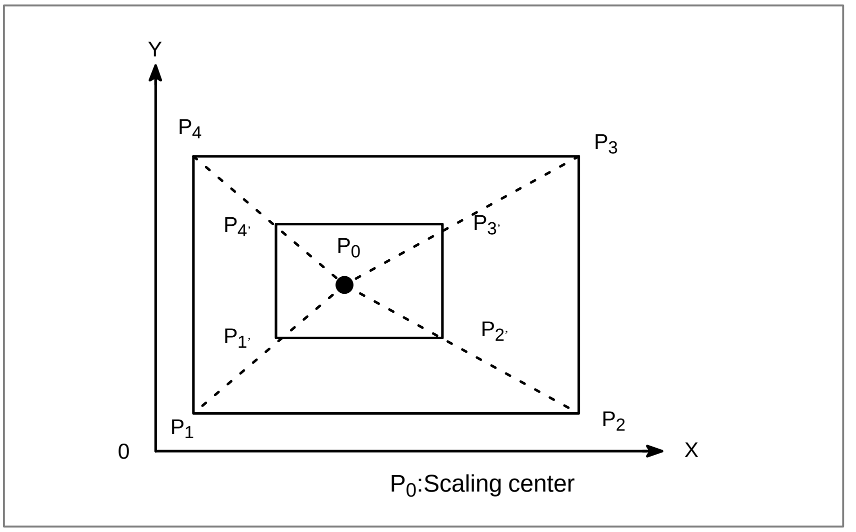


Fig.12.5 (a) Scaling(P_1 P_2 P_3 P_4→$P_{1'}$$P_{2'}$$P_{3'}$$P_{4'}$)

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

WARNING

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 beam 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).

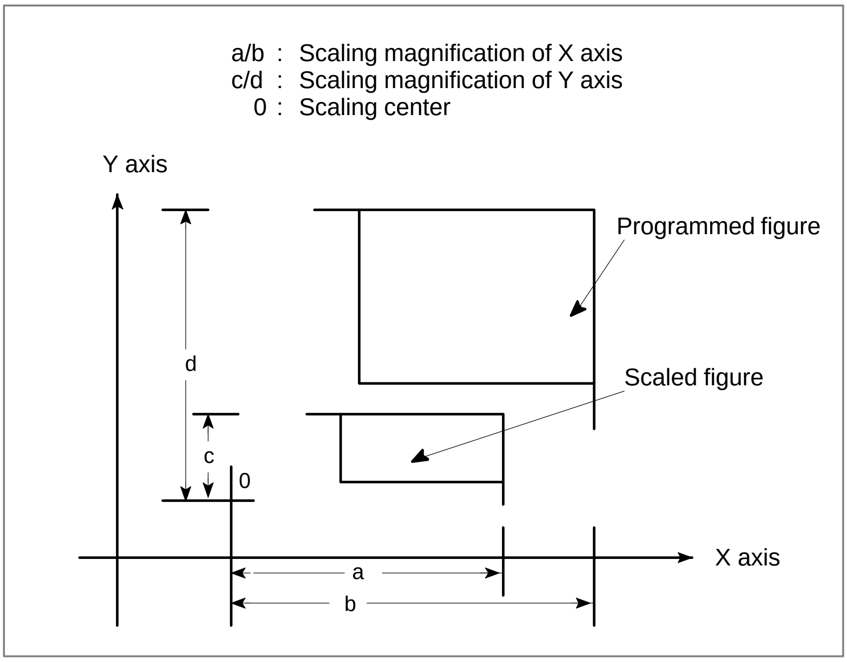


Fig.12.5 (b) Scaling of each axis

- **Scaling of circular interpolation**

Even if different magnifications are applie to each axis in circular interpolation, the beam 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 12.5 (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.

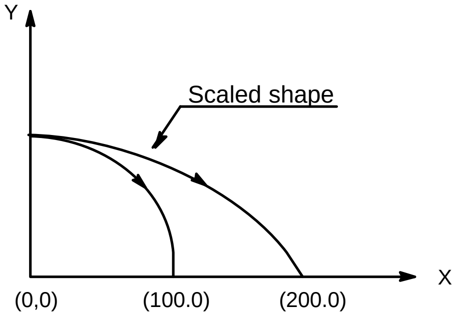


Fig.12.5 (c) Scaling for circular interpolation1

When different magnifications are applied to axes and a circular interpolation is specified with I, J and K, it becomes as following figure 12.5 (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.

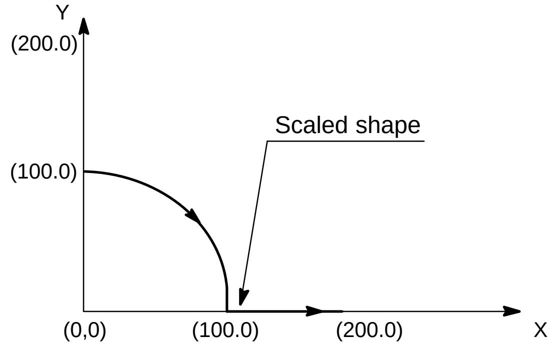


Fig.12.5 (d) Scaling for circular interpolation 2

- **Cutter compensation**

This scaling is not applicable to cutter compensation values, beam length offset values, and tool offset values (Fig.12.5 (e)).

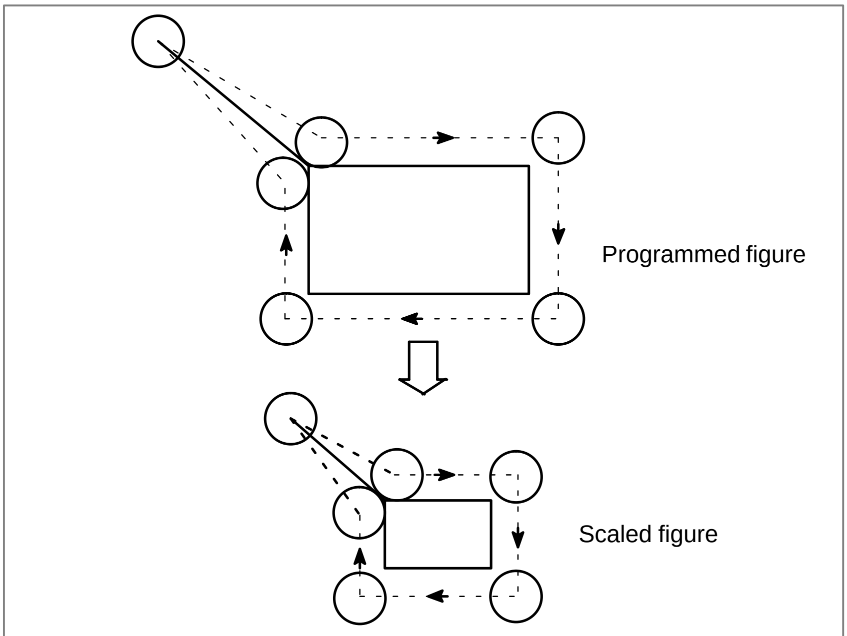


Fig.12.5 (e) Scaling during cutter compensation

- **Invalid scaling**

In manual operation, the travel distance cannot be increased or decreased using the scaling function.

- **Commands related to reference position return and coordinate system**

In scaling mode, G28, G30, or commands related to the coordinate system (G52 to G59) must not be specified. When any of these G codes is necessary, specify it after canceling scaling mode.

WARNING

1 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.
2 Before specifying the G code for reference position return (G27, G28, G29, G30) or coordinate system setting (G92), cancel the scaling mode.
3 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 beam movement by cutter compensation C. See the description of blocks that do not move the beam at II–12.3.3.

NOTE

1 The position display represents the coordinate value after scaling.
2 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;
```

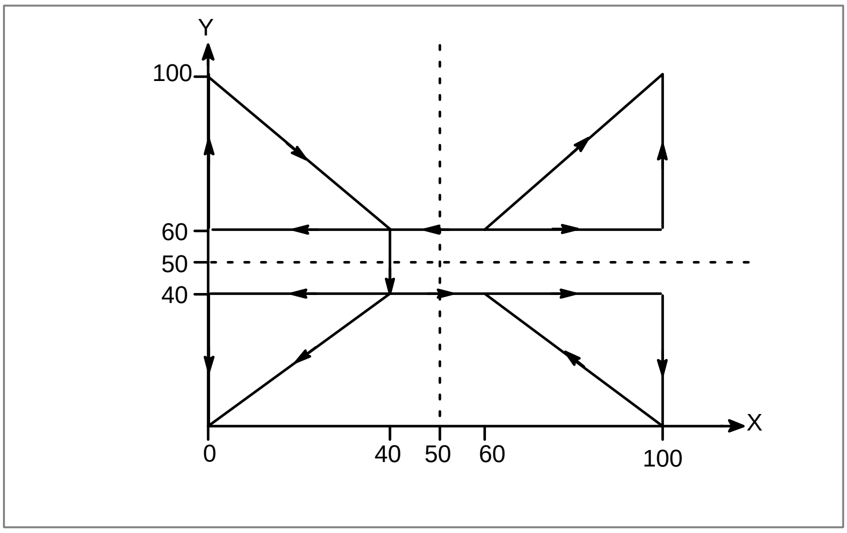


Fig.12.5 (f) Example of a mirror image program

12.6 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.

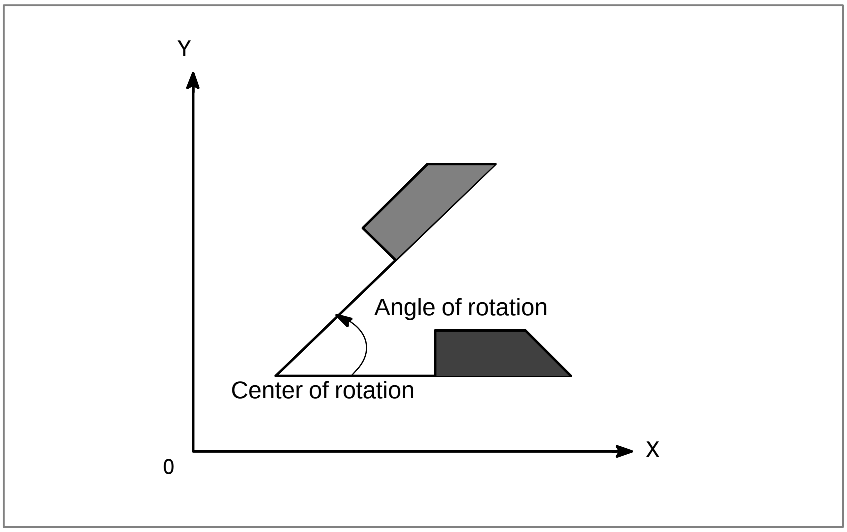


Fig.12.6 (a) Coordinate system rotation

Format

Format

{ **G17** / **G18** / **G19** } **G68** α_β_**R_** ; Start rotation of a coordinate system.

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. Bit 0 of parameter 5400 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

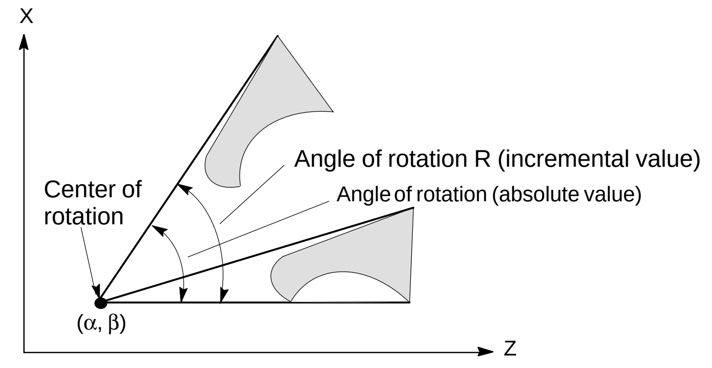


Fig.12.6 (b) Coordinate system rotation

> **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.

- **Incremental command in coordinate system rotation mode**

The center of rotation for an incremental command programmed after G68 but before an absolute command is the nozzle position when G68 was programmed (Fig.12.7 (c)).

- **Center of rotation**

When α_β_ is not programmed, the nozzle position when G68 was programmed is assumed as the center of rotation.

- **Angular displacement**

When R_ is not specified, the value specified in parameter 5410 is assumed as the angular displacement.

- **Coordinate system rotation cancel command**

The G code used to cancel coordinate system rotation (G69) may be specified in a block in which another command is specified.

- **Tool compensation**

Cutter compensation, tool length compensation, tool offset, and other compensation operations are executed after the coordinate system is rotated.

- **Relationship with three–dimensional coordinate conversion (G68, G69)**

As the G code for coordinate system rotation, G84 and G85 can be used in addition to G68 and G69. If the three–dimensional coordinate conversion option is added, G68 and G69 are the G codes for three–dimensional coordinate conversion.

Limitations

- **Commands related to reference position return and the coordinate system**

In coordinate system rotation mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling coordinate system rotation mode.

- **Nozzle attitude conversion**

In coordinate conversion, the two axes on the selected plane are rotated. When this function is used with a three–dimensional machine, rotation is not applied to the nozzle attitude. To convert the coordinate system for the nozzle attitude, use the three–dimensional conversion function.

- **Incremental command**

The first move command after the coordinate system rotation cancel command (G69) must be specified with absolute values. If an incremental move command is specified, correct movement will not be performed.

Explanations

Absolute/Incremental position commands

```
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 ;
```

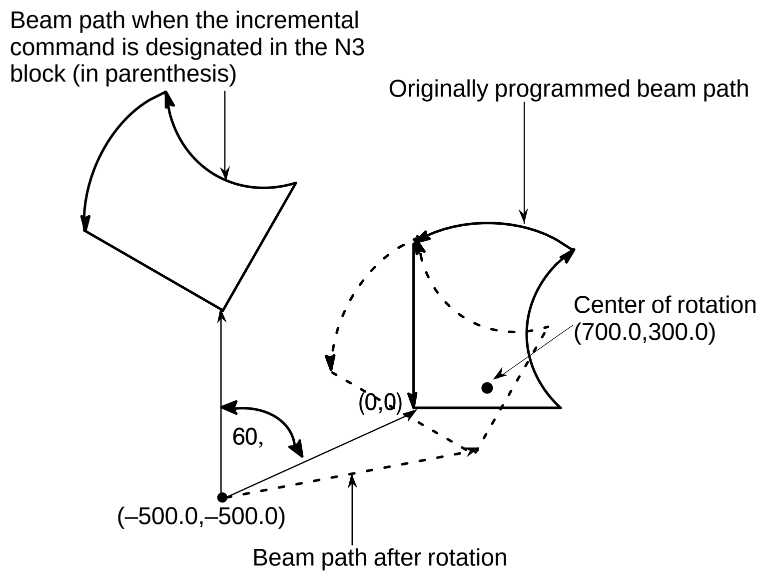


Fig.12.6 (c) Absolute/incremental command during coordinate system rotation

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 ;
```

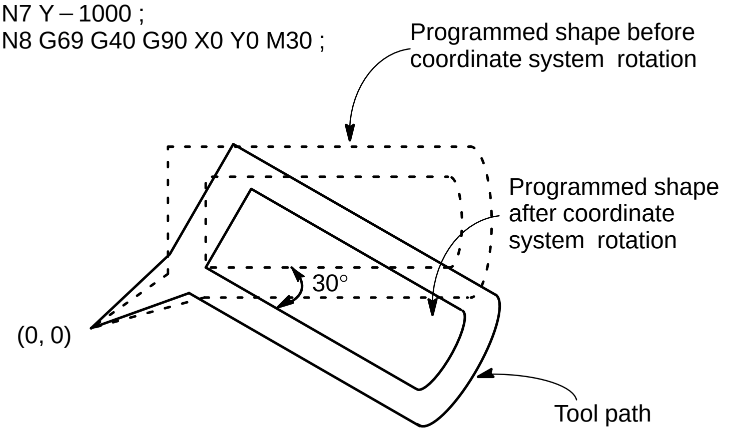


Fig.12.6 (d) Cutter compensation C and coordinate system rotation

- **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.12.6(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 ;
```

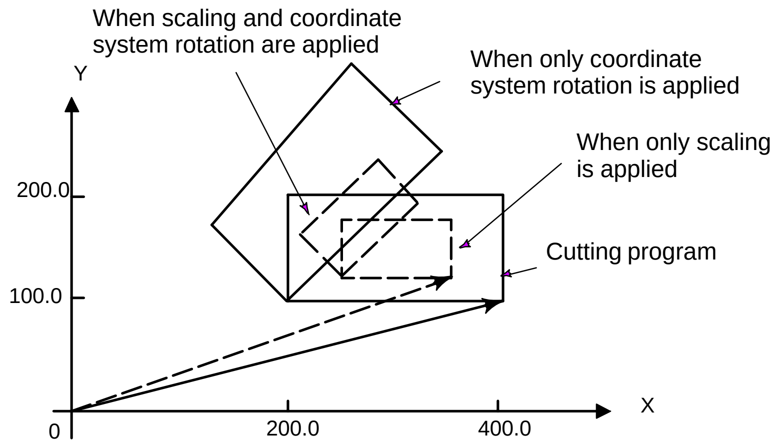


Fig.12.6 (e) Scaling and coordinate system rotation in cutter compensation C mode

- **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–10.0 ;
  X4.142 ;
  X7.071 Y–7.071 ;
  G40 ;
  M99 ;
```

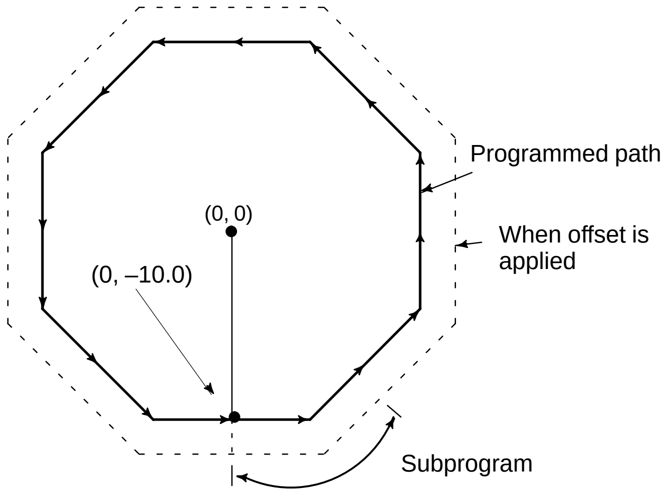


Fig.12.6 (f) Coordinate system rotation command

12.7 NORMAL DIRECTION CONTROL (G40.1, G41.1, G42.1 OR G150, G151, G152)

When a beam with a rotation axis (C–axis) is moved in the XY plane during cutting, the normal direction control function can control the beam so that the C–axis is always perpendicular to the beam path (Fig.12.7 (a)).

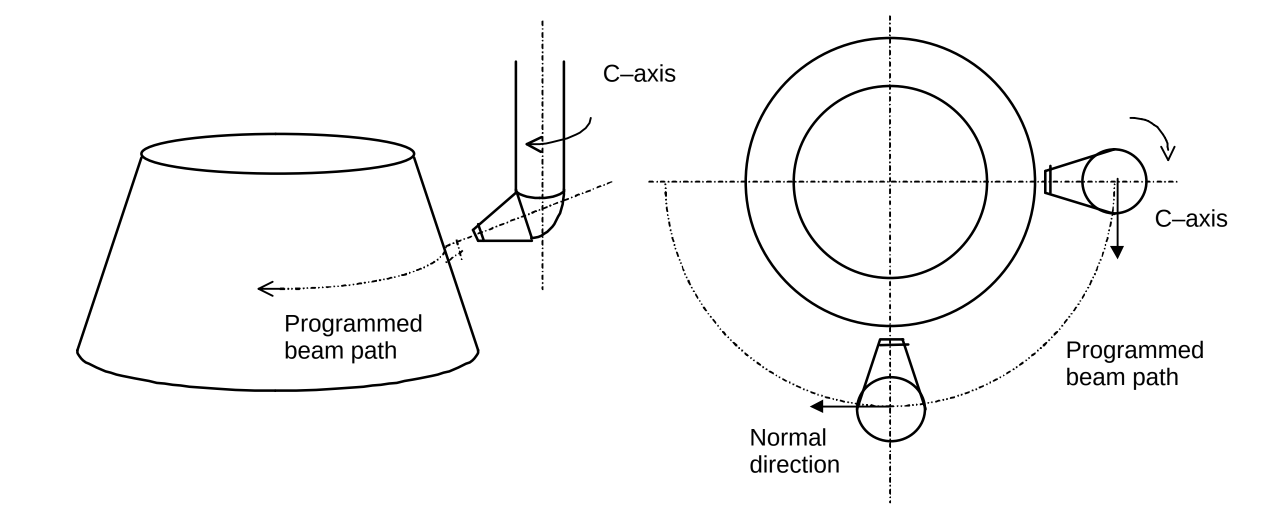


Fig.12.7 (a) Nozzle Operation (Example)

Format

G code	Function	Explanation
G41.1 or **G151**	Normal direction control left	If the workpiece is to the right of the nozzle path looking toward the direction in which the nozzle advances, the normal direction control left (G41.1 or G151) function is specified. After G41.1 (or G151) or G42.1 (or G152) is specified, the normal direction control function is enabled (normal direction control mode). When G40.1 (or G150) is specified, the normal direction control mode is canceled.
G42.1 or **G152**	Normal direction control right	
G40.1 or **G150**	Normal direction control cancel	

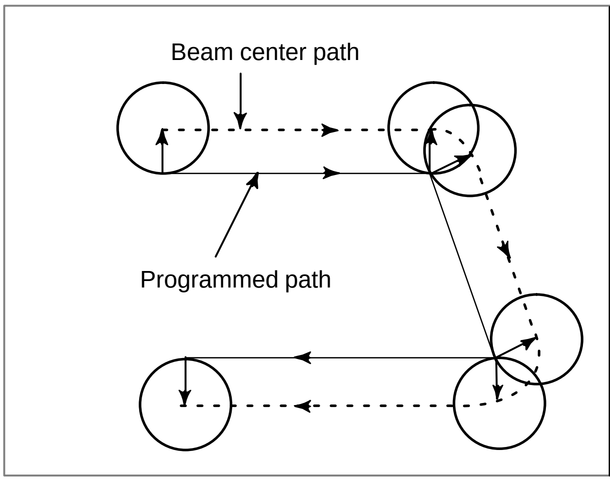


Fig.12.7 (b) Normal direction control left (G41.1)

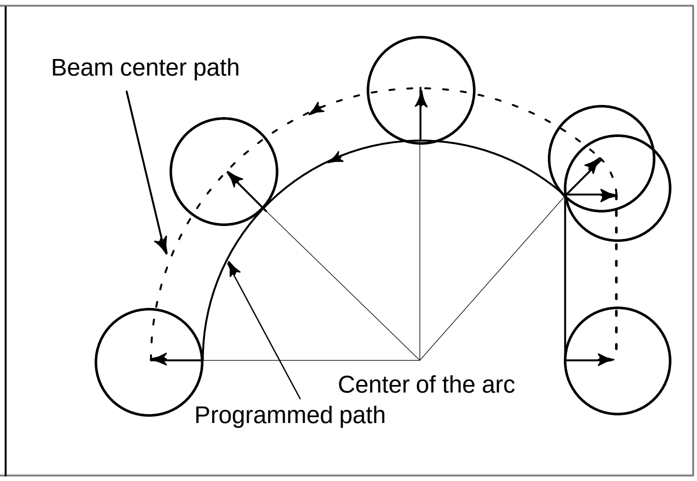


Fig.12.7 (c) Normal direction control right (G42.1)

Explanations

• Angle of the C axis

When viewed from the center of rotation around the C–axis, the angular displacement about the C–axis is determined as shown in Fig.12.7 (d). The positive side of the X–axis is assumed to be 0 , the positive side of the Y–axis is 90°, the negative side of the X–axis is 180°, and the negative side of the Y–axis is 270°.

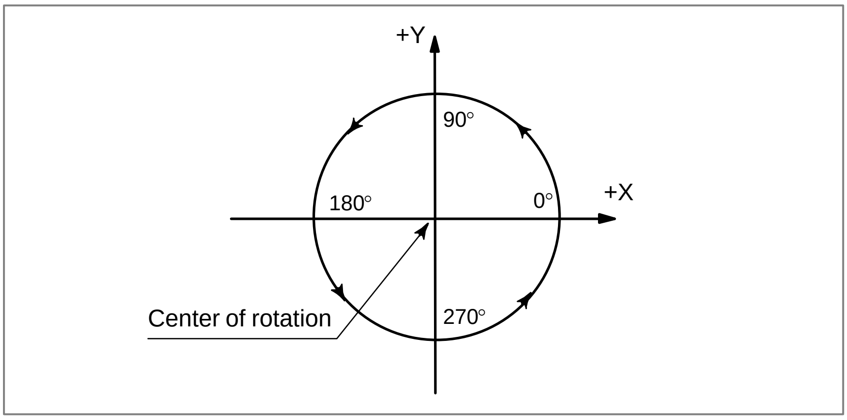


Fig.12.7 (d) Angle of the C axis

• Normal direction control of the C axis

When the cancel mode is switched to the normal direction control mode, the C–axis becomes perpendicular to the beam path at the beginning of the block containing G41.1 or G42.1.
In the interface between blocks in the normal direction control mode, a command to move the beam is automatically inserted so that the C–axis becomes perpendicular to the beam path at the beginning of each block. The beam is first oriented so that the C–axis becomes perpendicular to the beam path specified by the move command, then it is moved along the X– and Y axes.
In the cutter compensation mode, the beam is oriented so that the C–axis becomes perpendicular to the beam path created after compensation.
In single–block operation, the beam is not stopped between a command for rotation of the beam and a command for movement along the X– and Y–axes. A single–block stop always occurs after the beam is moved along the X– and Y–axes.

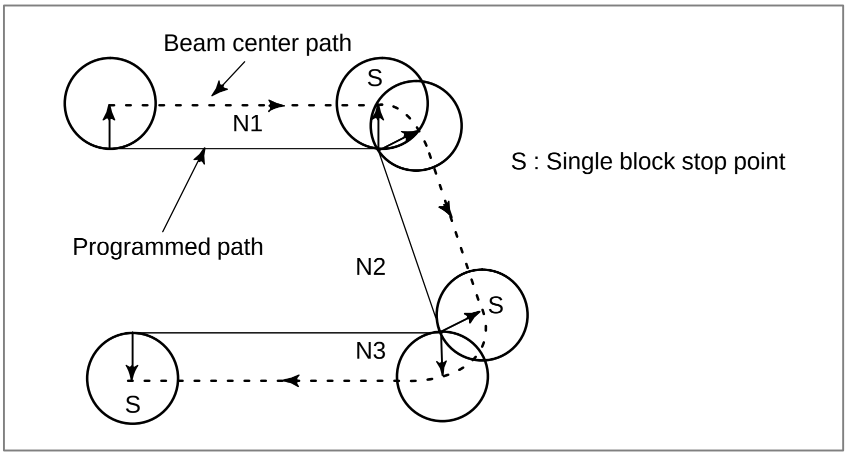


Fig.12.7 (e) Point at which a Single–Block Stop Occurs in the Normal Direction Control Mode

Before circular interpolation is started, the C–axis is rotated so that the C–axis becomes normal to the arc at the start point. During circular interpolation, the beam is controlled so that the C–axis is always perpendicular to the beam path determined by circular interpolation.

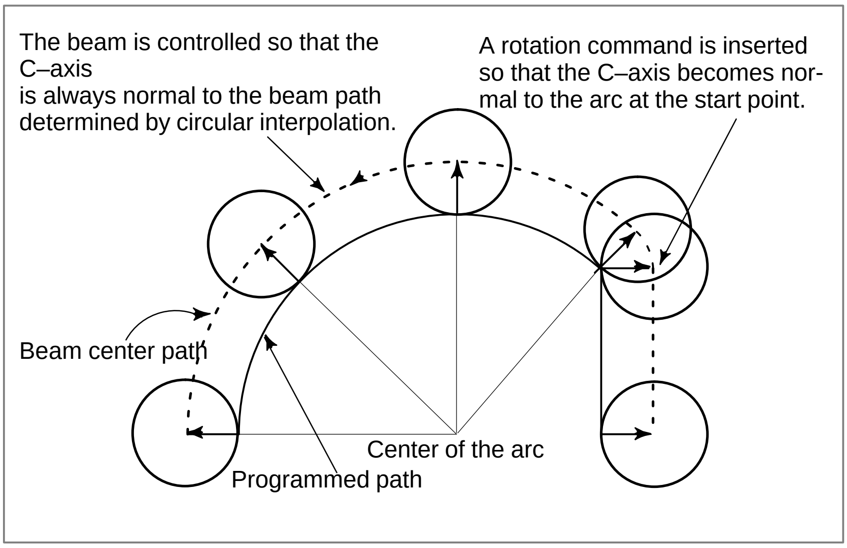


Fig.12.7 (f) Normal direction control of the circular interpolation

> **NOTE**
> During normal direction control, the C axis always rotates through an angle less than 180 deg. I.e., it rotates in whichever direction provides the shorter route.

- **C axis feedrate**

Movement of the tool inserted at the beginning of each block is executed at the feedrate set in parameter 5481. If dry run mode is on at that time, the dry run feedrate is applied. If the tool is to be moved along the X–and Y–axes in rapid traverse (G00) mode, the rapid traverse feedrate is applied.

The federate of the C axis during circular interpolation is defined by the following formula.

$$F \times \frac{\text{Amount of movement of the C axis (deg)}}{\text{Length of arc (mm or inch)}} \text{ (deg/min)}$$

F : Federate (mm/min or inch/min) specified by the corresponding block of the arc

Amount of movement of the C axis : The difference in angles at the beginning and the end of the block.

> **NOTE**
> If the federate of the C axis exceeds the maximum cutting speed of the C axis specified to parameter No. 1422, the federate of each of the other axes is clamped to keep the federate of the C axis below the maximum cutting speed of the C axis.

- **Normal direction control axis**

A C–axis to which normal–direction control is applied can be assigned to any axis with parameter No. 5480.

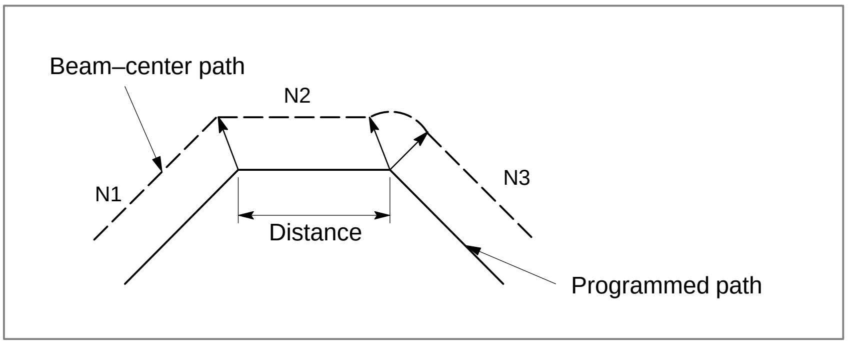
- **Angle for which figure insertion is ignored**

When the rotation angle to be inserted, calculated by normal–direction control, is smaller than the value set with parameter No. 5482, the corresponding rotation block is not inserted for the axis to which normal–direction control is applied. This ignored rotation angle is added to the next rotation angle to be inserted, the total angle being subject to the same check at the next block.
If an angle of 360 degrees or more is specified, the corresponding rotation block is not inserted.
If an angle of 180 degrees or more is specified in a block other than that for circular interpolation with a C–axis rotation angle of 180 degrees or more, the corresponding rotation block is not inserted.

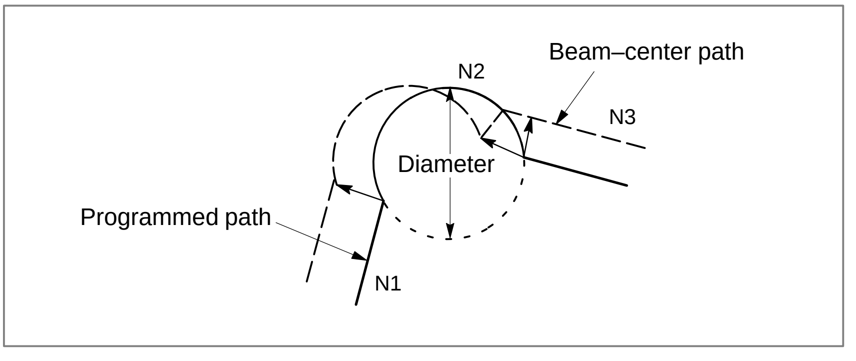
- **Movement for which arc insertion is ignored**

Specify the maximum distance for which machining is performed with the same normal direction as that of the preceding block.

- **Linear movement**
  When distance N2, shown below, is smaller than the set value, machining for block N2 is performed using the same direction as that for block N1.



- **Circular movement**
  When the diameter of block N2, shown below, is smaller than the set value, machining for block N2 is performed using the same normal direction as that for block N1. The orientation of the axis to which normal–direction control is applied, relative to the normal direction of block N2, does not change as machining proceeds along the arc.

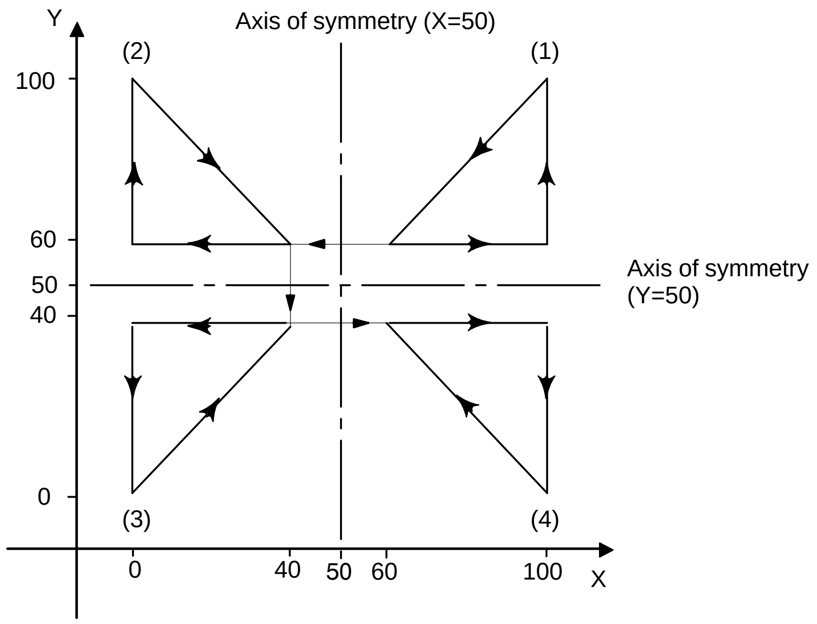


NOTE

1. Do not specify any command to the C axis during normal direction control. Any command specified at this time is ignored.
2. Before processing starts, it is necessary to correlate the workpiece coordinate of the C axis with the actual position of the C axis on the machine using the coordinate system setting (G92) or the like.
3. The helical cutting option is required to use this function. Helical cutting cannot be specified in the normal direction control mode.
4. Normal direction control cannot be performed by the G53 move command.
5. The C–axis must be a rotation axis.

12.8 PROGRAMMABLE MIRROR IMAGE (G50.1, G51.1)

A mirror image of a programmed command can be produced with respect to a programmed axis of symmetry (Fig.12.8 (a)).



(1) Original image of a programmed command

(2) Image symmetrical about a line parallel to the Y–axis and crossing the X–axis at 50

(3) Image symmetrical about point (50, 50)

(4) Image symmetrical about a line parallel to the X–axis and crossing the Y–axis at 50

Fig.12.8 (a) Programmable Mirror image

Format

G51.1 IP_ ; Setting a programmable image

A mirror image of a command specified in these blocks is produced with respect to the axis of symmetry specified by G51.1IP _;.

G50.1 IP_ ; Canceling a programmable mirror image

IP_ : Point (position) and axis of symmetry for producing a mirror image when specified with G51.1.
Axis of symmetry for producing a mirror image when specified with G50.1. Point of symmetry is not specified.

Explanations

- **Mirror image by setting**

If the programmable mirror image function is specified when the command for producing a mirror image is also selected by a CNC external switch or CNC setting (see III–4.7), the programmable mirror image function is executed first.

- **Mirror image on a single axis in a specified plane**

Applying a mirror image to one of the axes on a specified plane changes the following commands as follows :

Command	Explanation
Circular command	G02 and G03 are interchanged.
Cutter compensation	G41 and G42 are interchanged.
Coordinate rotation	CW and CCW (directions of rotation) are interchanged.

Limitations

- **Scaling/coordinate system rotation**

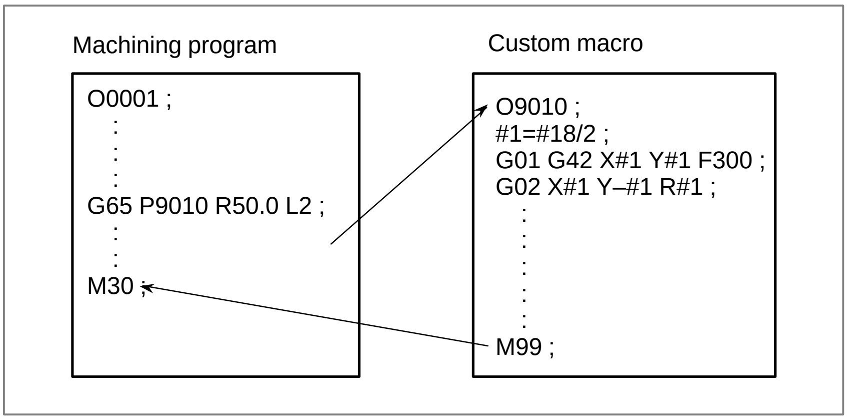
Processing proceeds from program mirror image to scaling and coordinate rotation in the stated order. The commands should be specified in this order, and, for cancellation, in the reverse order. Do not specify G50.1 or G51.1 during scaling or coordinate rotation mode.

- **Commands related to reference position return and coordinate system**

In programmable mirror image mode, G codes related to reference position return (G27, G28, G29, G30, etc.) and those for changing the coordinate system (G52 to G59, G92, etc.) must not be specified. If any of these G codes is necessary, specify it only after canceling the programmable mirror image mode.

13 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.



13.1 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. General–purpose programming languages 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 13.1 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 (#199) #500 – #531 (#999)	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. As an option, common variables #150 to #199 and #532 to #999 are also available. However, when these values are using.
#1000 –	System variables	System variables are used to read and write a variety of NC data items such as the current position and cutter compensation values.

> **NOTE**
> Common variables #150 to #199 and #532 to #999 are optional.

- **Range of variable values**

Local and common variables can have value 0 or a value in the following ranges :
-10^{47} to -10^{-29}
0
10^{-29} to 10^{47}
If the result of calculation turns out to be invalid, an P/S 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

- **Display the valiable**

```
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 ] [ MENU ] [  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, sequence numbers, and optional block skip numbers cannot be referenced using variables.

Example:

Variables cannot be used in the following ways:

```
O#1;
/#2G00X100.0;
N#3Y200.0;
```

13.2 SYSTEM VARIABLES

System variables can be used to read and write internal NC data such as cutter 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 13.2 (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 (function) (B–63523EN–1).

- **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 13.2 (b) System variables for tool compensation memory A

Compensation number	System variable
1 : 200 : 999	#10001(#2001) : #10200(#2200) : #10999

Table 13.2 (c) System variables for cutter compensation memory B

Compensation number	Geometry compensation	Wear compensation
1 : 200 : 999	#11001 (#2201) : #11200 (#2400) : #11999	#10001 (#2001) : #10200 (#2200) : #10999

Table 13.2 (d) System variables for tool compensation memory C

Compensation number	Beam length compensation (H)		Cutter compensation (D)	
	Geometric compensation	Wear compensation	Geomet-ric com-pensation	Wear com-pensation
1 : 200 : 999	#11001(#2201) : #11201(#2400) : #11999	#10001(#2001) : #10201(#2200) : #10999	#13001 : : #13999	#12001 : : #12999

- **Macro alarms**

Table 13.2 (e) 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."

- **Stop with a message**

Execution of the program can be stopped, and then a message can be displayed.

Variable number	Function
#3006	When "#3006=1 (MESSAGE);" is commanded in the macro, the program executes blocks up to the immediately previous one and then stops. When a message of up to 26 characters, which is enclosed by a control–in character ("(") and control–out character (")"), is programmed in the same block, the message is displayed on the external operator message screen.

- **Time information**

Time information can be read and written.

Table 13.2 (f) 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 2147483648 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 9544.371767 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, February 28, 2002 is represented as 20020228.
#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 13.2 (g) 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 13.2 (h) 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.

- **Settings**

Settings can be read and written. Binary values are converted to decimals.

#3005

	#15	#14	#13	#12	#11	#10	#9	#8
Setting							FCV	

	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC

#9 (FCV) : Whether to use the FS15 tape format conversion capability
#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–4.7)
The value obtained in binary is converted into decimal notation.

#3007

	#7	#6	#5	#4	#3	#2	#1	#0
Setting	8th axis	7th axis	6th axis	5th axis	4th axis	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 13.2 (i) 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 13.2 (j) System variables for modal information

Variable number	Function	
#4001	G00, G01, G02, G03, G33	(Group 01)
#4002	G17, G18, G19	(Group 02)
#4003	G90, G91	(Group 03)
#4004		(Group 04)
#4005		(Group 05)
#4006	G20, G21	(Group 06)
#4007	G40, G41, G42	(Group 07)
#4008		(Group 08)
#4009		(Group 09)
#4010		(Group 10)
#4011	G50, G51	(Group 11)
#4012	G65, G66, G67	(Group 12)
#4013		(Group 13)
#4014	G54–G59	(Group 14)
#4015	G61–G64	(Group 15)
#4016	G68, G69	(Group 16)
:	:	:
#4022		(Group 22)
#4102	B code	
#4107	D code	
#4109	F code	
#4111	H code	
#4113	M code	
#4114	Sequence number	
#4115	Program number	
#4119	S code	
#4120	T code	
#4130	P code (number of the currently selected additional workpiece coordinate system)	

Example:

When #1=#4001; is executed, the resulting value in #1 is 17, 18, or 19. If the specified system variable for reading modal information corresponds to a G code group which cannot be used, a P/S alarm is issued.

- **Current position**

Position information cannot be written but can be read.

Table 13.2 (k) System variables for position information

<table>
<tr><th>Variable number</th><th>Position information</th><th>Coordinate system</th><th>Cutter compensation value</th><th>Read operation during movement</th></tr>
<tr><td>#5001–#5008</td><td>Block end point</td><td>Workpiece coordinate system</td><td>Not included</td><td>Enabled</td></tr>
<tr><td>#5021–#5028</td><td>Current position</td><td>Machine coordinate system</td><td rowspan="3">Included</td><td rowspan="2">Disabled</td></tr>
<tr><td>#5041–#5048</td><td>Current position</td><td rowspan="2">Workpiece coordinate system</td></tr>
<tr><td>#5061–#5068</td><td>Skip signal position</td><td>Enabled</td></tr>
<tr><td>#5101–#5108</td><td>Deviated servo position</td><td></td><td></td><td>Disabled</td></tr>
</table>

- The first digit (from 1 to 8) represents an axis number.
- The nozzle 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 13.2 (l) System variables for workpiece zero point offset values

Variable number	Function
#5201 : #5208	First–axis external workpiece zero point offset value : Eighth–axis external workpiece zero point offset value
#5221 : #5228	First–axis G54 workpiece zero point offset value : Eighth–axis G54 workpiece zero point offset value
#5241 : #5248	First–axis G55 workpiece zero point offset value : Eighth–axis G55 workpiece zero point offset value
#5261 : #5268	First–axis G56 workpiece zero point offset value : Eighth–axis G56 workpiece zero point offset value
#5281 : #5288	First–axis G57 workpiece zero point offset value : Eighth–axis G57 workpiece zero point offset value
#5301 : #5308	First–axis G58 workpiece zero point offset value : Eighth–axis G58 workpiece zero point offset value
#5321 : #5328	First–axis G59 workpiece zero point offset value : Eighth–axis G59 workpiece zero point offset value
#7001 : #7008	First–axis workpiece zero point offset value (G54.1 P1) : Eighth–axis workpiece zero point offset value
#7021 : #7028	First–axis workpiece zero point offset value (G54.1 P2) : Eighth–axis workpiece zero point offset value
:	:
#7941 : #7948	First–axis workpiece zero point offset value (G54.1 P48) : Eighth–axis workpiece zero point offset value
#14001 : #14008	First–axis workpiece zero point offset value (G54.1 P1) : Eighth–axis workpiece zero point offset value
#14021 : #14028	First–axis workpiece zero point offset value (G54.1 P2) : Eighth–axis workpiece zero point offset value
:	:
#19980 : #19988	First–axis workpiece zero point offset value (G54.1 P300) : Eighth–axis workpiece zero point offset value

The following variables can also be used:

Axis	Function	Variable number	
First axis	External workpiece zero point offset	#2500	#5201
	G54 workpiece zero point offset	#2501	#5221
	G55 workpiece zero point offset	#2502	#5241
	G56 workpiece zero point offset	#2503	#5261
	G57 workpiece zero point offset	#2504	#5281
	G58 workpiece zero point offset	#2505	#5301
	G59 workpiece zero point offset	#2506	#5321
Second axis	External workpiece zero point offset	#2600	#5202
	G54 workpiece zero point offset	#2601	#5222
	G55 workpiece zero point offset	#2602	#5242
	G56 workpiece zero point offset	#2603	#5262
	G57 workpiece zero point offset	#2604	#5282
	G58 workpiece zero point offset	#2605	#5302
	G59 workpiece zero point offset	#2606	#5322
Third axis	External workpiece zero point offset	#2700	#5203
	G54 workpiece zero point offset	#2701	#5223
	G55 workpiece zero point offset	#2702	#5243
	G56 workpiece zero point offset	#2703	#5263
	G57 workpiece zero point offset	#2704	#5283
	G58 workpiece zero point offset	#2705	#5303
	G59 workpiece zero point offset	#2706	#5323
Fourth axis	External workpiece zero point offset	#2800	#5204
	G54 workpiece zero point offset	#2801	#5224
	G55 workpiece zero point offset	#2802	#5244
	G56 workpiece zero point offset	#2803	#5264
	G57 workpiece zero point offset	#2804	#5284
	G58 workpiece zero point offset	#2805	#5304
	G59 workpiece zero point offset	#2806	#5324

NOTE

To use variables #2500 to #2806 and #5201 to #5328, optional variables for the workpiece coordinate systems are necessary.
Optional variables for 48 additional workpiece coordinate systems are #7001 to #7948 (G54.1 P1 to G54.1 P48).
Optional variables for 300 additional workpiece coordinate systems are #14001 to #19988 (G54.1 P1 to G54.1 P300). With these variables, #7001 to #7948 can also be used.

- **Cutting condition setting function**

By reading the following custom macro variables, the most recently specified E code including program preprocessing can be read.

Table 13.2 (m) System variables for the cutting condition setting function

Variable number	Function
#6032	Look–ahead piercing E code
#6033	Look–ahead cutting E code

- #6032 and #6033 are variables for read only. If an attempt is made to write data to these variables, alarm 116 is issued.
- When reading these variables with the macro executor, add 100000 to variable numbers #6032 and #6033, and specify #106032 and #106033, respectively.
- If the cutting condition setting function option is not provided, these variables read 0.
- Use these variables in machining programs (such as custom macro calls and execution macro calls).
- E codes written with #6030 and #6031 cannot be read.

- **Gap controlled axis automatic save/restore function**

The Z–axis motion in the gap controlled axis automatic save/restore function can be modified by rewriting the contents of the following custom macro variables:

Table 13.2 (n) System variables for the gap controlled axis automatic save/restore function

Variable number	Function
#6048	X– and Y–axis motion start position (PRM. No.15865)
#6049	Deceleration start position (PRM. No.15866)
#6050	Function end position (PRM. No.15867)

For details of the function, refer to "FS16*i*–LB Connection Manual" (B–63663EN).

- **Function for beam output during approach**

The operation of the function for beam output during approach can be modified by rewriting the contents of the following custom macro variables:

Table 13.2 (o) System variables for the function for beam output during approach

Variable number	Function
#6051	Gap deviation value at which beam output starts (parameter No. 15870)
#6052	Adjustment ratio of gap control axis speed (parameter No. 15871)
#6053	Gap control axis control extension time (parameter No. 15872)

For details of the function, refer to "FS16*i*–LB Connection Manual" (B–63663EN).

NOTE
1 While the function for beam output during Z–axis approach is operating, do not change the content of #6051.
2 Do not assign any negative value to #6052 and #6053.

13.3 ARITHMETIC AND LOGIC OPERATION

The operations listed in Table 13.3(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 13.3 (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 Arcsine Tangent Arctangent	#i=SIN[#j]; #i=ASIN[#j]; #i=COS[#j]; #i=ACOS[#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 Natural logarithm Exponential function	#i=SQRT[#j]; #i=ABS[#j]; #i=ROUND[#j]; #i=FIX[#j]; #i=FUP[#j]; #i=LN[#j]; #i=EXP[#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, ASIN, ACOS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

• ARCSIN #i = ASIN[#j];

- The solution ranges are as indicated below:
  When the NAT bit (bit 0 of parameter 6004) is set to 0: 270° to 90°
  When the NAT bit (bit 0 of parameter 6004) is set to 1: –90° to 90°
- When #j is beyond the range of –1 to 1, P/S alarm No. 111 is issued.
- A constant can be used instead of the #j variable.

• ARCCOS #i = ACOS[#j];

- The solution ranges from 180° to 0°.
- When #j is beyond the range of –1 to 1, P/S alarm No. 111 is issued.
- A constant can be used instead of the #j variable.

- **ARCTAN #i = ATAN[#j]/[#k];**

- Specify the lengths of two sides, separated by a slash (/).
- The solution ranges are as follows:

  When the NAT bit (bit 0 of parameter 6004) is set to 0: 0° to 360°

  [Example] When #1 = ATAN[–1]/[–1]; is specified, #1 is 225.0.

  When the NAT bit (bit 0 of parameter 6004) is set to 1: –180° to 180°

  [Example] When #1 = ATAN[–1]/[–1]; is specified, #1 is –135.0.0.
- A constant can be used instead of the #j variable.

- **Natural logarithm #i = LN[#j];**

- Note that the relative error may become 10^{-8} or greater.
- When the antilogarithm (#j) is zero or smaller, P/S alarm No. 111 is issued.
- A constant can be used instead of the #j variable.

- **Exponential function #i = EXP[#j];**

- Note that the relative error may become 10^{-8} or greater.
- When the result of the operation exceeds 3.65 X 10^{47} (j is about 110), an overflow occurs and P/S alarm No. 111 is issued.
- A constant can be used instead of the #j variable.

- **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 beam 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 beam 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 (See III–9.7).

Example:

ROUND → RO
FIX → FI

- **Priority of operations**

$\boxed{1}$ 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, P/S alarm No. 118 occurs.

Example) $\#1=\underbrace{SIN\,[\underbrace{\underbrace{[\underbrace{[\underbrace{\#2+\#3}_{\boxed{1}}]*\#4}_{\boxed{2}}+\#5}_{\boxed{3}}]*\#6}_{\boxed{4}}]}_{\boxed{5}};$

$\boxed{1}$ 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 13.3 (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 (Note1) $\left\lvert \frac{\varepsilon}{a} \right\rvert$
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}	(Note2) $\text{Min} \left\lvert \frac{\varepsilon}{b} \right\rvert , \left\lvert \frac{\varepsilon}{c} \right\rvert$
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error (Note3) $\lvert \varepsilon \rvert$ degrees
a = ATAN [b] / [c] (Note4)	1.8×10^{-6}	3.6×10^{-6}	

NOTE

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.
5 If the result of the operation by the SIN, COS, or TAN function is less than 1.0×10^{-8} or is not 0 because of the precision of the operation, the result of the operation can be normalized to 0 by setting bit 1 of parameter No. 6004 to 1.

- 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], P/S alarm No. 112 occurs.

13.4 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 bit 5 of parameter SBM No. 6000 is 1.
- Macro blocks are not regarded as blocks that involve no movement in the cutter compensation mode (see II–13.7).

- **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 not include other command addresses except an O,N or L address have the same property as macro statements.
- The blocks not include other command addresses (included M99) except an O,N,P or L address have the same property as macro statements.

13.5 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 ...)

13.5.1 Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999 is specified, P/S alarm No. 128 occurs. A sequence number can also be specified using an expression.

GOTO n ;	n: Sequence number (1 to 99999)

Example:

```
GOTO 1;
GOTO#10;
```

13.5.2 Conditional Branch (IF Statement)

Specify a conditional expression after IF.

IF[<conditional expression>]GOTOn

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 condi-        IF [#1 GT 10] GOTO 2 ;
tion is not sa-        Processing           If the condition is satisfied
tisfied
                N2 G00 G91 X10.0 ;
                          :
```

IF[<conditional expression>]THEN

If the specified conditional expression is satisfied, a predetermined macro statement is executed. Only a single macro statement is executed.

If the values of #1 and #2 are the same, 0 is assigned to #3.

```
IF [#1 EQ #2] THEN#3=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 13.5.2 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
```

13.5.3 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 condi-        If the condi-     Processing
  tion is not sa-      tion is satis-
  tisfied              fied
                                      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, P/S 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), P/S 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 <vacant> and zero have different effects. In other types of conditional expressions, a <vacant> 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;
```

13.6 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.

13.6.1 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 ;
   :
G65 P9010 L2 A1.0 B2.0 ;
   :
M30 ;
```

```
O9010 ;
#3=#1+#2 ;
IF [#3 GT 360] GOTO 9 ;
G00 G91 X#3 ;
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	Address	Variable number	Address	Variable number
A	#1	K_3	#12	J_7	#23
B	#2	I_4	#13	K_7	#24
C	#3	J_4	#14	I_8	#25
I_1	#4	K_4	#15	J_8	#26
J_1	#5	I_5	#16	K_8	#27
K_1	#6	J_5	#17	I_9	#28
I_2	#7	K_5	#18	J_9	#29
J_2	#8	I_6	#19	K_9	#30
K_2	#9	J_6	#20	I_{10}	#31
I_3	#10	K_6	#21	J_{10}	#32
J_3	#11	I_7	#22	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.

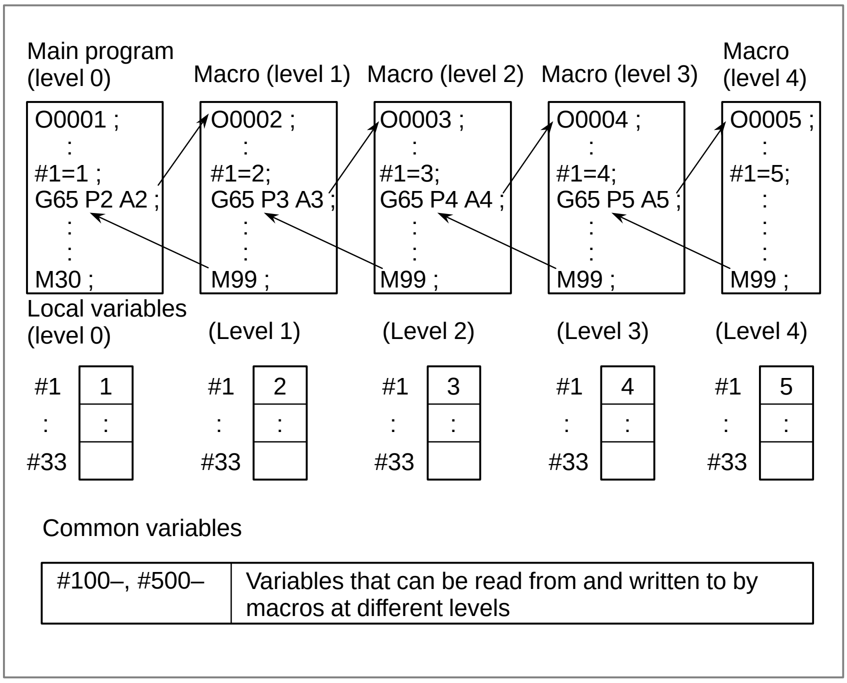
• Call nesting

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

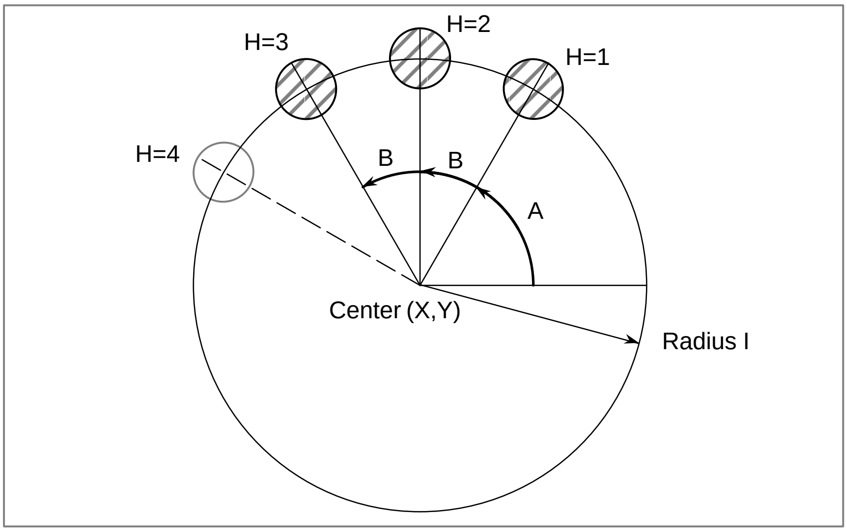
- Local variables from level 0 to 4 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G65 or G66), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the CNC.

- 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 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 radius (#26)
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 Z0;
G65 P9100 X100.0 Y50.0 Z–10.0 F500 I100.0 A0 B45.0 H5;
M30;
```

- **Macro program (called program)**

```
O9100;
  #3=#4003; .......... Stores G code of group 3.
  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 the X coordinate of the hole center.
  #6=#25+#4*SIN[#1]; .. Calculates the Y coordinate of the hole center.
  #5=#5+#26; ......... X coordinate of the drilling start position
  G90 G00 X#5 Y#6; .... Moves to the drilling start position.
  G02 I–#26 F#9; ...... Drilling
  #1=#1+#2; .......... Updates the angle.
  #11=#11–1; .......... Decrements the number of holes.
  END 1;
  G#3; ............... 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 drilling start position for the hole to be drilled
#6: Y coordinate of the drilling start position for the hole to be drilled

13.6.2 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**

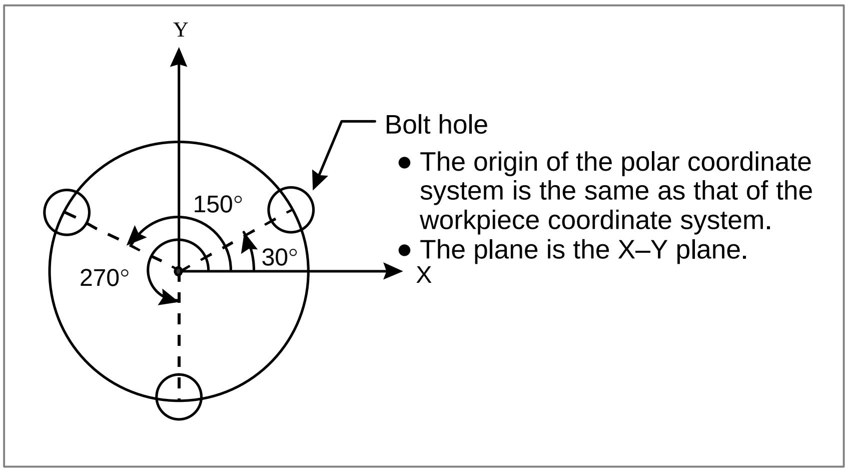
Modal calls can be nested by specifying another G66 code during a modal call.

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

Bolt hole circles are machined. Each time positioning is performed, a bolt hole is machined at that location.



- **Calling format**

```
G65 P9110 Rr;
```

R: Radius of a bolt hole circle . . . . (#18)

- **Program that calls a macro program**

O0001;
G17 G90 G16; . . Polar coordinate command, X–Y plane selection
The origin of the polar coordinate system is the same as that of the workpiece coordinate system.
G66 P9110 R10.0 F2000; . . . Macro modal call specification
G00 X100.0 Y30.0; . . . . Radius: 100 mm, angle: 30 deg.
G00 X150.0; . . . . Radius: 100 mm, angle: 150 deg.
G00 Y270.0; . . . . Radius: 100 mm, angle: 270 deg.
G15; . . . . Polar coordinate command cancel
M30;

- **Macro program (program called)**

O9110;
G91 G15;
G24; . . . . Piercing
G01 Y–#18; . . . . Bolt hole machining
G03 J#18; . . . . Bolt hole machining
G90 G16;
M99;

13.6.3 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 ;                          O9010 ;
  :                                :
G81 X10.0 Y20.0 Z–10.0 ;           :
  :                                :
M30 ;                            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 (N0.6050 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.

13.6.4 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 (9020 to 9029) 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.

Sample program

The time each assist gas is used is measured by using the macro call function with an M code.

- Measure the total time that assist gas types 1 to 3 are used.
- Determine the variables for storing the assist gas type numbers and the measured times, as follows:

#501	Total time assist gas type 1 is used
#502	Total time assist gas type 2 is used
#503	Total time assist gas type 3 is used

- Because system variable #3002 is used as the timer for time measurement, the time during which the cycle start lamp is lit is measured. However, the stop time due to single–block operation and feed hold is excluded.

- **Parameter specification**

Set 15 for parameter No. 6080.

- **Variable value setting**

Set ”0” for variables #501 to #503.

- **Calling format**

```
M15 Aa Bb;
```

A: Assist gas type . . . . (#1)
B: Flow pattern . . . . (#2)

- **Program that calls a macro program**

O0001;
M15 A1 B1; . . Selects assist gas type 1 and flow pattern 1.
. . . .
M15 A2 B1; . . Selects assist gas type 2 and flow pattern 1.
. . . .
M15 A3 B1; . . Selects assist gas type 3 and flow pattern 1.
. . . .
M15 A0; . . . . Deselects the assist gas.
. . . .
M30;

- **Macro program (program called)**

O9020;
IF [#100 EQ 0] GOTO 10;
IF [#100 GT 3] GOTO 10;
 # [500 + #100] = #3002 + # [500 + #100];
. . . . Updates the time that the assist gas is used.
N10 #100 = #1;
#3002 = 0;
IF [#1 EQ 0] GOTO 15; . . Checks the selected assist gas type.
IF [#1 EQ 3] GOTO 15;
IF [#2 EQ 0] GOTO 15; . . Checks the selected flow pattern.
IF [#2 EQ 3] GOTO 15;
G32 P#1 Q#2; . . . . Selects the assist gas.
M99;
M15 G32 P0; aa . . . . Stops the assist gas.
M99;

13.6.5 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 toNo. 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.

- **M code**

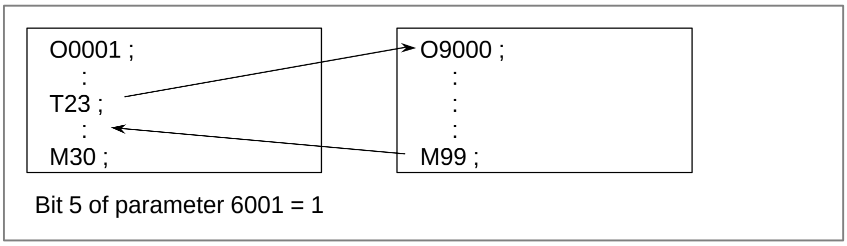
An M code in a macro program that has been called is treated as an ordinary M code.

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.

13.6.6 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 bit 5 of parameter TCS No.6001 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

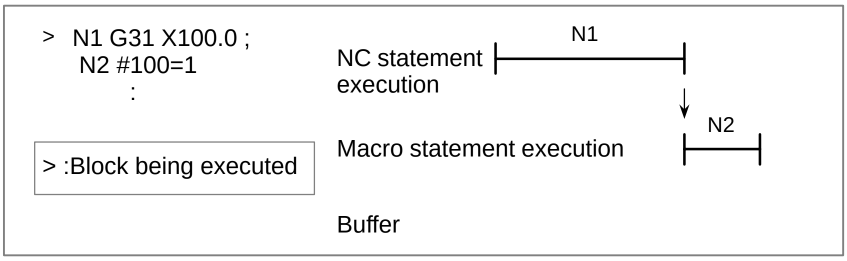
In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using a T code. A T code in such a macro or program is treated as an ordinary T code.

13.7 PROCESSING MACRO STATEMENTS

For smooth machining, the CNC prereads the NC statement to be performed next. This operation is referred to as buffering. During AI contour control mode, the CNC prereads not only the next block but also the multiple blocks. And in the cutter compensation mode (G41, G42), the CNC prereads the NC statements two or three blocks ahead to find intersections even if the CNC is not in AI contour control mode. Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Therefore, the timing of the macro statement execution is not always the specified order.
At the blocks containing M00, M01, M02 or M30, blocks containing M–codes for which buffering is suppressed by setting parameter (No.3411–3432), and blocks containing prevention buffering G codes such as G53, the CNC stops to preread the NC statement after that. Then, the stop of the macro statement execution is guaranteed until such M codes or G codes complete its execution.

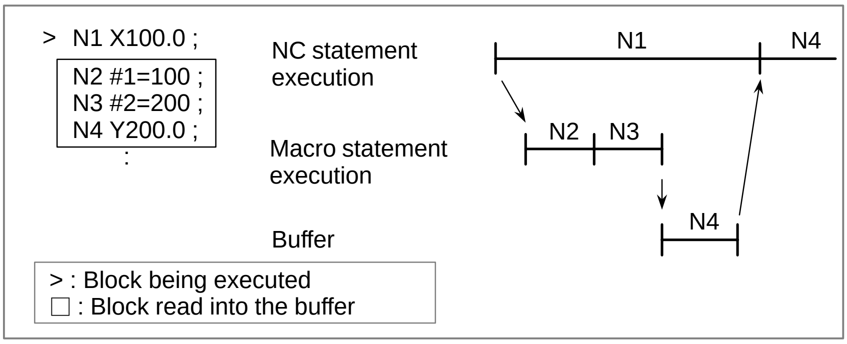
13.7.1 Details of NC Statements and Macro Statements Execution

- **When the next block is not buffered (M codes that are not buffered, G31, etc.)**



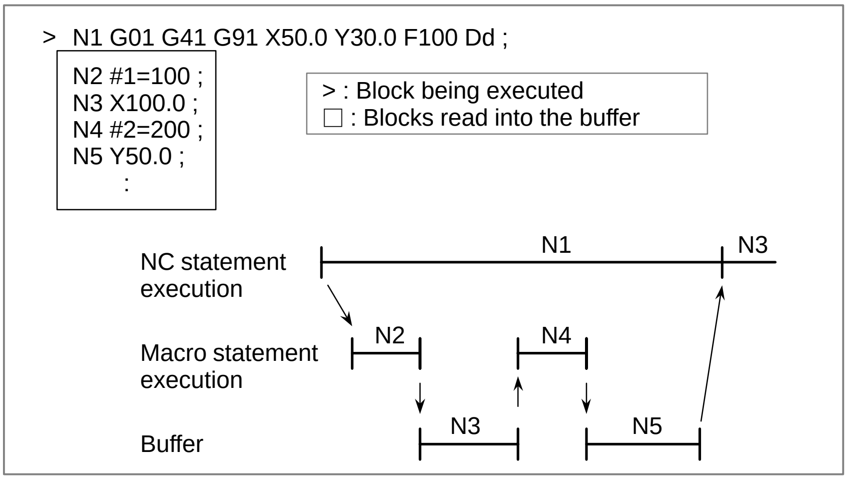
NOTE
In case that you need to execute the macro statement after completing the block just before the macro statement, specify M code or G code that are not buffered just before the macro statement. Specially, in case of reading / writing the system variables to control signals, coordinates, offset value, etc., it may different system variable data by the timing of the NC statement execution. To avoid this phenomenon, specify such M codes or G codes before the macro statement, if necessary.

- **Buffering the next block in other than cutter compensation mode (G41, G42) (normally prereading one block)**



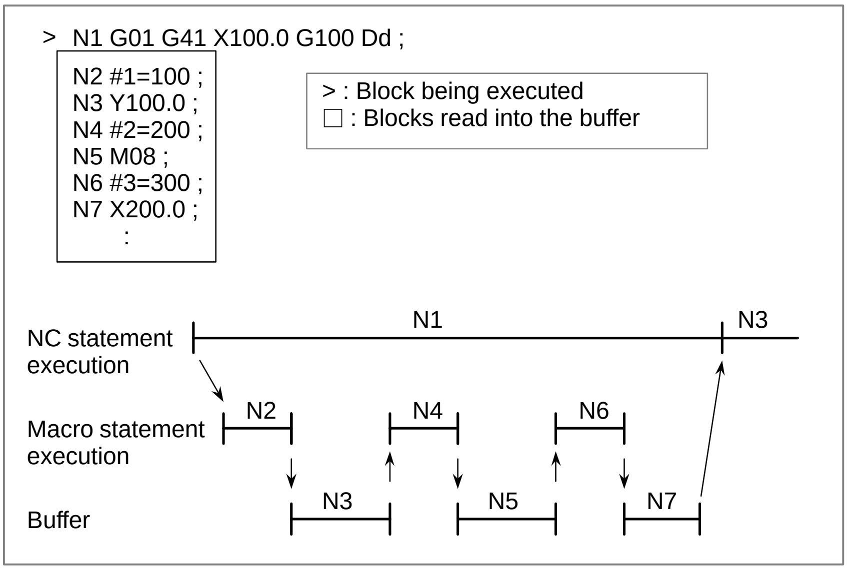
When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- **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.

13.7.2 Caution for Using System Variables

In case of using the following system variables in macro program and you need to execute the macro program after completing the block execution just before the macro program, M code which can prevent buffering (parameter No.3411–3432) or G53 command block is necessary just before the macro program.

Meaning	Read Write	Number of Variable	Note (In case not to command M code preventing buffering or G53 block.)
Interface signals	Read	#1000 – #1015 , #1032	The data is read at buffering a macro program.
	Write	#1100 – #1115 , #1132	The data is written at buffering a macro program.
Tool compensation values	Write	#10001 –	The data is written at buffering a macro program.
Macro alarms	Write	#3000	Macro alarm is generated at maximum 2 blocks before a macro program.
Program stop with message	Write	#3006	Program stops at maximum 2 blocks before a macro program.

Meaning	Read Write	Number of Variable	Note (In case not to command M code preventing buffering or G53 block.)
Time information	Read Write	#3001,#3002	The data is read / written at buffering a macro program.
	Read	#3011,#3012	The data is read at buffering a macro program.
Automatic operation control	Write	#3003, #3004	Setting data is available at maximum 2 blocks before a macro program.
Settings	Write	#3005	The data is written at buffering a macro program.
Mirror image	Read	#3007	The data is read at buffering a macro program.
Currently selected additional workpiece coordinate system	Read	#4130(P) #4014 (G54 – G59)	The data is read at maximum 3 blocks before a macro program.
Current position (Machine coordinate system)	Read	#5021 – #5028	The uncertain position in moving is read.
Current position (Workpiece coordinate system)	Read	#5041 – #5048	The uncertain position in moving is read.
Tool length offset value	Read	#5081 – #5088	The offset value of current execution block is read.
Deviated servo position	Read	#5101 – #5108	The uncertain deviation in moving is read.
Workpiece zero point offset value	Write	#5201 – #5328 #7001 – #7948 #14001 – #19988	The data is written at buffering a macro program.

Example)

```
O0001              O2000
N1 X10.Y10.;       (Mxx ;) Specify preventing buffering M code or G53
N2 M98P2000;       N100 #1=#5041;(Reading X axis current position)
N3 Y200.0;         N101 #2=#5042;(Reading Y axis current position)
   :                     :
                   M99;
```

In above case, the buffering of N2 block is done and the macro program of O2000 is read and executed during N1 block of main program O1000 execution. Therefore, the current position readings are executed during axes movements at N1 block. So the unexpected position data can be read to #1 and #2 because of axes movements. In this case, please specify M code preventing buffering Mxx ; (or G53 ;) just before N100 block of O2000. By this, the position data at completing the execution of N1 block can be read to #1 and #2 because O2000 is executed after completing the execution of N1 block of O0001.

13.8 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.

13.9 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.
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 macro programs and subprograms from being destroyed by accident. When the entire memory is cleared (by pressing the RESET and DELETE keys at the same time to turn on the power), the contents of memory such as custom macro programs are deleted.

- **Reset**

With a reset operation, local variables and common variables #100 to #149 are cleared to null values. They can be prevented from clearing 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**

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, P/S alarm No. 003 occurs.

13.10 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 the 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.

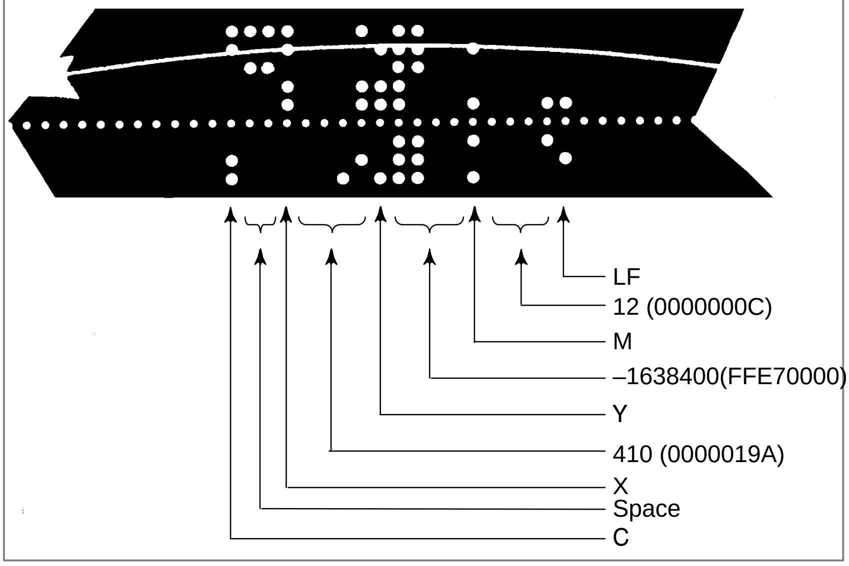
(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2–word (32–bit) data, including the decimal digits. It is output as binary data starting from the highest byte.

(iii) When specified data has been output, an EOB code is output according to the setting code (ISO).

(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] …]

- a: Character
- #b: Variable
- c: Number of significant digits in the integer part
- d: Number of significant decimal places

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 parameter 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 parameter PRT is 1, no code is output.

Example)

DPRNT [X#2 [53] Y#5 [53] T#30 [20]]

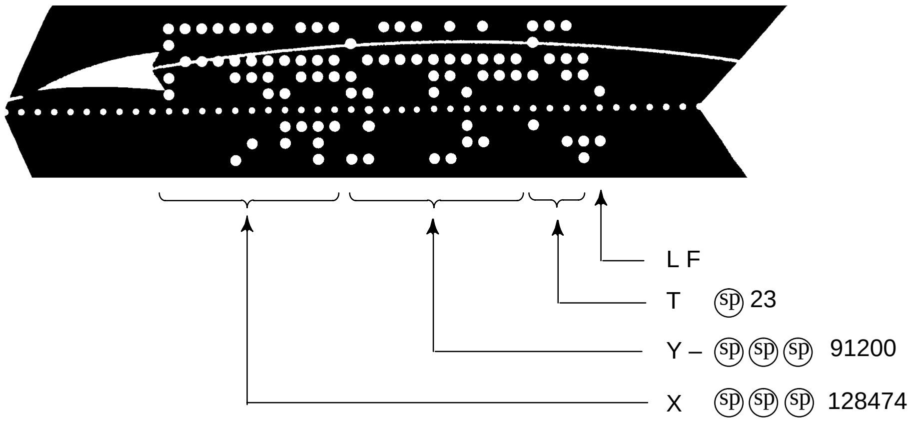
Variable value

#2=128.47398

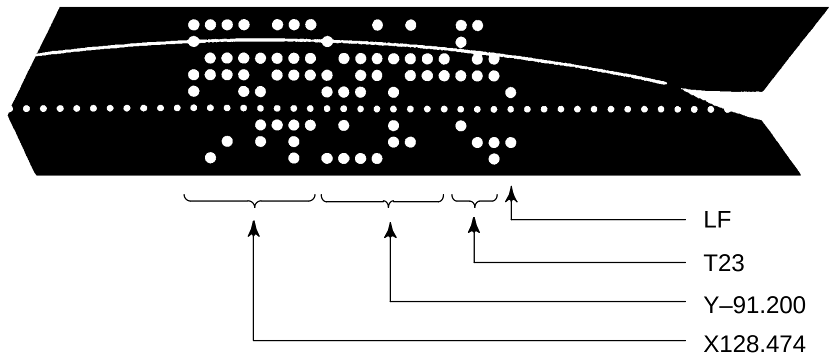
#5=–91.2

#30=123.456

(1) Parameter PRT(No.6001#1)=0



(2) Parameter PRT(No.6001#1)=0



- **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 setting 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, No.102 and No.103)
I/O channel 1 : Parameters (No.111, No.112 and No.113)
I/O channel 2 : Parameters (No.112, No.122 and No.123)

Never specify the output device FANUC Cassette or Floppy for punching.

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 of bit 4 of parameter 6001 is 1).

NOTE

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.

13.11 INTERRUPTION TYPE CUSTOM MACRO

When a program is being executed, another program can be called by inputting an interrupt signal (UINT) from the machine. This function is referred to as an interruption type custom macro function. Program an interrupt command in the following format:

Format

M96 P○○○○ ;	Enables custom macro interrupt
M97 ;	Disables custom macro interrupt

Explanations

Use of the interruption type custom macro function allows the user to call a program during execution of an arbitrary block of another program. This allows programs to be operated to match situations which vary from time to time.

(1) When a tool abnormality is detected, processing to handle the abnormality is started by an external signal.

(2) A sequence of machining operations is interrupted by another machining operation without the cancellation of the current operation.

(3) At regular intervals, information on current machining is read.
Listed above are examples like adaptive control applications of the interruption type custom macro function.

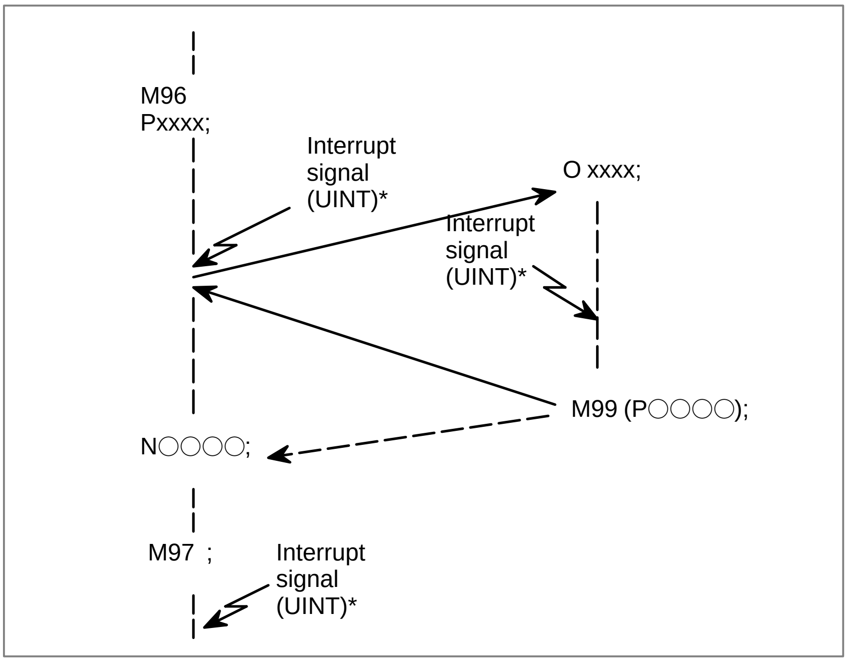


Fig.13.11 Interruption type sustom macro function

When M96Pxxxx is specified in a program, subsequent program operation can be interrupted by an interrupt signal (UINT) input to execute the program specified by Pxxxx.
When the interrupt signal (UINT, marked by * in Fig.13.11 is input during execution of the interrupt program or after M97 is specified, it is ignored.

13.11.1 Specification Method

Explanations

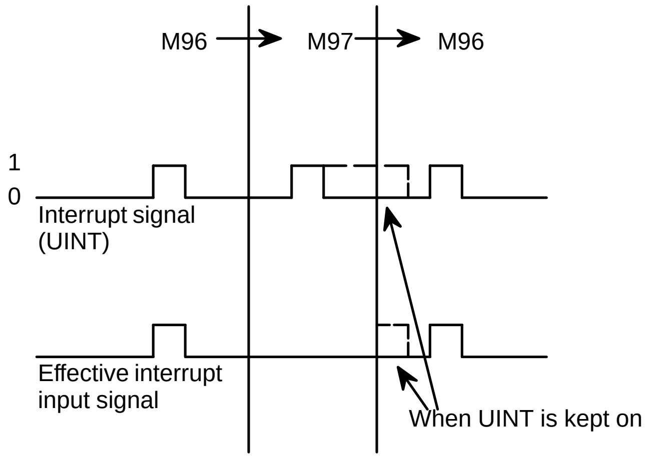
- **Interrupt conditions**

A custom macro interrupt is available only during program execution. It is enabled under the following conditions

– **When memory operation or MDI operation is selected**
– **When STL (start lamp) is on**
– **When a custom macro interrupt is not currently being processed**

- **Specification**

Generally, the custom macro interrupt function is used by specifying M96 to enable the interrupt signal (UINT) and M97 to disable the signal.
Once M96 is specified, a custom macro interrupt can be initiated by the input of the interrupt signal (UINT) until M97 is specified or the NC is reset. After M97 is specified or the NC is reset, no custom macro interrupts are initiated even when the interrupt signal (UINT) is input. The interrupt signal (UINT) is ignored until another M96 command is specified.



The interrupt signal (UINT) becomes valid after M96 is specified. Even when the signal is input in M97 mode, it is ignored. When the signal input in M97 mode is kept on until M96 is specified, a custom macro interrupt is initiated as soon as M96 is specified (only when the status–triggered scheme is employed); when the edge–triggered scheme is employed, the custom macro interrupt is not initiated even when M96 is specified.

NOTE
For the status–triggered and edge–triggered schemes, see Item "Custom macro interrupt signal (UINT)" of II– 13.11.2.

13.11.2 Details of Functions

Explanations

- **Subprogram–type interrupt and macro–type interrupt**

There are two types of custom macro interrupts: Subprogram–type interrupts and macro–type interrupts. The interrupt type used is selected by MSB (bit 5 of parameter 6003).

(a) Subprogram–type interrupt
An interrupt program is called as a subprogram. This means that the levels of local variables remain unchanged before and after the interrupt. This interrupt is not included in the nesting level of subprogram calls.

(b) Macro–type interrupt
An interrupt program is called as a custom macro. This means that the levels of local variables change before and after the interrupt. The interrupt is not included in the nesting level of custom macro calls. When a subprogram call or a custom macro call is performed within the interrupt program, this call is included in the nesting level of subprogram calls or custom macro calls. Arguments cannot be passed from the current program even when the custom macro interrupt is a macro–type interrupt.

- **M codes for custom macro interrupt control**

In general, custom macro interrupts are controlled by M96 and M97. However, these M codes, may already being used for other purposes (such as an M function or macro M code call) by some machine tool builders. For this reason, MPR (bit 4 of parameter 6003) is provided to set M codes for custom macro interrupt control.
When specifying this parameter to use the custom macro interrupt control M codes set by parameters, set parameters 6033 and 6034 as follows:
Set the M code to enable custom macro interrupts in parameter 6033, and set the M code to disable custom macro interrupts in parameter 6034.
When specifying that parameter–set M codes are not used, M96 and M97 are used as the custom macro control M codes regardless of the settings of parameters 6033 and 6034.
The M codes used for custom macro interrupt control are processed internally (they are not output to external units). However, in terms of program compatibility, it is undesirable to use M codes other than M96 and M97 to control custom macro interrupts.

- **Custom macro interrupts and NC statements**

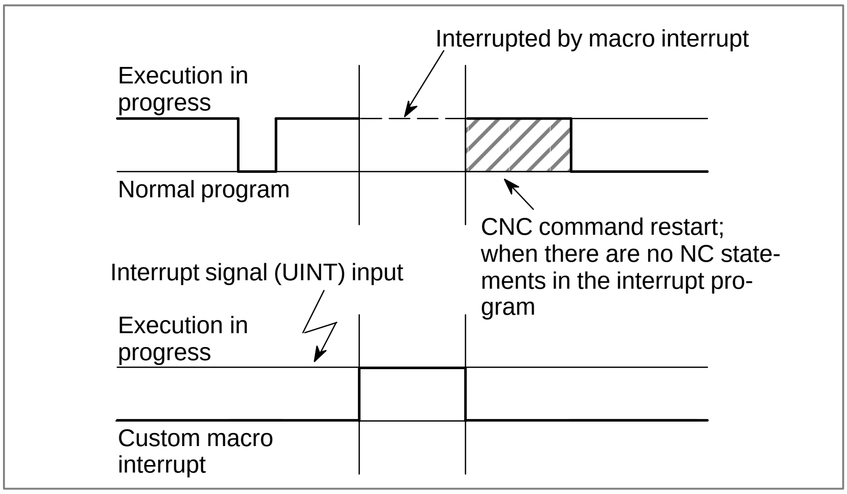
When performing a custom macro interrupt, the user may want to interrupt the NC statement being executed, or the user may not want to perform the interrupt until the execution of the current block is completed. MIN (bit 2 of parameter 6003)is used to select whether to perform interrupts even in the middle of a block or to wait until the end of the block.

Type I (when an interrupt is performed even in the middle of a block)

(i) When the interrupt signal (UINT) is input, any movement or dwell being performed is stopped immediately and the interrupt program is executed.

(ii) If there are NC statements in the interrupt program, the command in the interrupted block is lost and the NC statement in the interrupt program is executed. When control is returned to the interrupted program, the program is restarted from the next block after the interrupted block.

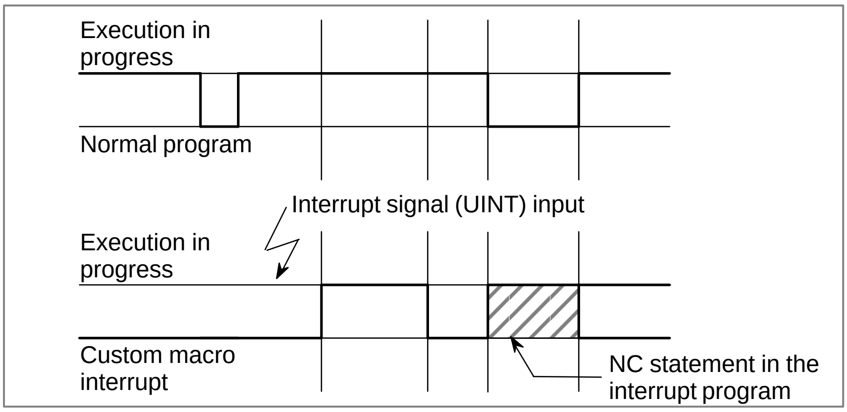
(iii) If there are no NC statements in the interrupt program, control is returned to the interrupted program by M99, then the program is restarted from the command in the interrupted block.



Type II (when an interrupt is performed at the end of the block)

(i) If the block being executed is not a block that consists of several cycle operations such as automatic reference position return (G28), an interrupt is performed as follows:
When an interrupt signal (UINT) is input, macro statements in the interrupt program are executed immediately unless an NC statement is encountered in the interrupt program. NC statements are not executed until the current block is completed.

(ii) If the block being executed consists of several cycle operations, an interrupt is performed as follows:
When the last movement in the cycle operations is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after all cycle operations are completed.



- **Conditions for enabling and disabling the custom macro interrupt signal**

The interrupt signal becomes valid after execution starts of a block that contains M96 for enabling custom macro interrupts. The signal becomes invalid when execution starts of a block that contains M97.
While an interrupt program is being executed, the interrupt signal becomes invalid. The signal become valid when the execution of the block that immediately follows the interrupted block in the main program is started after control returns from the interrupt program. In type I, if the interrupt program consists of only macro statements, the interrupt signal becomes valid when execution of the interrupted block is started after control returns from the interrupt program.

- **Custom macro interrupt during execution of a block that involves cycle operation**

For type I

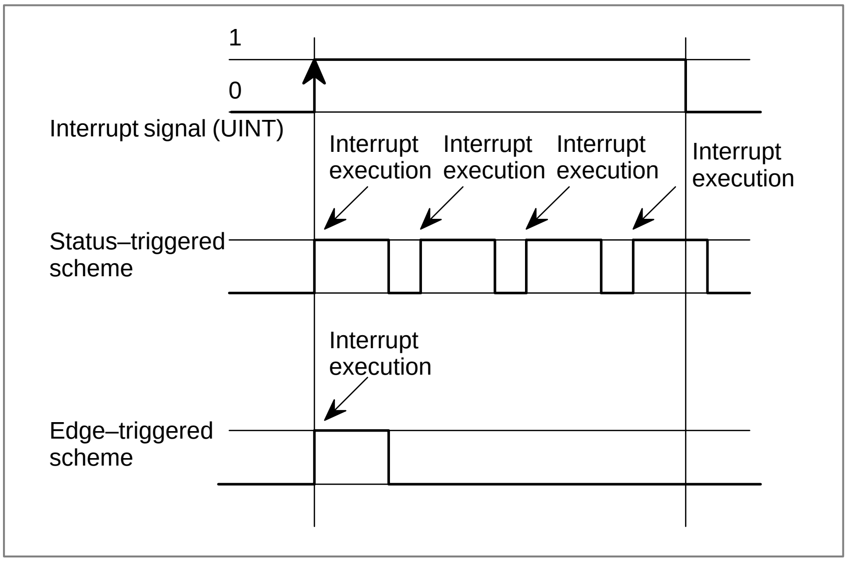
Even when cycle operation is in progress, movement is interrupted, and the interrupt program is executed. If the interrupt program contains no NC statements, the cycle operation is restarted after control is returned to the interrupted program. If there are NC statements, the remaining operations in the interrupted cycle are discarded, and the next block is executed.

For type II

When the last movement of the cycle operation is started, macro statements in the interrupt program are executed unless an NC statement is encountered. NC statements are executed after cycle operation is completed.

- **Custom macro interrupt signal (UINT)**

There are two schemes for custom macro interrupt signal (UINT) input: The status–triggered scheme and edge– triggered scheme. When the status–triggered scheme is used, the signal is valid when it is on. When the edge triggered scheme is used, the signal becomes valid on the rising edge when it switches from off to on status.
One of the two schemes is selected with TSE (bit 3 of parameter 6003). When the status–triggered scheme is selected by this parameter, a custom macro interrupt is generated if the interrupt signal (UINT) is on at the time the signal becomes valid. By keeping the interrupt signal (UINT) on, the interrupt program can be executed repeatedly.
When the edge–triggered scheme is selected, the interrupt signal (UINT) becomes valid only on its rising edge. Therefore, the interrupt program is executed only momentarily (in cases when the program consists of only macro statements). When the status–triggered scheme is inappropriate, or when a custom macro interrupt is to be performed just once for the entire program (in this case, the interrupt signal may be kept on), the edge–triggered scheme is useful.
Except for the specific applications mentioned above, use of either scheme results in the same effects. The time from signal input until a custom macro interrupt is executed does not vary between the two schemes.



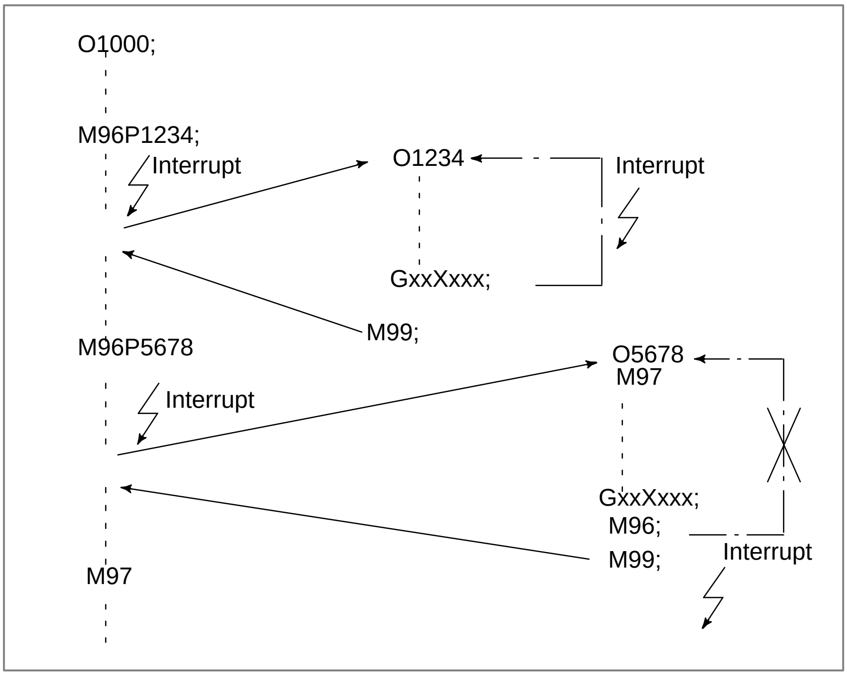
In the above example, an interrupt is executed four times when the status triggered scheme is used; when the edge– triggered scheme is used, the interrupt is executed just once.

- **Return from a custom macro interrupt**

To return control from a custom macro interrupt to the interrupted program, specify M99. A sequence number in the interrupted program can also be specified using address P. If this is specified, the program is searched from the beginning for the specified sequence number. Control is returned to the first sequence number found.

When a custom macro interrupt program is being executed, no interrupts are generated. To enable another interrupt, execute M99. When M99 is specified alone, it is executed before the preceding commands terminate. Therefore, a custom macro interrupt is enabled for the last command of the interrupt program. If this is inconvenient, custom macro interrupts should be controlled by specifying M96 and M97 in the program.

When a custom macro interrupt is being executed, no other custom macro interrupts are generated; when an interrupt is generated, additional interrupts are inhibited automatically. Executing M99 makes it possible for another custom macro interrupt to occur. M99 specified alone in a block is executed before the previous block terminates. In the following example, an interrupt is enabled for the Gxx block of O1234. When the signal is input, O1234 is executed again. O5678 is controlled by M96 and M97. In this case, an interrupt is not enabled for O5678 (enabled after control is returned to O1000).



NOTE

When an M99 block consists only of address O, N, P, L, or M, this block is regarded as belonging to the previous block in the program. Therefore, a single–block stop does not occur for this block. In terms of programming, the following 1 and 2 are basically the same. (The difference is whether G○○ is executed before M99 is recognized.)

1 G○○ X○○○ ;
M99 ;

2 G○○ X○○○ M99 ;

• Custom macro interrupt and modal information

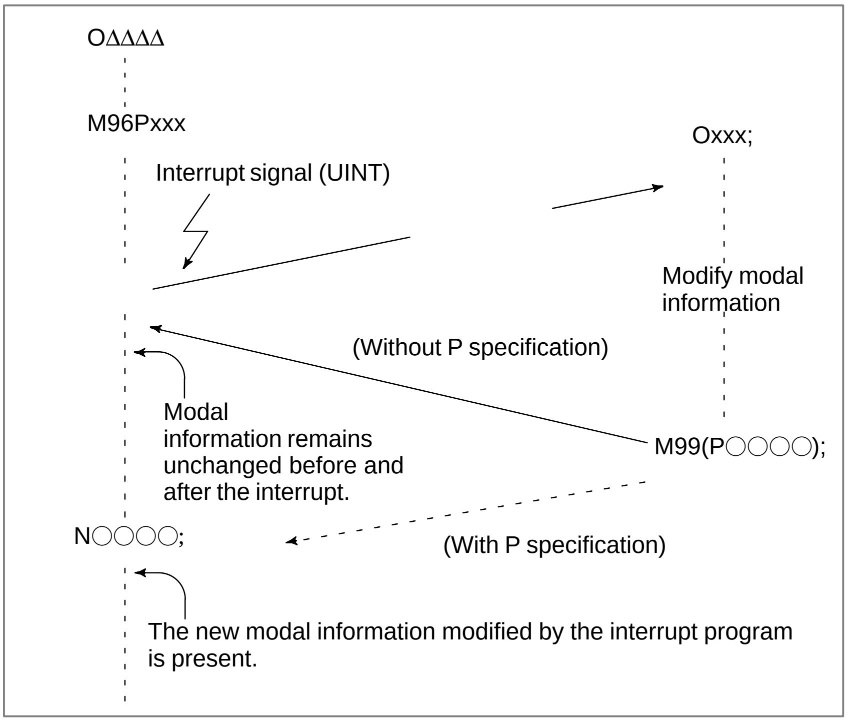
A custom macro interrupt is different from a normal program call. It is initiated by an interrupt signal (UINT) during program execution. In general, any modifications of modal information made by the interrupt program should not affect the interrupted program.

For this reason, even when modal information is modified by the interrupt program, the modal information before the interrupt is restored when control is returned to the interrupted program by M99.

When control is returned from the interrupt program to the interrupted program by M99 Pxxxx, modal information can again be controlled by the program. In this case, the new continuous information modified by the interrupt program is passed to the interrupted program. Restoration of the old modal information present before the interrupt is not desirable. This is because after control is returned, some programs may operate differently depending on the modal information present before the interrupt. In this case, the following measures are applicable:

(1) The interrupt program provides modal information to be used after control is returned to the interrupted program.

(2) After control is returned to the interrupted program, modal information is specified again as necessary.



Modal information when control is returned by M99

The modal information present before the interrupt becomes valid. The new modal information modified by the interrupt program is made invalid.

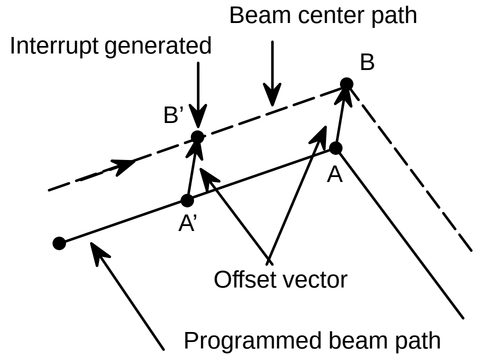
Modal information when control is returned by M99 P○○○○

The new modal information modified by the interrupt program remains valid even after control is returned. The old modal information which was valid in the interrupted block can be read using custom macro system variables #4001 to #4120.

Note that when modal information is modified by the interrupt program, system variables #4001 to #4120 are not changed.

- **System variables (position information values) for the interrupt program**

- The coordinates of point A can be read using system variables #5001 and up until the first NC statement is encountered.
- The coordinates of point A' can be read after an NC statement with no move specifications appears.
- The machine coordinates and workpiece coordinates of point B' can be read using system variables #5021 and up and #5041 and up.



- **Custom macro interrupt and custom macro modal call**

When the interrupt signal (UINT) is input and an interrupt program is called, the custom macro modal call is canceled (G67). However, when G66 is specified in the interrupt program, the custom macro modal call becomes valid. When control is returned from the interrupt program by M99, the modal call is restored to the state it was in before the interrupt was generated. When control is returned by M99Pxxxx;, the modal call in the interrupt program remains valid.

- **Custom macro interrupt and program restart**

When the interrupt signal (UINT) is input while a return operation is being performed in the dry run mode after the search operation for program restart, the interrupt program is called after restart operation terminates for all axes. This means that interrupt type II is used regardless of the parameter setting.

- **DNC operation and interruption type custom macro**

“Interruption type custom macro” cannot be done during DNC operation or executing a program with an external input–output device.

14 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. (5–digit) or compensation position No. for pitch errors compensation +10,000 (5–digit) **R_:** Parameter setting value (Leading zeros can be omitted.) **P_:** Axis No. 1 to 8 (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 8 (up to eight 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.

> **WARNING**
> 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.

> **NOTE**
> Other NC statements cannot be specified while in parameter input mode.

Examples

1. Set bit 2 (SBP) 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 Z–axis (3rd axis) and A–axis (4th 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
N1322P3R4500 ;	Modify Z axis
N1322P4R12000 ;	Modify A axis
G11 ;	cancel parameter entry mode

15 MEMORY OPERATION USING FS15 TAPE FORMAT

General

Memory operation of the program registered by FS15 tape format is possible with setting of the setting parameter (No. 0001#1).

Explanations

Data formats for cutter compensation and subprogram calling are different between the Series 16 and Series 15. The Series 15 data formats can be processed for memory operation.Other data formats must comply with the Series 16.When a value out of the specified range for the Series 16 is registered, an alarm occurs. Functions not available in the Series 16 cannot be registered or used for memory operation.

- **Address for the cutter compensation offset number**

Offset numbers are specified by address D in the Series 15. When an offset number is specified by address D, the modal value specified by address H is replaced with the offset number specified by address D.

- **Subprogram call**

If a subprogram number of more than four digits is specified, the four low–order digits are regarded as the subprogram number. If no repeat count is specified, 1 is assumed.

Table 15 (a) Subprogram call data format

CNC	Data format
Series 15	**M98 P○○○○○ L○○○○ ;** P : Subprogram number L : Repetition count (1 to 9999)
Series 16	**M98 P○○○○ □□□□ ;** Repetition count (1 to 999) Subprogram number

- **Address for the canned cycle repetition count**

The Series 15 and Series 16 use different addresses for the repeat count for canned cycles as listed in Table 15 (a).

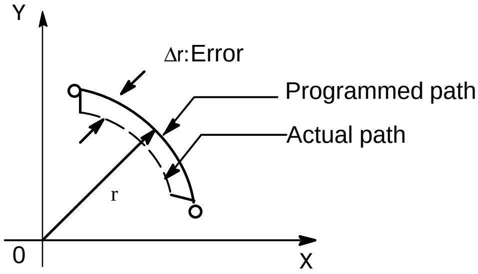
Table 15 (b) Address for times of repetition of canned cycle

CNC	Address
Series 15	L
Series 16	K

16 HIGH SPEED CUTTING FUNCTIONS

16.1 FEEDRATE CLAMPING BY ARC RADIUS

When an arc is cut at a high speed in circular interpolation, a radial error exists between the actual beam path and the programmed arc. An approximation of this error can be obtained from the following expression:



$$\Delta r = \frac{1}{2}\left(T_1^2+T_2^2\right)\frac{v^2}{r}$$

Δr : Maximum radial error (mm)
v : Feedrate (mm/s)
r : Arc radius (mm)
T_1 : Time constant (s) for exponential acceleration/deceleration of cutting feed
T_2 : Time constant of the servo motor (s)

When actual machining is performed, radius r of the arc to be machined and permissible error Dr are given. Then, maximum allowable feedrate v (mm/min) is determined from the above expression.

The function for clamping the feedrate by the arc radius automatically clamps the feedrate of arc cutting to the value set in a parameter. This function is effective when the specified feedrate may cause the radial error for an arc with a programmed radius to exceed the permissible degree of error.

For details, refer to the relevant manual published by the machine beam builder.

16.2 ADVANCED PREVIEW CONTROL (G08)

This function is designed for high–speed precise machining. With this function, the delay due to acceleration/deceleration and the delay in the servo system which increase as the feedrate becomes higher can be suppressed.
The beam can then follow specified values accurately and errors in the machining profile can be reduced.
This function becomes effective when advanced preview control mode is entered.
For details, refer to the relevant manual published by the machine beam builder.

Format

G08 P_
P1 : Turn on advanced preview control mode.
P0 : Turn off advanced preview control mode.

Explanations

- **Available functions**

In look–ahead control mode, the following functions are available:
(1) Linear acceleration/deceleration before interpolation
(2) Automatic corner deceleration function
For details on the above functions, see the connection manual (function) (B–63523EN–1). Each function, specific parameters are provided.

- **Reset**

Advanced preview control mode is canceled by reset.

Limitations

- **G08 command**

Specify G08 code only in a block.

- **Functions that can be specified**

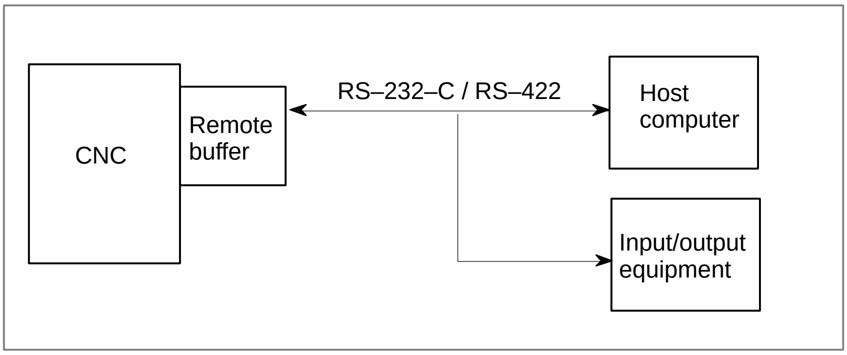
In the advanced preview control mode, the optional functions listed below can be specified.

NOTE
To use a function other than the following optional functions, turn the advanced preview control mode off, specify the desired function, then turn the mode on again.

· High–precision contour control
· Axis control by the PMC
(Bits 4 (G8R) and 3 (G8C) of parameter No. 8004 can be set to also use this function in the look–ahead control mode.)
· Single direction positioning
· Polar coordinate command
· Helical interpolation
· Program restart
· External deceleration
· Simple synchronous control
· Sequence number comparison and stop
· Position switch
(Bit 3 (PSF) of parameter No. 6901 can be set to also use this function in the advanced preview control mode. When this parameter is set to 1, the time of signal output varies. This may require ladder program modification
and other action.)
· Custom macro B
· Optional–angle chamfering/corner rounding
· Inch/metric conversion
· Programmable mirror image
· Automatic corner override
(Only the internal circular cutting feedrate change is valid.)
· Scaling
· Coordinate system rotation
· Workpiece coordinate system
· Figure copy
· Workpiece coordinate system preset
· Cutter compensation C
· Corner circular interpolation
· Tool offset
· Graphic display
· Dynamic graphic display

16.3 HIGH–SPEED REMOTE BUFFER

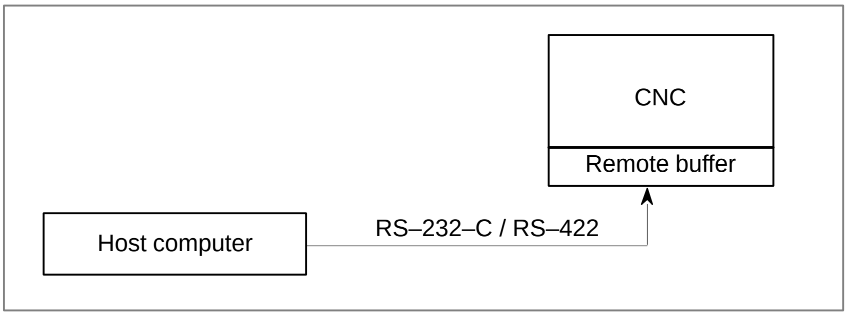
A remote buffer can continuously supply a large amount of data to the CNC at high speeds when connected to the host computer or input/output equipment via a serial interface.



When the remote buffer is connected online to the host computer, fast and reliable DNC operation is possible.
The remote buffer function includes high–speed remote buffer A and high–speed remote buffer B for high–speed machining. High–speed remote buffer A uses binary data. High–speed remote buffer B uses NC language. For details on remote buffer specifications, refer to the "Remote Buffer Supplement" (B–61802E–1).

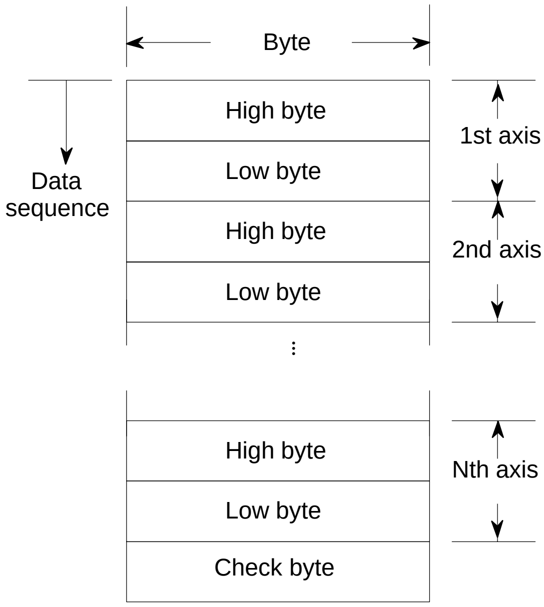
16.3.1 High–Speed Remote Buffer A (G05)

Specify G05 only in a block using normal NC command format. Then specify move data in the special format explained below. When zero is specified as the travel distance along all axes, normal NC command format can be used again for subsequent command specification.



Format

VBinary input operation enabled : G05;
VBinary input operation disabled : The travel distance along all axes are set to zero.
VData format for binary input operation



In the data format for binary input operation, the travel distance along each axis (2 bytes) per unit time is specified. The travel distances along all axes are placed sequentially from the first axis, then a check byte is added. (The data length for one block is [2 $\times$ N + 1] bytes).
All data must be specified in binary.

Explanations

- **Selecting the unit time**

The unit time (in ms) can be selected by setting bits 4, 5, and 6 of parameter IT0,IT1,IT2 No. 7501.

- **Travel distance data**

The following unit is used for specifying the travel distance along each axis. (A negative travel distance is indicated in 2's complement.)

Increment system	IS–B	Unit
Millimeter machine	0.001	mm
Inch machine	0.0001	inch

The data format of the travel distance is as follows. The bits marked * are used to specify a travel distance per unit time.

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
*	*	*	*	*	*	*	0	*	*	*	*	*	*	*	0

Example: When the travel distance is 700 μm per unit time
(millimeter machine with increment system IS–B)

15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
0	0	0	0	1	0	1	0	0	1	1	1	1	0	0	0

- **Check Byte**

All bytes of the block except for the check byte ([2*N] bytes) are summed up, and any bits above 8th bit are discarded.

- **Transfer speed**

The CNC reads (2 x N + 1)–byte data (where N is the number of axes) for every unit time that is set in the parameter. To allow the CNC to continue machining without interruption, the following minimum baud rate is required for data transfer between the host and remote buffer:

$$(2\times N+1) \times \frac{11}{T} \times 1000 \text{ baud (T : Unit time)}$$

- **Cutter compensation**

If G05 is specified in cutter compensation mode, the P/S 178 alarm is issued.

- **Feed hold and interlock**

Feed hold and interlock are effective.

- **Mirror image**

The mirror image function (programmable mirror image and setting mirror image) cannot be turned on or off in the G05 mode.

- **Acceleration / deceleration type**

In binary input operation mode, when beam movement starts and stops in cutting feed mode, exponential acceleration/deceleration is performed (the acceleration/deceleration time constant set in parameter No. 1622 is used).

Limitations

- **Modal command**

In binary input operation mode, only linear interpolation as specified in the defined data format is executed (equivalent to the incremental command for linear interpolation).

- **Invalid functions**

The single block, feedrate override, and maximum cutting feedrate clamp functions have no effect. The program restart, block restart, and high–speed machining functions cannot be used. In addition, miscella–neous functions cannot be executed in binary operation.

- **Memory registration**

No data can be stored in memory.

16.4 DISTRIBUTION PROCESSING TERMINATION MONITORING FUNCTION FOR THE HIGH–SPEED MACHINING COMMAND (G05)

During high–speed machining, the distribution processing status is monitored. When distribution processing terminates, P/S alarm No. 000 and P/S alarm No. 179 are issued upon completion of the high–speed machining command (according to the setting of ITPDL (bit 7 of parameter No. 7501)).

These P/S alarms can be canceled only by turning off the CNC power.

Explanations

- **High–speed machining command**

High–speed machining using the high–speed remote buffer A function.

- **Distribution processing termination**

Failure to perform normal distribution processing because distribution processing required for high–speed machining exceeded the CNC processing capacity, or because distribution data sent from the host was delayed for some reason

Alarm

<table>
<tr><th>Alarm No.</th><th>Message</th><th>Contents</th></tr>
<tr><td>000</td><td>PLEASE TURN OFF POWER</td><td rowspan="2">During high–speed machining, distribution processing was terminated.
Related parameters:
Remote buffer transfer baud rate (parameter No. 133)
Number of controlled axes in high–speed machining (parameter No. 7150)
High–speed axis selection during high–speed machining (bit 0 of parameter No. 7510)</td></tr>
<tr><td>179</td><td>PARAM. (PRM No. 7510) SETTING ERROR</td></tr>
</table>

16.5 HIGH–PRECISION CONTOUR CONTROL (G05)

Some machining errors are due to the CNC. Such errors include machining errors caused by acceleration/deceleration after interpolation. To eliminate these errors, the following functions are performed at high speed by an RISC processor. These functions are called high–precision contour control functions.

(1) Function for multiple–block look–ahead acceleration/deceleration before interpolation. This function eliminates machining errors due to acceleration/deceleration.

(2) Automatic speed control function which enables smooth acceleration/deceleration by considering changes in the figure and speed and allowable acceleration for the machine. This is performed by reading multiple blocks in advance.

For details on high–precision contour control using RISC, refer to the relevant manual published by the machine tool builder.

Format

G05P10000 ;	Start HPCC mode
G05P0 ;	End HPCC mode

Explanations

- **HPCC mode**

The mode used to perform high–precision contour control using RISC is called HPCC mode.
To start the HPCC mode in a certain block, specify G05P10000 before that block. To end the HPCC mode, specify G05P0 at the point at which to end the mode.

- **Data that can be specified**

The following data can be specified in HPCC mode:

G00 : Positioning (Note)
G01 : Linear interpolation
G02 : Circular interpolation, helical interpolation (CW)
G03 : Circular interpolation, helical interpolation (CCW)
G17 : Plane selection (XpYp plane)
where, Xp is the X–axis or a parallel axis;
G18 : Plane selection (ZpXp plane)
where, Yp is the Y–axis or a parallel axis;
G19 : Plane selection (YpZp plane)
where, Zp is the Z–axis or a parallel axis.
G40 : Cutter compensation cancel
G41 : Cutter compensation, left
G42 : Cutter compensation, right
G50 : Scaling cancel
G51 : Scaling start
G68 : Coordinate rotation start
G69 : Coordinate rotation cancel
G90 : Absolute command
G91 : Incremental command
Dxxx : Specifying a D code
Fxxxxx : Specifying an F code
Nxxxxx : Specifying a sequence number
G05P10000 : Setting the HPCC mode
G05P0 : Canceling the HPCC mode
I, J, K, R : I, J, K, and R specified for circular interpolation

S, P, Q, E : Specifying a laser output condition (the feedrate and tool compensation amount cannot be changed)

Data for movement along axis : Data for moving the tool along the axis set in parameter No. 1020 (any axis selected from X, Y, Z, U, V, W, A, B, and C)

( ) : Control–in and control–out commands (comment specification)

/n : Optional block skip command (n is a number.)

Mxxxx : Auxiliary function (Note)

Txxxx : Auxiliary function (Note)

Bxxxx : Auxiliary function (Note)

M98, M198, etc. : Subprogram call

NOTE

1 G00, auxiliary functions, subprogram call (M98, M198), and macro call M and T codes can be specified in the HPCC mode only when bit 1 of parameter MSU No. 8403 is 1. If these codes are specified when MSU is not 1, an alarm is issued.
(Alarm No.5012 for G00 and alarm No.9 for auxiliary functions and subprogram calls)

2 To specify the following functions in HPCC mode, the following parameters are required. Specifying any of the following functions without setting the corresponding parameter causes an alarm.
Helical interpolation : Parameter G02 (No.8485*)
(Alarm to be issued: No.28)
Scaling, coordinate rotation : Parameter G51 (No. 8485)
(Alarm to be issued: No.10)

- **When unspecifiable data is specified**

In the HPCC mode, specifying unspecifiable data causes an alarm. To specify a program containing unspecifiable data, specify G05P0 to exit from the HPCC mode before specifying the program.

< Sample program >

Main program

```
O0001 ;
G05P10000 ;   HPCC–ON
G00X100.Y200. ;
G91G01X100Y200Z300F2000 ;
X200Y300Z400 ;
X300Y400Z500 ;
X400Y500Z600 ;
X300Y400Z500 ;
M98P0002 ; SUB PROGRAM
X10. ;
G05P0 ;    HPCC–OFF
G90G51X0Y0Z0 ;
X500Y400Z300 ;
X600Y500Z400 ;
G50 ;
G05 P10000 ;  HPCC–ON
X100Y200 ;
X200Y400 ;
G05P0 ;    HPCC–OFF
G04X3. ;
M30 ;
```

Subprogram

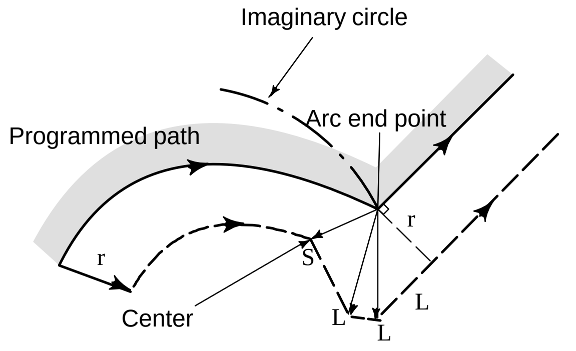
```
O0002 ;
G00X50.Y50. ;
M11 ;
G02I20.F3000 ;
G01X100. ;
G03I80. ;
G01X–50. ;
G02I100.F5000 ;
      :
G01X200.Y300.F1500 ;
X50.Y100.Z150. ;
T24 ;
M99 ;
```

Note) When bit 1 of parameter MSU No. 8403 is 1

- **Cutter compensation C**

When the cutter compensation C option is provided, cutter compensation C is enabled even in HPCC mode. Operation in the offset mode is the same as when HPCC mode is not set, except in the following cases:

- When the end point for an arc does not lie on the arc
  In the HPCC mode, when the end point for an arc does not lie on the arc, the start point and end point are connected with a smooth curve; no arc leading line is created. In this case, the system assumes an imaginary circle to perform cutter compensation C. The center of the imaginary circle is the same as the center of the arc, but the imaginary circle passes through the end point. Under the assumption that cutter compensation has been performed with respect to the imaginary circle, the system creates a vector and performs compensation.



- When the offset mode is canceled temporarily
  In the HPCC mode, automatic reference position return (G28) and automatic return from the reference position (G29) cannot be specified. Therefore, commands that must cancel the offset mode temporarily cannot be specified.

When using cutter compensation C in the HPCC mode, note the following points:

(1) When G05 P10000 and G05 P0, and G41/G42 and G40 are to be specified together, G41/G42 to G40 must be nested between G05 P10000 and G05 P0. This means that HPCC mode cannot be started or canceled in cutter compensation (G41/G42) mode. If such a specification is made, the P/S alarm No.0178 or P/S alarm No.5013 P/S alarm is issued.

(Example of a correct program)

```
    :
G05 P10000 ;              ┐
    :                     │
G41 X__ Y__ D01 ;  ┐      │
    :              │ Cutter compensation
G40 X__ Y__ ;      ┘ (G41) mode
    :                     │ HPCC mode
G42 X__ Y__ D02 ;  ┐      │
    :              │ Cutter compensation
G40 X__ Y__ ;      ┘ (G42) mode
    :                     │
G05 P0 ;                  ┘
    :
```

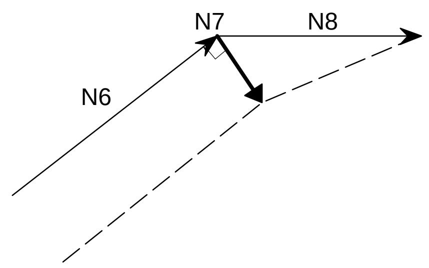
(Example of an incorrect program (1))

```
    :
G41 X__ Y__ D01 ;
    :
G05 P10000 ;  ──→  When the start of HPCC mode is specified
                   in cutter compensation mode, the P/S
                   alarm No.0178 is issued.
```

(Example of an incorrect program (2))

```
    :
G05 P10000 ;
    :
G41 X__ Y__ D01 ;
    :
G05 P0 ;      ──→  When cancellation of HPCC mode is specified
                   in cutter compensation mode, the P/S alarm
                   No.5013 alarm is issued.
```

(2) When a block containing no movement operation is specified together with the cutter compensation cancel code (G40), a vector with a length equal to the offset value is created in a direction perpendicular to the movement direction of the previous block. Cutter compensation mode is canceled while this vector still remains. This vector is canceled when the next move command is executed.




```
:
N6 G91 X100. Z100. ;
N7 G40 ;
N8 X100. ;
:
```

If cutter compensation mode is canceled while a vector still remains and HPCC mode is canceled before a move command is specified, the P/S alarm No.5013 is issued.

```
:
N6 G91 X100. Z100. ;
N7 G40 ;
N8 G05 P0 ;        ——→  The P/S alarm No. 5013 is issued.
:
```

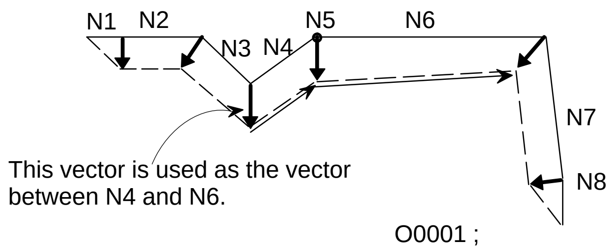
(3) When an offset value is changed during cutter compensation C in HPCC mode, the new offset value is not used until a block specifying a D code appears.

- **Positioning and auxiliary functions**

When bit 1 of parameter MSU No. 8403 is set to 1, G00, M, T, and B codes can be specified even in HPCC mode. When specifying these codes in HPCC mode, note the following:

(1) When a G00, M, S, T, or B code is specified in cutter compensation mode, the offset vector created in the previous block is maintained.

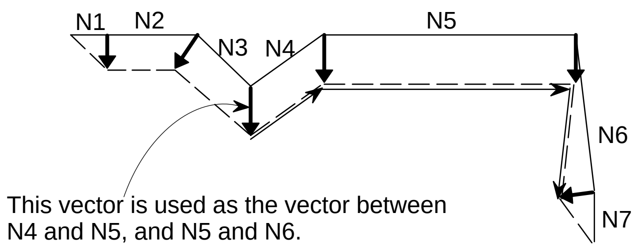
(Example 1) When the following program is executed for machining with offset value D1 set to 10 mm, the start point of N6 is determined by the vector created between N3 and N4:



— Programmed path
– – Beam path
→ An incorrect offset value is used in this range.

```
O0001 ;
G92 G90 X–10. Y20. ;
G05 P10000 ;
N1 G01 G42 X0 D1 F1000 ;
N2 X20. ;
N3 X40. Y0 ;
N4 X60. Y20 ;
N5 M01 ;
N6 X80. ;
N7 X90. Y–20. ;
N8 G40 Y–50. ;
G05 P0 ;
M30
```

(Example 2) When the following program is executed for machining with offset value D1 set to 10 mm, the start point of N5 is determined by the vector created between N3 and N4. If the simplified G00 execution function is enabled (by setting bit 7 of parameter SG0 No. 8403 to 1), a correct vector can be obtained at the intersection of N4 and N5.



— Programmed path
– – Beam path
→ An incorrect offset value is used in this range.

```
O0001 ;
G92 G90 X–10. Y20. ;
G05 P10000 ;
N1 G01 G42 X0 D1 F1000 ;
N2 X20. ;
N3 X40. Y0 ;
N4 X60. Y20 ;
N5 G00 X80. ;
N6 G01 X90. Y–20. ;
N7 G40. Y–50. ;
G05 P0 ;
M30
```

(2) When G00 is specified with bit 7 of parameter SG0 No. 8403 set to 1, the following points should be noted:

·Since the G00 command is replaced by the G01 command, the beam moves at the feedrate set in parameter No. 8481 even when data is specified for two axes.

Example) If the following is specified when parameter No. 8481 is set to 1000 mm/min, F1000 is used instead of F1414
G00 X100. Y100. ;

- Since the G00 command is replaced by the G01 command, rapid traverse override is disabled and cutting feed override is enabled.
- For acceleration/deceleration after interpolation, the time constant used for cutting feed acceleration/deceleration after interpolation is selected.
- Linear and bell–shaped acceleration/deceleration before inter–polation in HPCC mode is enabled.
- No position check is performed.
- Linear interpolation type positioning is performed.

• Status display

When G05P10000 is specified, “HPCC” starts blinking at the right–bottom of the screen. While “HPCC” is blinking, the system performs automatic operation in HPCC mode.

Display example for when the system is in HPCC mode
(Program screen)

```
PROGRAM(MEMORY)                         O1234  N00010
 G05 P10000 ; ──────────────────> Executed block
 N10 X10. Y10. Z10. ;  ──────────> Block being executed
 N20 X10. Y10. Z10. ;
 / N30 X10. Y10. Z10. ;
 /2 N40 X10. Y10. Z10. ;
 N50 X10. Y10. Z10. ;
 N60 X10. Y10. Z10. ;
 N70 (FANUC Series 16) ;
 N80 X10. Y10. Z10. ;
 N90 X10. Y10. Z10. ;
 N100 X10. Y10. Z10. ;
 N110 X10. Y10. Z10. ;
 G05 P0 ;
 MEM STRT MTN  * * *          01 : 23 : 45      HPCC
( PRGRM )(        )(        )( NEXT )( (OPRT) )
```

Limitations

- **Modes that can be specified**

Before G05P10000 can be specified, the following modal values must be set. If they are not set, the P/S alarm No. 5012 is issued.

G code	Meaning
G13.1	Cancels polar coordinate interpolation.
G15	Cancels a polar coordinate command.
G40	Cancels cutter compensation.
G40.1	Cancels normal direction control.
G50	Cancels scaling.
G50.1	Cancels the programmable mirror image function.
G64	Cutting mode
G69	Cancels coordinate conversion.
G94	Feed per minute
M97	Cancels interrupt type macros.

- **Single block**

The G05P10000 block cannot be executed in the single block mode.

- **Second feedrate override and optional block skip**

The second feedrate override and optional block skip functions cannot be used in HPCC mode unless these options are provided.

- **Invalid command**

Externally–requested deceleration, feed at address F with one digit, and automatic corner override commands are ignored.

- **MDI operation**

Switching to the MDI mode cannot be performed in HPCC mode. In addition, MDI operation is not possible.

- **Interlock**

Interlock (for each axis and in each direction) is disabled in HPCC mode.

- **Mirror image and machine lock**

In HPCC mode, never change the external mirror image signal (DI signal), parameter–set mirror image, and each–axis machine lock.

- **Calculator–type input**

In HPCC mode, calculator type input (when bit 0 of parameter DPI No. 3401 is 1) is ignored.

- **Program reset**

A program containing G05P10000; cannot be restarted.

- **Custom macro**

No custom macros can be specified in HPCC mode.

- **Scaling**

In scaling for each axis, a negative magnification cannot be used to create a mirror image.

16.6 HIGH–SPEED LINEAR INTERPOLATION (G05)

The high–speed linear interpolation function processes a move command related to a controlled axis not by ordinary linear interpolation but by high–speed linear interpolation. The function enables the high–speed execution of an NC program including a series of minute amounts of travel.

Format

G05 P2 ; Start high–speed linear interpolation
G05 P0 ; End high–speed linear interpolation

A block for specifying G05 must not contain any other command.

Explanations

- **High–speed linear interpolation mode**

The high–speed linear interpolation start command G05 P2; places the system in high–speed linear interpolation mode, in which high–speed linear interpolation is executed. The high–speed linear interpolation end command G05 P0; places the system in the standard NC program operation mode.
At power–up or in the NC reset state, the system enters the standard NC program operation mode.

- **Commands in high–speed linear interpolation mode**

The commands that can be programmed in high–speed linear interpolation mode are:
X/Y/Z/C–axis incremental travel distance command, cutting feedrate command, and high–speed linear interpolation end command.
In high–speed linear interpolation mode, an address other than those listed in the following table is ignored.

Address	Description
X––.––	X–axis incremental travel distance
Y––.––	Y–axis incremental travel distance
Z––.––	Z–axis incremental travel distance
C––.––	C–axis incremental travel distance
G05 P0 ;	High–speed linear interpolation end command

- **X/Y/Z/C–axis incremental traveling distance**

A travel distance specified in high–speed linear interpolation mode is regarded as being an incremental travel distance, regardless of the G90/G91 mode setting.

- **Cutting feedrate**

Specify a cutting feedrate in high–speed linear interpolation mode. If no cutting feedrate is specified, the modal F value is assumed.

Maximum feedrate	Interpolation period: 8 msec		Interpolation period: 4 msec	
(IS–B, metric input)	122,848	mm/min	245,696	mm/min
(IS–B, inch input)	12,284.8	inch/min	24,569.6	inch/nim

(Maximum feedrate) =

$$122,848 \times \frac{8}{\text{(interpolation period)}} \text{ (IS–B, metric input)}$$

Minimum feedrate	Interpolation period: 8 msec	Interpolation period: 4 msec
(IS–B, metric input)	4 mm/min	8 mm/min
(IS–B, inch input)	0.38 inch/min	0.76 inch/mim

$$(\text{Minimum feedrate}) = 4 \times \frac{8}{(\text{interpolation period})} \quad (\text{IS–B, metric input})$$

- **Interpolation period**

In high–speed linear interpolation mode, the NC interpolation period can be changed. As the interpolation period decreases, the machining speed and precision increase.
IT2, IT1, and IT0 bits (bits 6, 5, and 4 of parameter 7501)

IT2	IT1	IT0	Interpolation period
0	0	0	8 msec in high–speed linear interpolation mode
0	1	0	4 msec in high–speed linear interpolation mode
0	0	1	2 msec in high–speed linear interpolation mode
0	1	1	1 msec in high–speed linear interpolation mode
1	1	1	0.5 msec in high–speed linear interpolation mode

Limitations

- **Controlled axes**

Up to four axes can be controlled. The names of the controlled axes are X, Y, Z, and C. Any other axis name is ignored. Set X, Y, Z, then C in axis name setting parameter 1020.

- **Enabled interpolation**

Only the linear interpolation function can be executed. Circular interpolation and other interpolation functions cannot be executed.

- **Absolute command**

Movement cannot be specified by absolute values. A specified travel distance is always considered as an incremental travel distance, regardless of the G90/G91 mode setting.

- **Feed per revolution**

The feed per revolution command cannot be specified. Feed per minute is always assumed, regardless of the G94/G95 mode setting.

- **Cutter compensation**

High–speed linear interpolation commands cannot be specified in cutter compensation mode (G41/G42). If the high–speed linear interpolation start command is specified in cutter compensation mode, P/S alarm No. 178 is issued.

- **Modes related to the coordinate system**

The high–speed interpolation commands cannot be specified in polar coordinate interpolation mode (G12.1), scaling mode (G51), or coordinate system rotation mode.

- **Single–block operation**

Single–block operation is disabled in high–speed linear interpolation mode.

```
   :
G05 P2  ;            ┐
X10 Z20 F1000 ;      │
   :                 │
   :                 ├ Handled as a single block
   :                 │
Y30 ;                │
G05 P0 ;             ┘
   :
```

- **Feed hold**

Feed hold is disabled in high–speed linear interpolation mode.

- **Cutting feed override**

The cutting feed override function is enabled. Because of the intermediate buffer between high–speed linear interpolation processing and axis move command processing, the override is applied only after the elapse of a slight delay after the override signal is switched.

- **Maximum cutting feedrate for each axis**

The maximum cutting feedrate for each axis (parameter 1430) is invalid in high–speed linear interpolation mode. The maximum cutting feedrate for all axes (parameter 1422) becomes valid.

- **Custom macro/optional block skip**

No macro variables or macro statements can be used in high–speed linear interpolation mode. If their use is attempted, P/S alarm No. 009 is issued. When an optional block skip symbol / is specified, P/S alarm No. 009 is issued as well.

- **Comment**

No comment can be specified.

- **G codes**

If a G code other than G05 P0 is specified in high–speed linear interpolation mode, P/S alarm No. 010 is issued.

Example

<Sample program>

```
NC program
O0001 ;              ┐
G00 X0 Y0 Z0 ;       │
   :                 ├ Standard operation mode
   :                 │
   :                 ┘
G05 P2 ;       <──── High–speed linear interpolation
                     start command
X10 Y20 F1000 ;      ┐ High–speed linear interpolation
X5   Y6   Z7 ;       │ mode (high–speed linear
   :                 ├ interpolation)
   :                 │
   :                 ┘
G05 P0 ;       <──── High–speed linear interpolation
                     end command
G00 DDD ;            ┐
   :                 │
   :                 ├ Standard operation mode
   :                 │
M02 ;                ┘
%
```

16.7 AI CONTOUR CONTROL FUNCTION/AI NANO CONTOUR CONTROL FUNCTION

Overview

The AI contour control/AI nano contour control function is provided for high–speed, high–precision machining. This function enables suppression of acceleration/deceleration delays and servo delays that become larger with increases in the feedrate and reduction of machining profile errors.

Look–ahead acceleration/deceleration before interpolation is enabled for up to 40 blocks in AI contour control or for up to 180 blocks in AI nano contour control. This enables execution of smooth acceleration/ deceleration extending over multiple blocks and higher machining.

The AI nano contour control function calculates a position command to be output to the digital servo in nanometers (nm) with nano–interpolation, so the machine can be moved smoothly and the surface precision is improved.

Explanation

This function is enabled by setting the AI contour control or AI nano contour control mode.

- **Format**

G05.1 Q _ ;

Q1 : AI contour control/AI nano contour control mode on
Q0 : AI contour control/AI nano contour control mode off

NOTE
1 Always specify G05.1 in an independent block.
2 The AI contour control/AI nano contour control mode is also canceled by a reset.
3 When the AI nano contour control option is installed, bit 0 (NAN) of parameter No. 7053 is set to 1 to set the AI contour control mode.

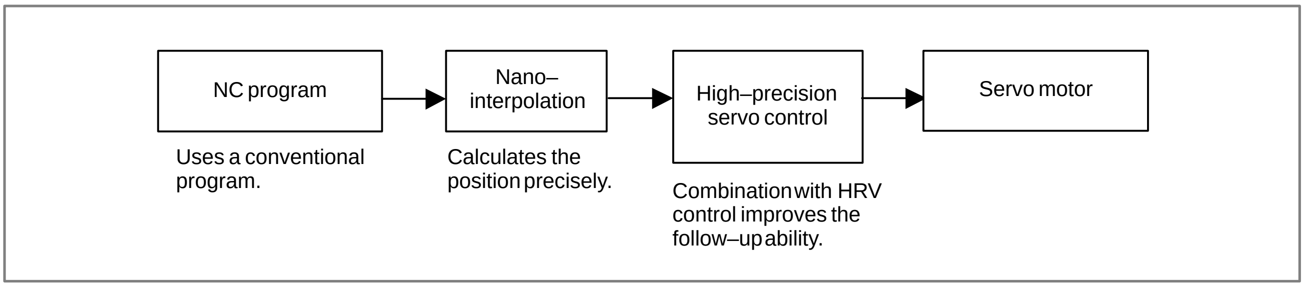
- **Functions valid in the AI contour control/AI nano contour control mode**

The functions listed below are valid in the AI contour control/AI nano contour control mode:

· Nano–interpolation (only in the AI nano contour control mode)
· Look–ahead linear acceleration/deceleration before interpolation
· Look–ahead bell–shaped acceleration/deceleration before interpolation (The option of look–ahead bell–shaped acceleration/deceleration before interpolation is required.)
· Automatic corner deceleration
· Feedrate clamping by acceleration
· Feedrate clamping by arc radius
· Block overlap (up to five blocks)
· Advanced preview feed forward

- **Nano–interpolation (only in the AI nano contour control mode)**

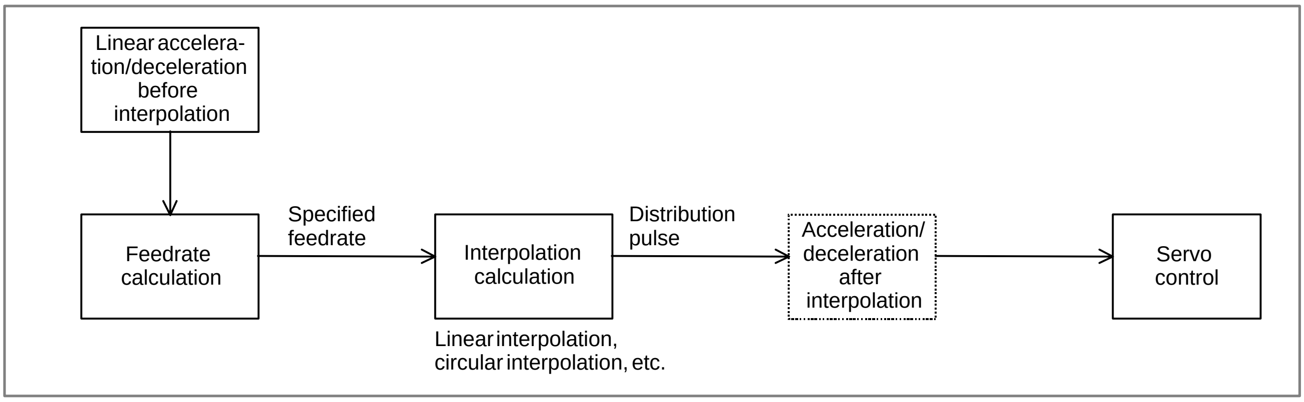
Nano–interpolation calculates a position command to be output to the digital servo in nanometers (nm) for a conventional program. Using the calculation results, a smooth position command is output to the servo, so the machine can be moved smoothly and the surface precision is improved.



> **NOTE**
> The positioning precision depends on the detection unit.

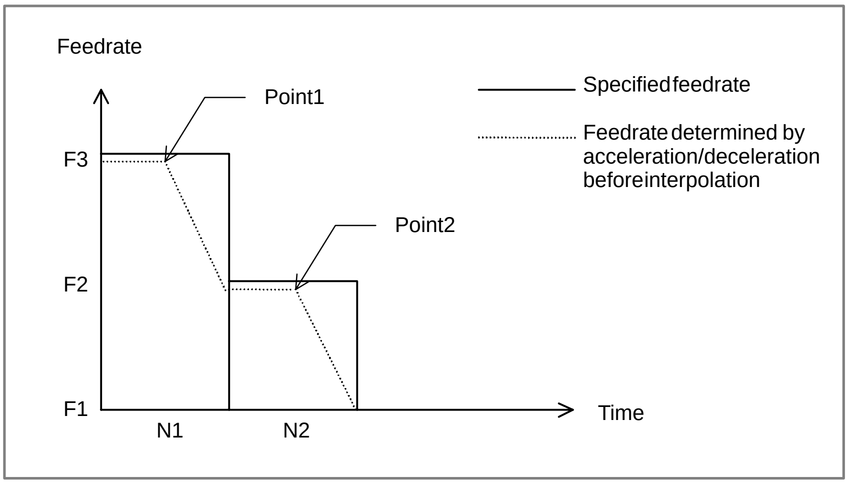
- **Look–ahead linear acceleration/deceleration before interpolation**

For a cutting feed command in the feed per minute mode, linear acceleration/deceleration can be applied before interpolation, that is, for the specified feedrate by reading up to 40 blocks (in the AI contour control mode) or 180 blocks (in the AI nano contour control mode) in advance. With acceleration/deceleration after interpolation, the interpolated data is changed because acceleration/deceleration is applied to the data. With acceleration/deceleration before interpolation, the interpolated data cannot be changed by acceleration/deceleration because acceleration/deceleration is applied to the feedrate data before interpolation. For this reason, the interpolated data can always be applied to the specified straight line or curve to eliminate machining profile errors caused by acceleration/deceleration delays.



(Example of deceleration)

Deceleration is started in a prior block so that the feedrate specified for the target block is reached at the execution.

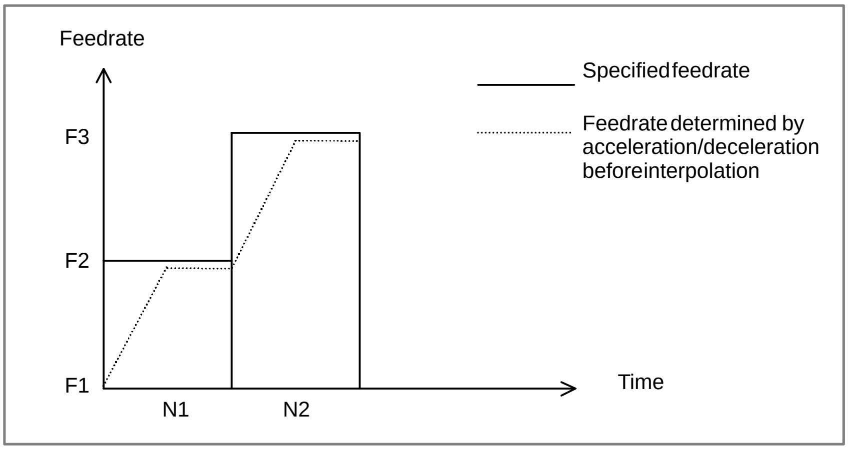


To decelerate from feedrate F3 to F2, deceleration must start with point 1. To decelerate from feedrate F2 to F1, deceleration must start with point 2.

Multiple blocks can be read in advance to perform deceleration extending over several blocks.

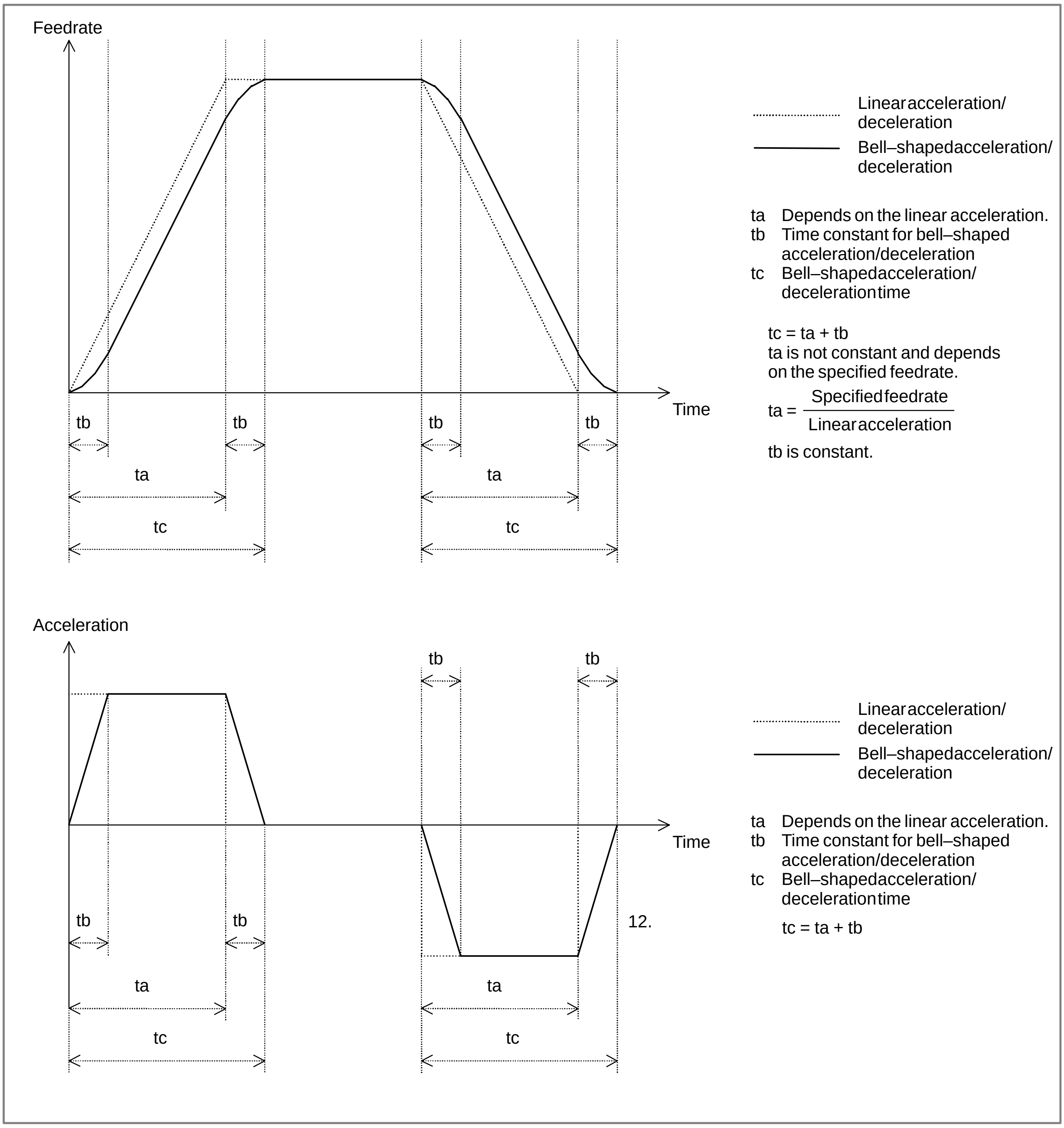
(Example of acceleration)

Acceleration is performed so that the feedrate specified for the target block is reached at the execution.



- **Look–ahead bell–shaped acceleration/deceleration before interpolation**

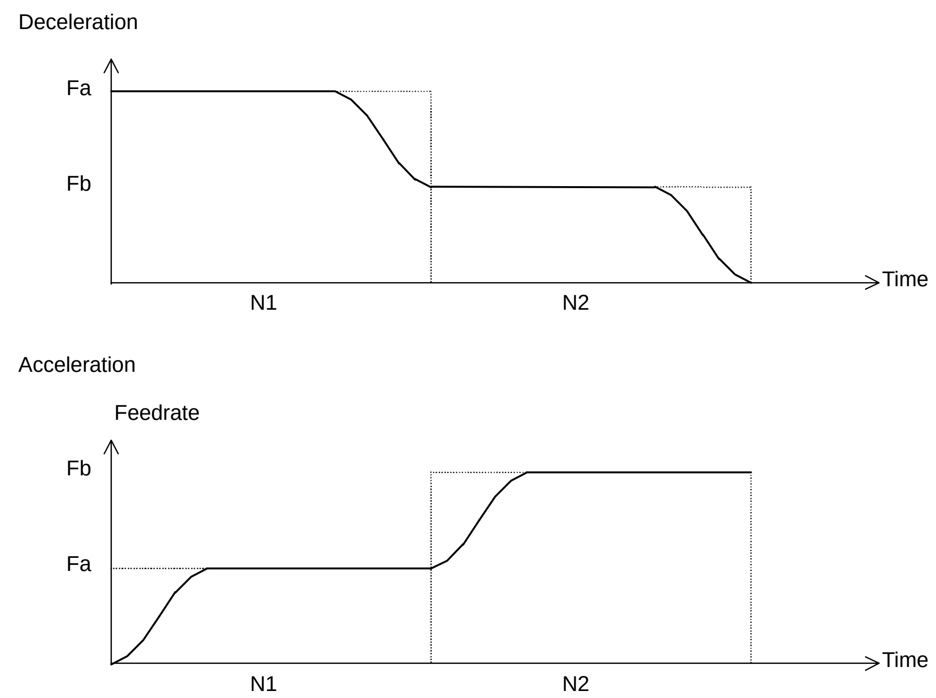
Linear acceleration/deceleration before interpolation for cutting feed in the AI contour control/AI nano contour control mode can be changed to bell–shaped acceleration/deceleration before interpolation. With bell–shaped acceleration/deceleration before interpolation, smooth acceleration/deceleration can be applied to the cutting feedrate to reduce the shock on the machine by fluctuations in acceleration that are involved in changes in the cutting feedrate.
To use this function, the option of look–ahead bell–shaped acceleration/ deceleration before interpolation is required.



When the feedrate is changed, deceleration and acceleration are performed as follows:

For deceleration: Bell–shaped deceleration is started in the preceding block so that deceleration terminates by the beginning of the block in which the feedrate is changed.

For acceleration: Bell–shaped acceleration is started after the beginning of the block in which the feedrate is changed.



- **Automatic corner deceleration**

Between contiguous two blocks, the feedrate difference for an axis may exceed the setting (parameter No. 1783). In this case, the feedrate at the corner is calculated as follows based on the axis for which the ratio of the actual feedrate difference to the allowable feedrate difference is the largest. Deceleration is performed so that the feedrate is reached at the interface of the blocks.

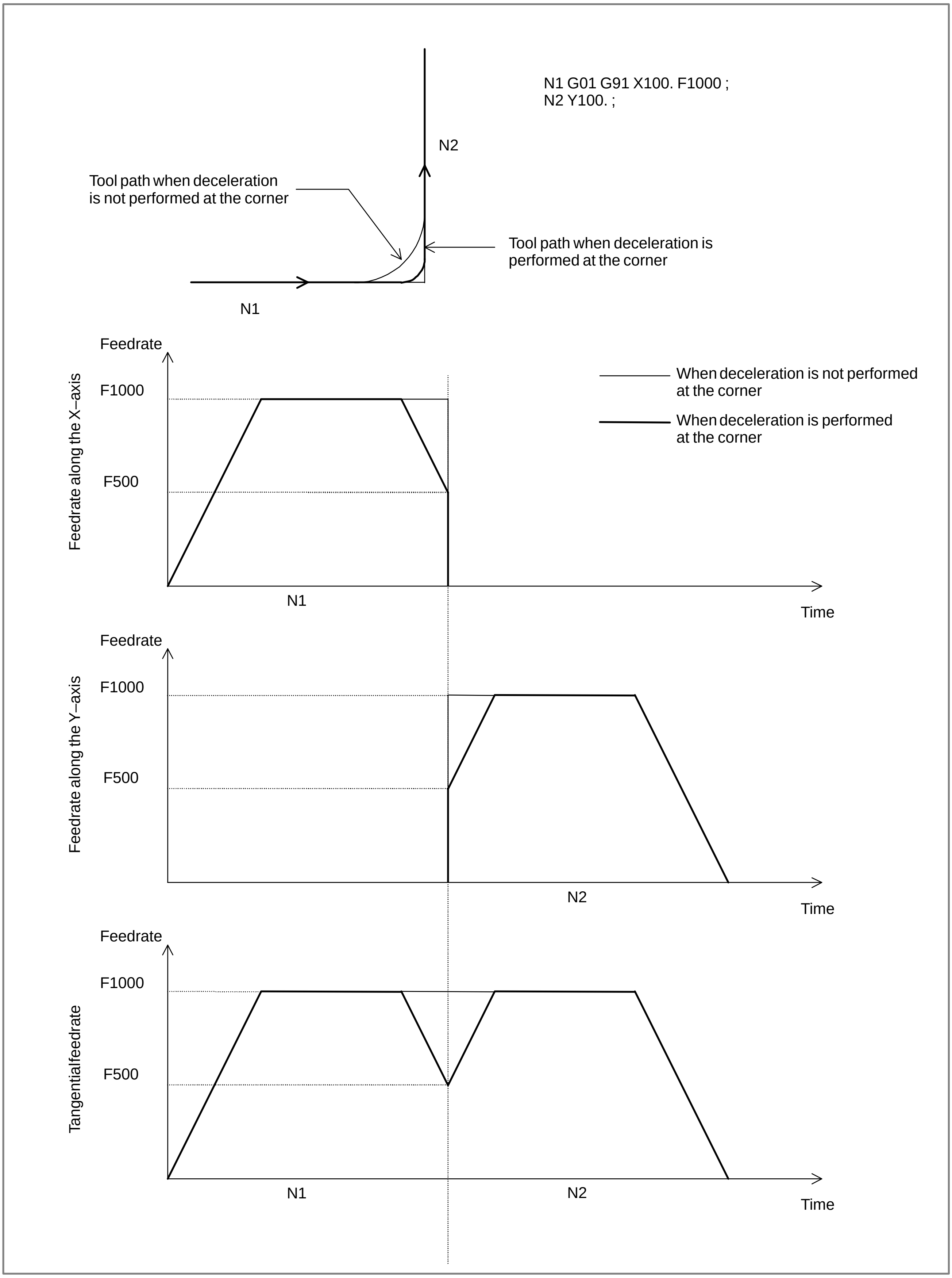
The change in the feedrate for each axis (Vx, Vy, ...) during movement at the specified feedrate F is compared with the setting of parameter No. 1783 (Vprm–x, Vprm–y, ...). If the change in the feedrate for any axis exceeds the setting of the parameter, the following expression is used:

$$\mathrm{Rmax} = \max\left[\frac{\mathrm{Vx}}{\mathrm{Vprm}-\mathrm{x}}, \frac{\mathrm{Vy}}{\mathrm{Vprm}-\mathrm{y}}, , ,\right]$$

The required feedrate (Fc) is obtained as follows and deceleration is performed at the corner:

$$\mathrm{Fc} = \mathrm{F} \times \frac{1}{\mathrm{Rmax}}$$

For example, assume that the tool move direction changes by 90 degrees from movement along the X–axis to that along the Y–axis. Also assume that the specified feedrate is 1000 mm/min and the allowable feedrate difference (parameter No. 1783) is 500 mm/min. Deceleration is performed as shown in the figure below:


N1 G01 G91 X100. F1000 ;
N2 Y100. ;
N2
Tool path when deceleration is not performed at the corner
Tool path when deceleration is performed at the corner
N1
Feedrate
Feedrate along the X–axis
F1000
F500
When deceleration is not performed at the corner
When deceleration is performed at the corner
N1
Time
Feedrate
Feedrate along the Y–axis
F1000
F500
N2
Time
Feedrate
Tangentialfeedrate
F1000
F500
N1
N2
Time

- **Feedrate clamping by acceleration**

When continuous minute straight lines form curves as shown in the example in the figure below, the feedrate difference for each axis at each corner is not so large. For this reason, deceleration according to the feedrate difference is not effective. Continuous small feedrate differences make a large acceleration for each axis as a whole, however.

In this case, deceleration is performed to suppress the shock on the machine and machining errors caused by too large acceleration. The feedrate is decreased so that the acceleration for each axis that is obtained using the expression below does not exceed the allowable acceleration setting for all axes.

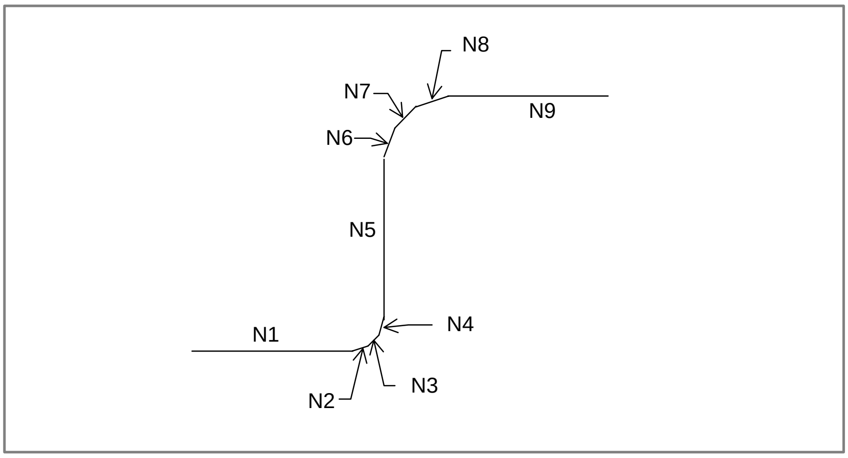
The allowable acceleration is set based on the maximum cutting feedrate (parameter No. 1432) and time required to reach the feedrate (parameter No. 1785).

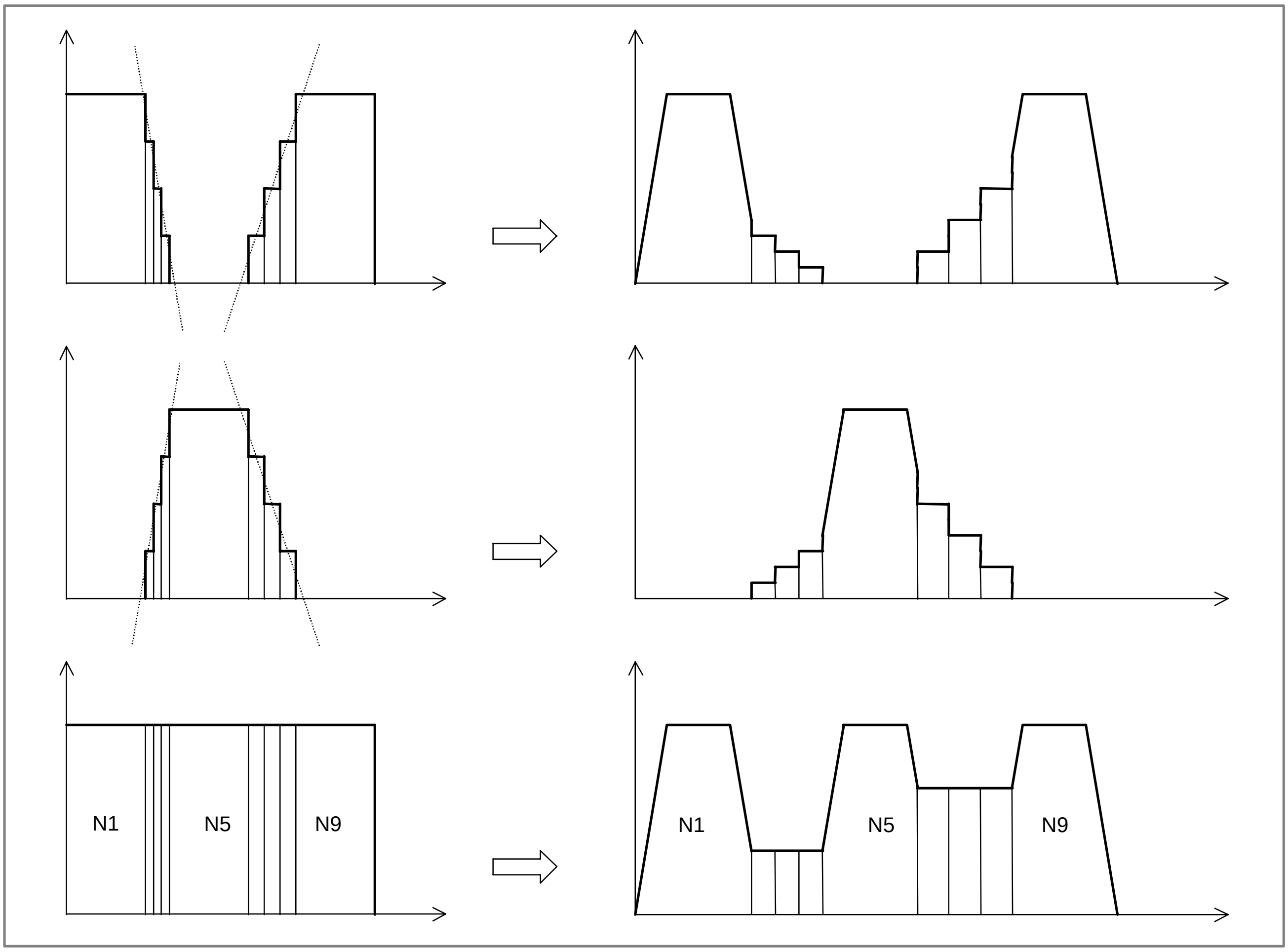
Acceleration for each axis =

$$\frac{\text{Feedrate difference for each axis at each corner}}{\max\left[\dfrac{\text{Travel distance in preceding block}}{F}, \dfrac{\text{Travel distance in following block}}{F}\right]}$$

The decreased feedrate is obtained for each corner. The decreased feedrate obtained at the start or end point of the block, whichever is lower, is used as the actual feedrate.

(Example) In the example below, deceleration is performed because the acceleration (gradient of each dotted line in the feedrate graphs) from N2 to N4 and from N6 to N8 is too large.





- **Feedrate clamping by arc radius**

The maximum allowable feedrate v for an arc of radius r specified in a program is calculated using the arc radius R and maximum allowable feedrate V (setting of a parameter) for the radius as follows so that the acceleration in an arc block does not exceed the allowable value. If the specified feedrate exceeds the feedrate v, the feedrate is automatically clamped to the feedrate v.

$$\text{Maximum allowable feedrate} = \frac{V^2}{R}$$

R : Arc radius V : Feedrate for arc radius R

then, the maximum allowable feedrate v for an arc of radius r can be obtained using the following expression:

$$v = \sqrt{(r/R)} \times V$$

NOTE
As the specified arc radius becomes smaller, the maximum allowable feedrate v becomes lower. If the maximum allowable feedrate v is lower than the setting of parameter No. 1732 (lower feedrate limit for feedrate clamping by arc radius), it can be assumed to be the setting of the parameter to prevent the maximum allowable feedrate from becoming too low.

- **Rapid traverse**

By setting the corresponding parameter, the linear or non–linear interpolation type can be selected. (In the AI nano contour control mode, the non–linear interpolation type cannot be selected.)
When the linear interpolation type is selected, acceleration/deceleration is performed before interpolation and linear interpolation type positioning is used for movement. For acceleration/deceleration, linear or bell–shaped acceleration/deceleration can be selected by setting the corresponding parameter. (To select bell–shaped acceleration/deceleration, the option of bell–shaped acceleration/deceleration for rapid traverse is required.)
The feedrate during movement and acceleration for acceleration/deceleration before interpolation are obtained as follows:

(1) Feedrate during movement

The minimum value obtained using the following expression for each axis along which movement is done is used as the feedrate during movement:

$$\text{Rapid traverse rate for each axis (parameter No. 1420)} = \frac{\text{Amount of travel in block}}{\text{Amount of travel for each axis}}$$

(2) Acceleration for acceleration/deceleration before interpolation

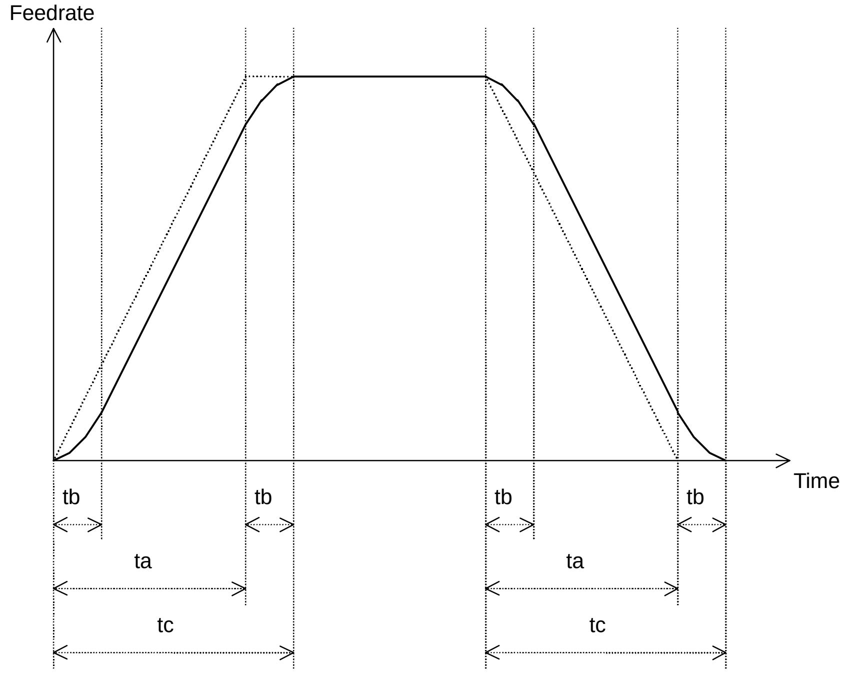
- For linear acceleration/deceleration

The minimum value obtained using the following expression for each axis along which movement is done is used as the acceleration for linear acceleration/deceleration before interpolation during movement:

$$\frac{\text{Rapid traverse rate for each axis (parameter No. 1420)}}{\text{Time constant for each axis (parameter No. 1620)}} \times \frac{\text{Amount of travel in block}}{\text{Amount of travel for each axis}}$$

- For bell–shaped acceleration/deceleration

The time constant set in parameter No. 1621 (time constant for bell–shaped acceleration/deceleration for rapid traverse for each axis) for the axis for which the minimum value is obtained using the above expression is applied to the feedrate obtained using the above acceleration.



Linear acceleration/ deceleration

Bell–shaped acceleration/ deceleration

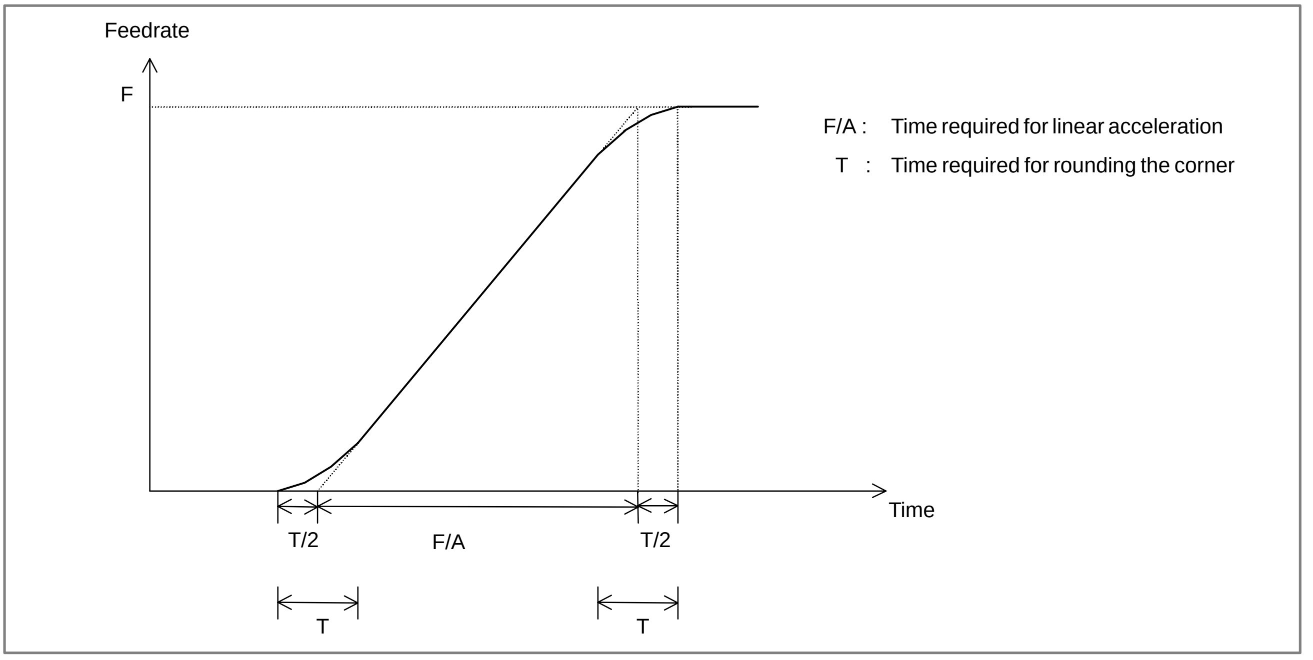
ta Depends on the linear acceleration.
tb Time constant for bell–shaped acceleration/deceleration
tc Bell–shaped acceleration/ deceleration time

tc = ta + tb
ta is not constant, but varies with the rapid traverse rate and rapid traverse time constant of a specified axis.

tb is determined by an axis.

If the feedrate during movement is F, the acceleration for linear acceleration/deceleration is A, the time constant for bell–shaped acceleration/deceleration is T, the time required for acceleration/deceleration can be obtained as follows:

Time required for acceleration/deceleration
= F/A (linear acceleration/deceleration)
= F/A+T (bell–shaped acceleration/deceleration)



When the non–linear interpolation type is selected, movement is performed at the feedrate set in parameter No. 1420 with acceleration/deceleration set in parameter No. 1620. The corresponding value can be set in parameter No. 1621 to select bell–shaped acceleration/deceleration. (To select bell–shaped acceleration/deceleration, the option of bell–shaped acceleration/deceleration for rapid traverse is required.)

NOTE
1 Rapid traverse block overlap is disabled.
2 To use bell–shaped acceleration/deceleration, the option of bell–shaped acceleration/deceleration for rapid traverse is required.

- **Corresponding parameter numbers in the normal mode, advanced preview control mode, and AI contour control/AI nano contour control mode**

In the following tables, AI control means the AI contour control/AI nano contour control mode.

(1) Parameters related to linear acceleration/deceleration before interpolation

Parameter	Parameter number		
	Normal	Advanced preview control	AI contour
Acceleration/deceleration type (A or B)	FWB/1602#0		None
Parameter 1 for setting the acceleration	1630	1770	
Parameter 2 for setting the acceleration	1631	1771	
Feedrate at which an overtravel alarm occurs	1784		

(2) Parameters related to automatic corner deceleration

Parameter	Parameter number		
	Normal	Advanced preview control	AI contour
Method for determining whether to perform automatic corner deceleration (angle/feedrate difference)	CSD/1602#4		None
Lower feedrate limit (control according to the angle)	1778	1777	None
Angle at which automatic corner deceleration is performed (control according to the angle)	1740	1779	None
Allowable feedrate difference for all axes (control according to the feedrate difference)	1780		None
Allowable feedrate difference for each axis (control according to the feedrate difference)	1783		
Acceleration/deceleration type (Type A/Type B)	FWB/1602#0		None

(3) Parameter related to feedrate clamping by acceleration

Parameter	Parameter number		
	Normal	Advanced preview control	AI contour
Parameter for determining the allowable acceleration	None		1785

(4) Parameters related to feedrate clamping by arc radius

Parameter	Parameter number		
	Normal	Advanced preview control	AI contour
Arc radius corresponding to the upper feedrate limit	1731		
Upper feedrate limit at arc radius R	1730		
Lower clamped feedrate limit	1732		

(5) Other parameters

Parameter	Parameter number		
	Normal	Advanced preview control	AI contour
Precision of radius errors in circular interpolation	PCIR1/3403#0		None
Maximum cutting feedrate (common to all axes)	1422	1431	1422
Maximum cutting feedrate (for each axis)	1430	1432	
Rapid traverse movement type*	LRP/1401#1		AIR/ 7054#1 LRP/ 1401#1
Time constant for bell–shaped acceleration/ deceleration for rapid traverse	1621		
Arc radius corresponding to the upper feedrate limit	1731		

* For AI nano–contour control, the rapid traverse movement type is not set with a parameter, but is always linear interpolation positioning.

Alarms

Number	Message	Description
5110	IMPROPER G CODE (G05.1 Q1 MODE)	An unspecifiable G code was specified in the AI contour control/AI nano contour control mode.
5111	IMPROPER MODAL G CODE (G05.1 Q1)	An unavailable modal G code was found when the AI contour control/AI nano contour control mode was specified.
5112	G08 CAN NOT BE COMMANDED (G05.1 Q1)	The look–ahead control command (G08) was specified in the AI contour control/AI nano contour control mode.
5114	CAN NOT ERROR IN MDI MODE (G05.1 Q1)	Intervention by manual operation was performed during execution of the G28, G30, G30.1, or G53 command (linear interpolation type) in the AI contour control mode. After that, automatic operation was restarted at a position other than the stop position.

Number	Message	Description
5156	ILLEGAL AXIS OPERATION (AICC)	In the AI contour control/AI nano contour control mode, the controlled axis selection signal (PMC axis control) changed. In the AI contour control/AI nano contour control mode, the simple synchronous axis selection signal changed.
5157	PARAMETER 0 (AICC)	The setting of the parameter for specifying the maximum cutting feedrate (No. 1422, 1432, or 1420) is 0. The setting of the parameter for specifying acceleration/deceleration before interpolation (No. 1770 or 1771) is 0.

Notes

1) This function requires the AI contour control function or AI nano contour control function option.
When the AI contour control function option is installed, the look–ahead control function (G08P1) can also be specified. When the AI nano contour control function option is installed, the AI contour control function and advanced preview control function (G08P1) can also be specified.
2) When the total distance of blocks read in advance reaches the distance for decelerating from the current feedrate, deceleration is started. When look–ahead operation proceeds and the total distance of blocks increases by termination of deceleration, acceleration is started again. If a series of blocks with a small amount of travel are specified, the deceleration and acceleration may be alternated, which prevents the feedrate from being constant. In this case, specify a lower feedrate.
3) When the dry run signal is inverted from “0” to “1” or from “1” to “0” during movement along an axis, acceleration/deceleration is performed to the specified feedrate without deceleration to feedrate 0.
4) If a non–movement block or one–shot G code command such as G04 is found in the AI contour control/AI nano contour control mode, the movement is decelerated and temporarily stopped in the preceding block.
5) For acceleration/deceleration after interpolation, use linear or bell–shaped acceleration/deceleration. Exponential acceleration/ deceleration cannot be used.
6) This function cannot be used together with the three–dimensional cutting function.

• Specifications

Axis control ○ : Can be specified. × : Cannot be specified.

Name	Function
Number of controlled axes	3 to 8 To use four to eight axes, another option is required.
Number of simultaneously controlled axes	Up to 6 To use three or more simultaneously controlled axes, another option is required.
Axis name	The basic three axes are fixed to X, Y, and Z. Other axes are any of U, V, W, A, B, and C.
Least input increment	0.001 mm, 0.001 deg, 0.0001 inch
Simple synchronous control	○ If the AI contour control/AI nano contour control option is installed, however, switching between synchronous and normal operation cannot be performed during automatic operation (when the automatic operation signal (OP) is set to 1) regardless of whether the AI contour control/AI nano contour control mode is set. In this case, switching causes a PS213 alarm.
Twin table	×
Tandem control	○ Full preloading cannot be performed, however.
Inch/metric conversion (G20, G21)	○ (*1)
Interlock	○
Interlock for each axis	○ Movement along all axes stops. To stop movement only along the interlock axis in non–linear interpolation type positioning in the AI contour control mode, set bit 5 (AIL) of parameter No. 7054 to 1 and bit 4 (XIK) of parameter No. 1002 to 0.
Machine lock	○ When the machine lock signal for each axis (MLK1 to MLK8) is turned on or off, acceleration/deceleration is not applied to the axis for which machine lock is performed.
Stroke check before movement	×
Mirror image	○
Stored pitch error compensation	○
Gradient compensation	○
Straightness compensation	○
Position switch	○ Set bit 3 (PSF) of parameter No. 6901 to 1. When this parameter is set to 1, the signal output timing changes.
Unexpected disturbance torque detection	○
Manual handle interruption	○ (AI contour control) Manual handle interruption is disabled during switching to the AI contour control mode.
	× (AI nano contour control)

Interpolation functions

○ : Can be specified. × : Cannot be specified.

Name	Function
Positioning (G00)	○
Single direction positioning (G60)	○ To perform single direction positioning in the AI contour control/AI nano contour control mode, set bit 4 (ADP) of parameter No. 7055 to 1.
Exact stop (G09)	○
Exact stop mode (G61)	○
Linear interpolation (G01)	○
Circular interpolation (G02, G03)	○ (Circular interpolation for multiple quadrants is enabled.)

Name	Function
Dwell (G04)	○ (Dwell with the time in seconds or speed specified) For dwell with the speed specified, another option is required.
Polar coordinate interpolation (G12.1, G13.1)	×
Cylindrical interpolation (G07.1)	×
Helical interpolation (G02, G03)	○ (Circular interpolation + linear interpolation for up to four axes) When the helical interpolation option is installed, linear interpolation for up to two axes is performed. When the helical interpolation B option is installed, linear interpolation for up to four axes is performed. Specify the feedrate including the helical axis in the feedrate command.
Hypothetical axis interpolation (G07)	×
Skip function (G31)	○ (*1)
High–speed skip function (G31)	○ (*1)
Multistage skip function (G31 Px)	○ (*1)
Reference position return (G28)	○ (*1) To execute G28 in the status in which the reference position is not established, set bit 2 (ALZ) of parameter No. 7055 to 1.
Reference position return check (G27)	○ (*1)
2nd, 3rd, and 4th reference position return (G30)	○ (*1)
Floating reference position return (G30.1)	○ (*1)
Normal direction control (G41.1, G42.1)	○ (AI contour control) Set bit 2 (ANM) of parameter No. 5484 to 1.
	× (AI nano contour control)
High–speed cycle cutting (G05)	×
High–speed linear interpolation (G05P2)	×

Feed functions ○ : Can be specified. × : Cannot be specified.

Name	Function
Rapid traverse rate	Up to 240 m/min (0.001 mm)
Rapid traverse rate override	F0, 25, 50, 100 %
Rapid traverse rate override in increments of 1%	0 to 100 %
Feed per minute (G94)	○
Cutting feedrate clamp	○
Bell–shaped acceleration/deceleration for rapid traverse	○
Positioning according to the optimum acceleration	×
Linear acceleration/deceleration after cutting feed interpolation	○
Bell–shaped acceleration/deceleration after cutting feed interpolation	○
Linear acceleration/deceleration before cutting feed interpolation	○ (Up to 40 blocks are read in advance in the AI contour control mode.) (Up to 180 blocks are read in advance in the AI nano contour control mode.)
Feedrate override	0 to 254 %
Second feedrate override	×

Name	Function
One–digit F code feed	○ To enable feedrate change using a manual handle, set bit 1 (AF1) of parameter No. 7055 to 1.
Override cancel	○
External deceleration	○
Look–ahead bell–shaped acceleration/deceleration before interpolation	○
High–precision contour control (G05P10000)	○

Program input ○ : Can be specified. × : Cannot be specified.

Name	Function
Control in/control out command ()	○
Optional block skip command (/n: n is a number.)	○
Absolute command (G90)/ incremental command (G91)	○
Decimal point programming/pocket calculator type decimal point programming	○
10–fold input unit	○
Plane selection (G17, G18, G19)	○
Rotation axis specification	○
Rotation axis roll over	○
Polar coordinate command (G16)	×
Local coordinate system (G52)	○ (*1)
Machine coordinate system (G53)	○ (*1)
Workpiece coordinate system (G54 to G59) (G54.1Pxx)	○
Workpiece coordinate system (G92)	○ (*1)
Workpiece coordinate system preset (G92.1)	○ (*1)
Arbitrary angle chamfering/corner rounding	×
Programmable data input (G10)	○ (*1) Only the tool offset value, workpiece origin offset, and parameter can be changed.
Custom macro B	○ See the description of ”Notes on using custom macros.”
Addition of custom macro common variables	○
Pattern data input	×
Interruption type custom macro	×
Arc radius R programming	○
Automatic corner override (G62)	○ Set bit 0 (ACO) of parameter No. 7055 to 1.
Automatic corner deceleration	○
Feedrate clamping by arc radius	○
Scaling (G51)	○
Coordinate system rotation (G68)	○
Programmable mirror image (G51.1)	○
Retrace	×
F15 tape format	○

Auxiliary functions/spindle–speed functions

○ : Can be specified. × : Cannot be specified.

Name	Function
Auxiliary function (Mxxxx)	○ The function code and function strobe signals are output only.
Second auxiliary function (Bxxxx)	○ The function code and function strobe signals are output only.
High–speed M/S/T/B interface	○
Multiple miscellaneous function specification	○
M code group check	○

Laser function

○ The command can be issued.
× The command cannot be issued.

Name	Function
Laser output command	○ G01S_P_Q_; Can be issued.
Piercing command (G24)	○ High–speed piercing is included.
Assist gas command (G32)	○
Z–axis gap control (G13/G14)	○ Performing Z–axis gap control prevents the Z–axis from being subjected to AI contour control.
Power control (G63)	○
Machining condition setting function	○ The E code command can be issued. However, neither a cutter compensation amount nor a feedrate can be specified.
Edge cutting and start–up cutting	○
Step control	×
Beam delay control	○ A delay amount is specified in a separate parameter.
Cutting restart function	×
Constant–optical path length control	○ A mirror axis under constant–optical path length control cannot be submitted to AI contour control.
Extended buffer	○
Interaction control	○
Attitude control A and B	○ Cannot be used simultaneously with helical interpolation B.
Three–dimensional transform function (G98/G99)	×
Three–dimensional coordinate transform function (G68/G69)	×
Spatial corner rounding insertion (G33/G34)	×
Spatial arc interpolation (G12)	×
Near–point search	×
Machining head A–axis length compensation function (G71)	×
Program restart (laser specification)	×

Tool compensation functions

○ : Can be specified. × : Cannot be specified.

Name	Function
Tool function (Txxxx)	○ The function code and function strobe signals are output only.
Tool offset memory B	○
Tool offset memory C	○
Tool offset (G45 to G48)	×
Cutter compensation C (G40, G41, G42)	○

Other functions ○ : Can be specified. × : Cannot be specified.

Name	Function
Cycle start/feed hold	○
Dry run	○
Single block	○
Sequence number comparison and stop	○
Program restart	○ For the time constant for acceleration/deceleration during movement to the restart position, the following parameters are used: When exponential acceleration/deceleration is used: Parameter Nos. 1624 and 1625 When linear/bell–shaped acceleration/deceleration is used: Parameter No. 1622 To set the acceleration/deceleration type, use bits 0 and 1 of parameter No. 1610.
Macro executor (execution macro)	×
MDI operation	○
Manual intervention	○

NOTE
1 The above tables include a function that another option is required for specifying.
2 Multiple blocks are not read in advance.

- **Conditions for setting the AI contour control/AI nano contour control mode**

When G05.1 Q1 is specified, the modal G codes must be set as listed below. If one of these conditions is satisfied, a PS5111 alarm occurs.

G code(s)	Description
G00 G01 G02 G03	Positioning Linear interpolation Circular interpolation (CW) Circular interpolation (CCW)
G13.1	Polar coordinate interpolation cancel mode
G15	Polar coordinate command cancel
G40	Cutter compensation cancel
G40.1	Normal direction control cancel mode
G50	Scaling cancel
G50.1	Programmable mirror image cancel
G64	Cutting mode
G67	Macro modal call cancel
G69	Coordinate system rotation cancel
G94	Feed per minute

17 AXIS CONTROL FUNCTIONS

17.1 SIMPLE SYNCHRONOUS CONTROL

It is possible to change the operating mode for two or more specified axes to either synchronous operation or normal operation by an input signal from the machine.
Synchronous control can be performed for up to four pairs of axes with the Series 16, according to the parameter setting (parameter No. 8311).
The following operating modes are applicable to machines having two tables driven independently by separate control axes. The following example is of a machine with two tables driven independently by the Y axis and V axis. If the axis names and axis sets that are actually being used differ from those in the example, substitute the actual names for those below.

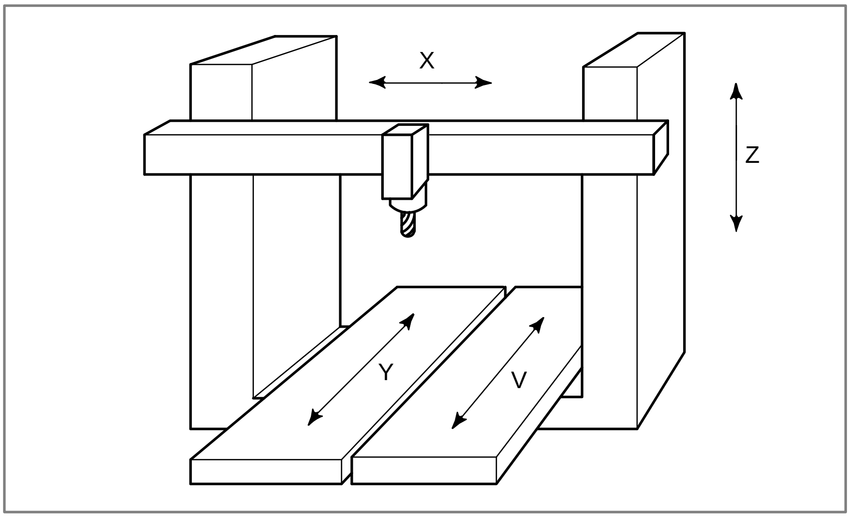


Fig.17.1 Example of axis configuration of the machine operated by simple synchronous control

Explanations

- **Synchronous operation**

This mode is used for, for example, machining large workpieces that extend over two tables.
While operating one axis with a move command, it is possible to synchronously move the other axis. In the synchronous mode, the axis to which the move command applies is called the master axis, and the axis that moves synchronously with the master axis is called the slave axis. In this example, it is assumed that Y axis is the master axis and V axis is the slave axis. Here, the Y axis and the V axis move synchronously in accordance with program command Yyyyy issued to the Y axis (master axis).
Synchronous operation is possible during automatic operation, jog feed, manual handle feed using the manual pulse generator, and incremental feed.

• Normal operation

This operating mode is used for machining different workpieces on each table. The operation is the same as in ordinary CNC control, where the movement of the master axis and slave axis is controlled by the independent axis address (Y and V). It is possible to issue the move commands to both the master axis and slave axis in the same block.

(1) The Y axis moves normally according to program command Yyyyy issued to the master axis.

(2) The V axis moves normally according to program command Vvvvv issued to the slave axis.

(3) The Y axis and the V axis move simultaneously according to program command YyyyyVvvvv.
Both automatic and manual operations are the same as in ordinary CNC control.

• Switching between synchronous operation and normal operation

For how to switch between the synchronous operation and normal operation modes, refer to the relevant manual published by the machine beam builder.

• Automatic reference position return

When the automatic reference position return command (G28) and the 2nd/3rd/4th reference position return command (G30) are issued during synchronous operation, the V axis follows the same movement as the Y axis returns to the reference position. If the V axis is positioned at the reference position after the return movement is complete, the reference position return complete signal of the V axis goes on when that of the Y axis goes on.
As a rule, commands G28 and G30 must be issued in the normal operating mode.

• Automatic reference position return check

When the automatic reference position return check command (G27) is issued during synchronous operation, the V axis and Y axis move in tandem. If both the Y axis and the V axis have reached their respective reference positions after the movement is complete, the reference position return complete signals go on. If either axis is not at the reference position, an alarm is issued. As a rule, command G27 must be issued in the normal operating mode.

• Specifying the slave axis

When a move command is issued to the slave axis during synchronous operation, a P/S alarm (No. 213) is issued.

• Master axis and slave axis

The axis to be used as the master axis is set in parameter No. 8311.
The slave axis is selected by an external signal.

• Displaying actual speed for master axis only

Setting bit 7 (SMF) of parameter No. 3105 to 1 suppresses display of the actual speed of the slave axes.

Limitations

- **Setting a coordinate system**

In synchronous axis control, commands that require no axis motion, such as the workpiece coordinate system setup command (G92) and the local coordinate system setup command (G52), are set to the Y axis by program command Yyyyy issued to the master axis.

- **External deceleration, interlock, and machine lock**

For signals such as external deceleration, interlock, and machine lock, only the signals issued to the master axis are valid in the synchronous operating mode. Signals issued to other axes are ignored.

- **Pitch error compensation**

Both the pitch error and backlash are compensated independently for the master axis and the slave axis.

- **Manual absolute**

Turn on the manual absolute switch during synchronous operation. If it is off, the slave axis may not move correctly.

- **Synchronization error check using positional deviation**

The difference between the master axis and slave axis in servo positional deviation is always monitored. If the difference exceeds the parameter–set limit, an P/S alarm (No. 213) is issued.

- **Synchronization error check using machine coordinates**

The difference between the master axis and slave axis in machine coordinates is always monitored. If the difference exceeds the parameter–set limit, an P/S alarm (No. 407) is issued.

- **Synchronization**

When the power is turned on, compensation pulses are output for the slave axis to match the machine position of the master axis with the machine position of the slave axis. (This is enabled only when the absolute position detection function is used.)

- **Compensation for out–of–synchronism**

Compensation for out–of–synchronism (where the difference between the master and slave axes in servo positional deviation is always monitored and the servo motor for the slave axis is compensated to reduce the difference) is not performed.

- **Manual reference position return**

When the machine is manually returned to the reference position during synchronous operation, both the master axis and the slave axis move synchronously until the acceleration movement is complete. However, grid detection thereafter is carried out independently.

17.2 ROTARY AXIS ROLL–OVER

The roll–over function prevents coordinates for the rotation axis from overflowing. The roll–over function is enabled by setting bit 0 of parameter ROAx 1008 to 1.

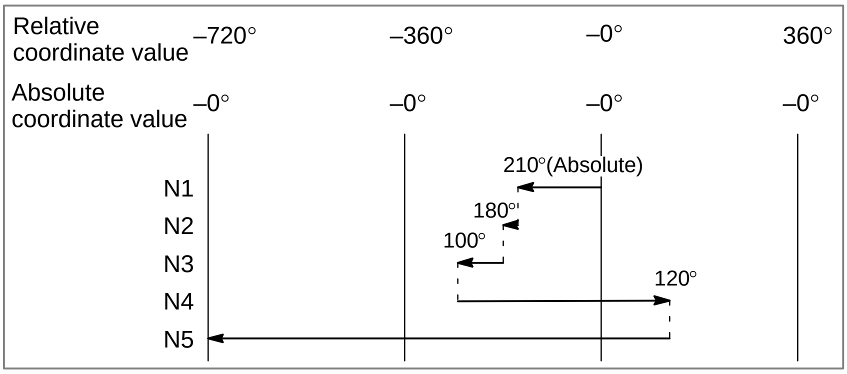
Explanations

For an incremental command, the tool moves the angle specified in the command. For an absolute command, the coordinates after the tool has moved are values set in parameter No. 1260, and rounded by the angle corresponding to one rotation. The tool moves in the direction in which the final coordinates are closest when bit 1 of parameter RABx No. 1008 is set to 0. Displayed values for relative coordinates are also rounded by the angle corresponding to one rotation when bit 2 of parameter RRLx No. 1008 is set to 1.

Examples

Assume that axis A is the rotating axis and that the amount of movement per rotation is 360.000 (parameter No. 1260 = 360000). When the following program is executed using the roll–over function of the rotating axis, the axis moves as shown below.

G90 A0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 G90 A–150.0 ;	N1	–150	210
N2 G90 A540.0 ;	N2	–30	180
N3 G90 A–620.0 ;	N3	–80	100
N4 G91 A380.0 ;	N4	+380	120
N5 G91 A–840.0 ;	N5	–840	0



17.3 TANDEM CONTROL

When enough torque for driving a large table cannot be produced by only one motor, two motors can be used for movement along a single axis. Positioning is performed by the main motor only. The submotor is used only to produce torque. With this tandem control function, the torque produced can be doubled.

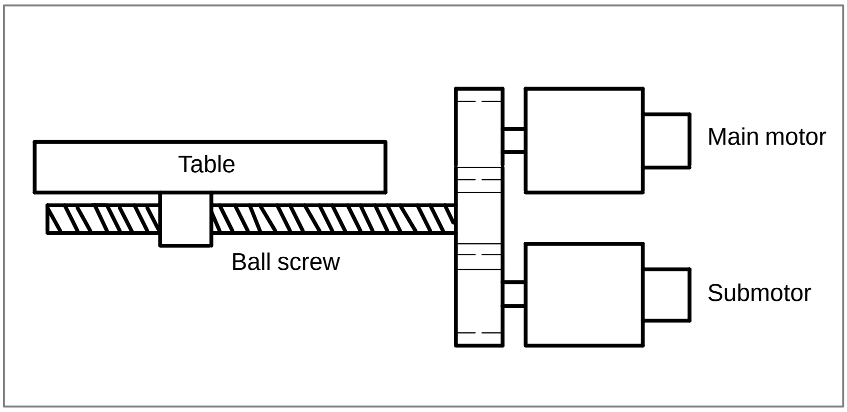


Fig.17.3 Example of operation

In general, the NC regards tandem control as being performed for one axis. However, for servo parameter management and servo alarm monitoring, tandem control is regarded as being performed for two axes. For details, refer to the relevant manual published by the machine tool builder.

18 SPECIFYING THE LASER FUNCTION

18.1 CONTOUR MACHINING (G01, G02, G03, AND G12)

The laser power can be controlled in a block containing machining commands (linear interpolation G01 and circular interpolation G02 and G03).

Format

$$\left.\begin{array}{l}\textbf{G01}\\\textbf{G02}\\\textbf{G03}\end{array}\right\}\ \textbf{S\_P\_Q\_;}$$

S_: Peak power (W)
P_: Pulse frequency (Hz)
Q_: Pulse duty (%)

Explanations

- **Control method**

When a machining command is specified with S, P, and Q, the values specified for S, P, and Q are set for the corresponding parameters, so that the laser beam is controlled according to these parameters.Thus, S, P, and Q, described in Format, need not necessarily be specified each time a command is specified.
If any of the items is not specified, the value set previously for the corresponding parameter is assumed.

- **Peak power**

Specifies the peak power of the laser.

- **Pulse frequency**

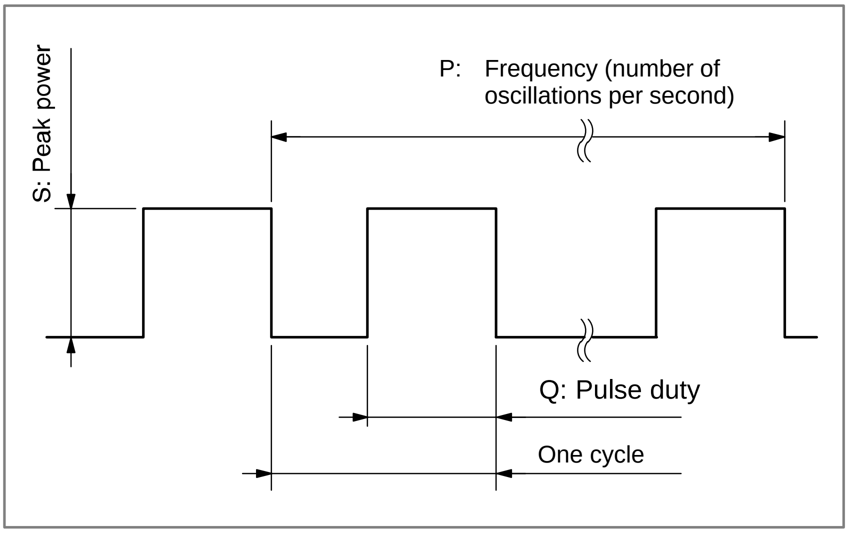
Specifies the pulse frequency at which laser beam pulses are to be output.

- **Pulse duty**

Specifies the pulse duty (ratio of the pulse ON time to the pulse period) at which laser beam pulses are to be output.

Table 18.1 Valid Ranges

Item	Valid range	Unit	Parameter
S_	0 to 7000W	1W	15040
P_	5 to 2000Hz	1Hz	15041
Q_	0 to 100%	1%	15042



Limitations

- **Limitations on the output power**

The peak power that can actually be output is limited by the capacity of the laser oscillator. Thus, the maximum and minimum peak powers that can be specified are specified by parameters (PRM.15210 for the maximum, PRM.15211 for the minimum).

- **Override**

Override can be applied to each of the items, peak power, pulse frequency, and pulse duty.

Item	Valid override range	Unit
S_	0 to 200%	1%
P_	0 to 200%	1%
Q_	0 to 150%	10%

- **Pulse frequency resolution**

Although the minimum unit of the pulse frequency is 1 Hz, the resolution is 0.01 ms. The actual minimum unit depends on this resolution.

18.2 PIERCING FUNCTION (G24)

Stable piercing can be performed in the shortest time by changing the output during piercing (drilling) in a step fashion to achieve the optimum power.

Format

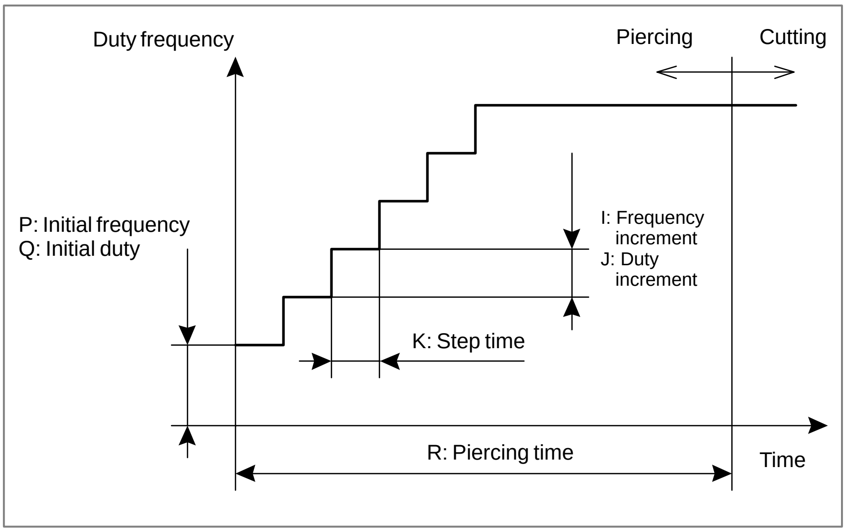
G24 S_P_Q_I_J_K_H_R_;

S_ : Peak power (W)
P_ : Initial pulse frequency (Hz)
Q_ : Initial pulse duty (%)
I_ : Pulse frequency increment (Hz)
J_ : Pulse duty increment (%)
K_ : Step time (msec)
H_ : Number of steps
R_ : Piercing time (msec)

Explanations

- **Control method**

The average output is increased by increasing the pulse frequency and the pulse duty for every unit time while maintaining the peak power of the laser beam.
When G24 is specified with S to R, the values specified for S to R are set for the corresponding parameters, so that the laser beam is controlled with these parameters.
Thus, S to R, described in Format, need not necessarily be specified each time the command is specified. If any of the items is not specified, the value set previously for the corresponding parameter is assumed. If, however, the number of steps (H) is omitted, the value set for the parameter is not used, but the number of steps is assumed to be 0. This is in order to maintain compatibility with older models.



- **Peak power**

Specifies the peak power during piercing.

- **Initial pulse frequency**

Specifies the initial pulse frequency during piercing.

- **Initial pulse duty**

Specifies the initial pulse duty (ratio of the pulse ON time to the pulse period) during piercing.

- **Pulse frequency increment**

Specifies the pulse frequency increment per step during piercing.

- **Pulse duty increment**

Specifies the increment of the pulse duty (ratio of the pulse ON time to the pulse period) per step during piercing.

- **Step time**

Specifies the duration of a single step during piercing.

- **Number of steps**

Specifies the total number of steps during piercing.

- **Piercing time**

Specifies the duration that piercing is to be performed. The duration must be greater than the step time multiplied by the number of steps. If it is shorter, piercing will terminate once the piercing time has elapsed, before the specified number of steps are executed.

Table 18.2 Valid Ranges

Item	Valid range	Unit	Parameter
S_	0 to 7000W	1W	15080
P_	5 to 2000Hz	1Hz	15081
Q_	0 to 100%	1%	15082
I_	0 to 2000Hz	1Hz	15083
J_	0 to 100%	1%	15084
K_	0 to 32767msec	1msec	15085
H_	0 to 32767	1	15086
R_	0 to 99999999msec	1msec	15087

- **Data change command**

If only the data for piercing is to be changed without performing piercing, G10 can be used to specify the parameters corresponding to the desired data items.

G10 L50;	Data setting mode
N_R_;	N=Parameter number, R=Data item
N_R_:	N=Parameter number, R=Data item
. . . . .	
G11;	Data setting mode cancel

- **Function for changing the piercing time externally**

During piercing, the piercing time can be extended or shortened by using an external signal.

- When ”Extend” is performed during piercing, the final continuation time is extended.
- When ”Shorten” is performed during piercing, piercing terminates at the rising edge of a signal.
- Whether the piercing time set for a parameter is to be updated with the actual piercing time when Extend or Shorten is performed can be specified using a parameter.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
15003							HPT	

HPT Specifies whether the piercing time is to be updated when Extend or Shorten is performed.

1 : Does not update the time.
0 : Updates the time (default).

Limitations

- **Override**

Override cannot be applied to the data items for piercing (peak power, pulse frequency, pulse duty, etc.).

18.3 LASER POWER CONTROL (G63)

The laser power control function ensures uniform machining by controlling the peak power, pulse frequency, and pulse duty when the actual feedrate changes from that specified for corners and other locations.

Format

G63 P1;	**Laser power control mode**
G63 P0;	**Laser power control mode cancel**

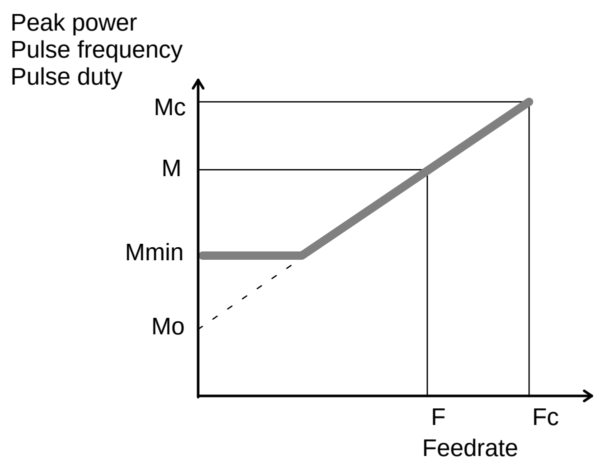
NOTE
G63 must be specified independently. (Do not specify other G codes in the block.)

Explanations

- **Control method**

The laser power control function controls the power specification, pulse frequency, and pulse duty, using the formula shown below:

$$M = Mo + (Mc–Mo)\frac{F}{Fc}$$



Mo : Power, frequency, or duty at a feedrate (F) of 0.
Set with a parameter.

Mmin : Minimum clamp value for the power, frequency, or duty (parameter).
Set with a parameter.

Mc : Power, frequency, or duty specified when laser power control mode is ON.
Set with a program.

Fc : Feedrate specified when laser power control mode is ON.
Set with a program.

Fig.18.3 Relationship between the Feedrate and the Beam Output Conditions

- **Specifying whether to enable or disable the function**

Whether the laser power control function is to be enabled or disabled can be specified using the laser setup screen or a parameter.

1) Specification using the laser setup screen
The laser power control function is enabled by setting the power control item on the laser setup screen to 1.
- Power control (on the laser setup screen)
1: The laser power control function can be used.
0: The laser power control function cannot be used. When this value is set, laser power control mode, described later, cannot be set.

2) Specification using a parameter
The laser power control function is enabled by setting bit 0 (LPC) of parameter No. 15000 to 1.
- Bit 0 (LPC) of parameter No. 15000
1: The laser power control function can be used.
0: The laser power control function cannot be used. When this value is set, laser power control mode, described later, cannot be set.

- **Selecting the control item**

Any of the control items, peak power, pulse frequency, and pulse duty, can be selected using a parameter (PRM.15096). More than one item can be selected.
If none of these items is selected, no power control is performed.

- **Axes that can be subjected to power control**

The power control function uses the nozzle feedrate to calculate each laser output factor (peak power, pulse frequency, and pulse duty). To get hold of this feedrate accurately, you must specify which machine axis is related to the operation of the nozzle, using parameter No. 15089.
If this parameter is omitted, the first and second axes are assumed to have been set up.
An axis subjected to simplified synchronization control is related to the movement of the nozzle. This axis, together with the master axis, should be excluded from the setting.

- **Laser power control mode**

If the laser power control function is enabled using the laser setup screen or the appropriate parameter, the peak power, pulse frequency, and pulse duty can be controlled in proportion to the feedrate by specifying G63P1.
The system is returned from laser power control mode either by specifying G63P0 or by entering a reset signal (laser power control mode is canceled).

- **Specification with E codes**

The power control mode can be turned on and off also by specifying E codes.
Ten types of cutting conditions for the power control function can be registered as cutting data for the cutting condition setting function.
These condition data types are assigned E901 to E910 and are treated as an auxiliary data group of cutting condition data (E1 to E10).
Each set of cutting condition data includes an item for setting power control conditions. When this cutting condition item is specified in a machining program, the corresponding power control conditions are transferred to parameters, and the power control mode is entered.
Based on the feedrate, power, frequency, and duty cycle set for specified cutting conditions (E1 to E10), the rates of change in output conditions from the power, frequency, and duty cycle at a feedrate of 0 set in specified power control conditions are calculated according to the change in feedrate.
To cancel the power control function, specify cutting conditions that do not include any power control condition setting (0 is set in the power control condition item).
The following power control conditions can be registered:

Item	Transfer destination parameter	Setting range	Increment system
Minimum peak power (clamp value)	15090	0 to 9999	W
Peak power at a feedrate (F) of 0	15097	0 to 9999	W
Minimum pulse frequency (clamp value)	15091	5 to 2000	Hz
Pulse frequency at a feedrate (F) of 0	15098	5 to 2000	Hz
Minimum pulse duty cycle (clamp value)	15092	0 to 100	%
Duty cycle at a feedrate (F) of 0	15099	0 to 100	%
Allowable variation in feedrate (dead–band width)	15450	0 to 32767	mm/min or %

The following show the cutting data screen and the power control data screen:

```
[CUTTING]    ACTIVE DATA NO. CUTTING = 5    PIERCING = 103

NO.  FEED    PWR  FREQ   DUTY   GAS   GAS  GAS  GAP  OFSET    EDGE START  POWER
                                PRESS.KIND TIME                    UP     CTL

 1   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
 2   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
 3   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
 4   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
 5   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***

 6   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
 7   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
 8   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
 9   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
10   ***.**  **** ****   ***    **.*   *   *.* **.**** ***.**** *** ***   ***
```

Cutting data screen

```
[POWER CONTROL]     ACTIVE DATA NO. CUTTING = 5     PIERCING = 103

  NO.        POWER          FREQUENCY          DUTY      PWR./SPEED VAR.
          MIN    F=0      MIN    F=0       MIN    F=0

  901     ****   ****     ****   ****      ***    ***        *****
  902     ****   ****     ****   ****      ***    ***        *****
  903     ****   ****     ****   ****      ***    ***        *****
  904     ****   ****     ****   ****      ***    ***        *****
  905     ****   ****     ****   ****      ***    ***        *****

  906     ****   ****     ****   ****      ***    ***        *****
  907     ****   ****     ****   ****      ***    ***        *****
  908     ****   ****     ****   ****      ***    ***        *****
  909     ****   ****     ****   ****      ***    ***        *****
  910     ****   ****     ****   ****      ***    ***        *****
```

Power control data screen

- **Specification with an external signal**

Power control mode can be turned on/off using an external signal. For details, refer to the “FANUC Series 16*i*/160*i*–LB Connection Manual (B–63663EN).”

- **Using the function with the machining condition setting functions (E code commands)**

If, in power control mode, an E code is specified to change the machining conditions, the items corresponding to Mc and Fc in Fig 18.3 are changed with the corresponding machining conditions specified with the E code. Then, processing continues.
If, however, the edge machining/startup machining (E201 to E205) or the step function (E301 to E305) is specified, this takes precedence and power control mode is canceled.

- **Using the function together with the edge cutting, start–up cutting, or step control function**

If a step control command (E301 to E305) is issued, it is impossible to shift to the power control mode. If a step control command is issued in the power control mode, the power control mode is canceled automatically. If an edge cutting/start–up cutting command (E201 to E205) is issued, the behavior of the machine varies depending on the setting of bit 7 (EGM) of parameter No. 15096.

If EGM = 0
When the edge cutting/start–up cutting mode is entered (E201 to E205 is issued in the machining program), the power control mode is canceled.

If EGM = 1
The power control mode is not canceled even when the edge cutting/start–up cutting mode is entered. Power control is kept at a halt after corner edge piercing begins until a movement through a return distance is completed. When power control is at a halt, beam output conditions are controlled in edge cutting/start–up cutting.

• When using G01 to specify positioning

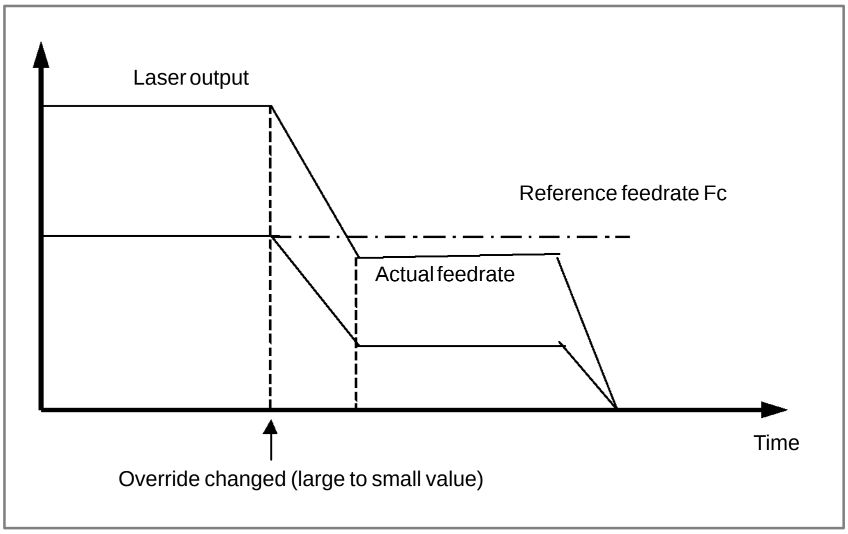
When G01 is used instead of G00 to perform positioning, beam output must be stopped during positioning operation.
When G01S0 or G01Q0 is specified because G01 is used for positioning in the power control mode, the power and pulse duty cycle are clamped at the minimum peak power (parameter No. 15090) and minimum pulse duty cycle (parameter No. 15092) of power control, so the beam cannot sometimes be stopped.
When bit 4 (SQ0) of parameter No. 15096 is set to 1, power control is stopped if either the specified peak power (parameter No. 15040) or the specified pulse duty cycle (parameter No. 15042) is set to 0.
This can surely stop the beam also at the time of positioning.
When the beam is stopped using G01S0, however, the minimum output power (parameter No. 15211) must be set to 1.

• Reference feedrate when the feedrate override value has been changed

The reference feedrate (Fc in Fig. 18.3) is usually used as a specified feedrate when the power control mode is entered. By setting bit 5 of parameter No. 15096, a feedrate obtained by applying a feedrate override to a specified feedrate can be used as a reference feedrate.

1) When the specified feedrate when the power control mode is entered is the reference feedrate (bit 5 of parameter No. 15096 is set to 0)

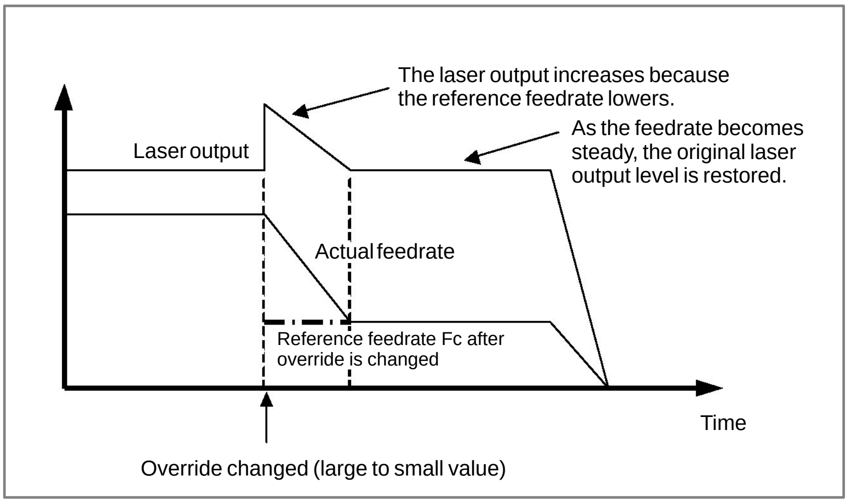
In this case, even when the feedrate override value is changed in the power control mode, the reference feedrate is kept unchanged, but the actual feedrate changes. So, the laser output changes according to the actual feedrate.



Actual feedrate and laser output (1)

2) When the reference feedrate is obtained by applying a feedrate override to a specified feedrate (bit 5 of parameter No. 15096 is set to 1)

In this case, as the override value is changed, the reference feedrate changes. Therefore, when the feedrate is steady, the laser output is kept unchanged; during acceleration/deceleration due to change in override, the laser output changes.
If the feedrate override value is decreased, the actual feedrate and laser output are changed as shown below.



Actual feedrate and laser output (2)

- **Reference feedrate when the feedrate is changed using an F command**

When an F command is specified in the power control mode, the reference feedrate is assumed as follows as in the case where the override is changed:

When bit 5 of parameter No. 15096 is set to 0
The reference feedrate is not changed.

When bit 5 of parameter No. 15096 is set to 1
The reference feedrate is changed to the value specified with address F.

(G code specification)

Sample

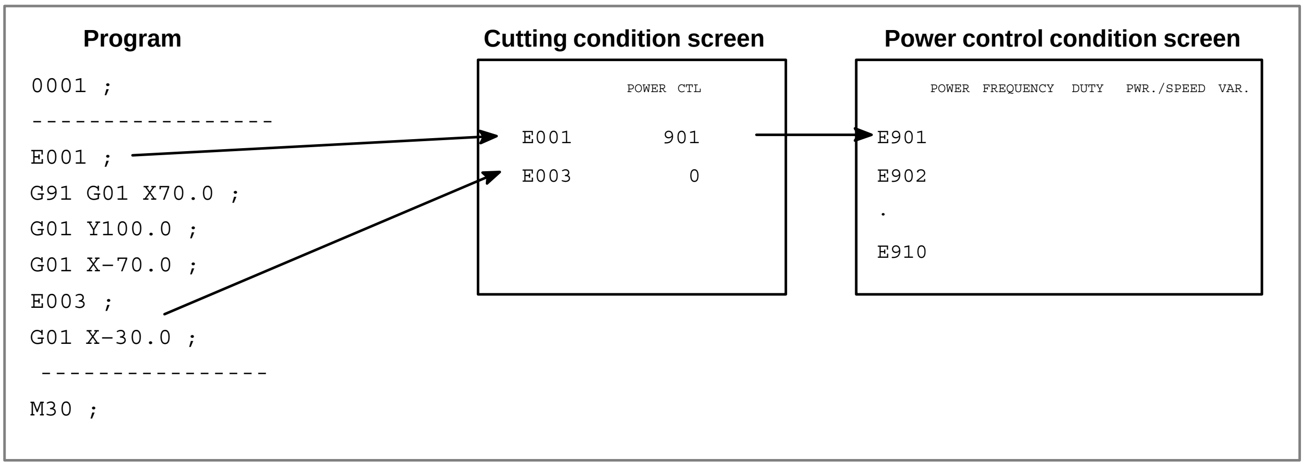
(If controlling the power specification, pulse frequency, and pulse duty)

```
O0001;
 . . . . . . . . . . . . . . .              Both Mo and Mmin are assumed to be 0.
G01  S1000 P1000 Q100 F1000;    *1
G63  P1;              Laser power control mode
G91  G01 X70.0;       Power: 1000 W, frequency: 1000 Hz, duty: 100%
G01  Y100.0 F500;     Power: 500 W, frequency: 500 Hz, duty: 50%
G01  X–70.0 F750;     Power: 750 W, frequency: 750 Hz, duty: 75%
G63  P0;              Laser power control mode cancel
G01  X30.0;           Power: 1000 W, frequency: 1000 Hz, duty: 100%
 . . . . . . . . . . . . . .
M30;
```

> ***1**
> Before issuing G63P1, you must specify a machining condition that corresponds to Mc shown in Fig. 18.3.

Specification using E codes
(When the power command, pulse frequency, and pulse duty cycle are controlled)

When an E code for cutting is specified in a program, the corresponding cutting conditions are searched for, and if the conditions include a power control condition setting, the power control mode is entered.
In the following example, the E001 command enters the power control mode, and the E003 command cancels the power control mode.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
15000			FLT					LPC

LPC

0 : Disables the power control function.
1 : Enables the power control function.

If 0 is set, power control mode cannot be set. The G code or input signal for entering power control mode is ignored.

FLT During power control, the first–order lag filter is:

0 : Disabled.
1 : Enabled.

When this parameter is set to 1, the first–order filter is applied to the output command. This delays the output command.
The time constant of the filter is set in parameter No. 15094.

	#7	#6	#5	#4	#3	#2	#1	#0
15089	LP8	LP7	LP6	LP5	LP4	LPZ	LPY	LPX

LP* Specifies whether to use the *–axis in feedrate calculation.

0 : Not use.
1 : Use.

The composite feedrate for the axes selected in this parameter is used to calculate each laser output factor. If no axis is specified, it is assumed that the first axis (LPX) and the second axis (LPY) are specified.

15090	Minimum power specification value

[Data unit] W

[Data range] 0 to 7000

Minimum clamp value for the power specification value that corresponds to Mmin. If the power specification value calculated from the current feedrate is less than the value set for this parameter, clamping is performed with this value.

15091	Minimum pulse frequency

[Data unit] Hz

[Data range] 5 to 2000

Minimum clamp value for the pulse frequency that corresponds to Mmin in Fig.18.3. If the frequency calculated from the current feedrate is less than the value set for this parameter, clamping is performed with this value.

15092	Minimum pulse duty

[Data unit] %

[Data range] 0 to 100

Minimum clamp value for the pulse duty that corresponds to Mmin in Fig.18.3. If the duty calculated from the current feedrate is less than the value set for this parameter, clamping is performed with this value.

15094	Power control filter time constant

[Unit of data] ms

[Valid data range] 8 to 32767

This parameter specifies a time constant for the primary delay filter used to delay the power command for synchronization with machine feed.
The primary delay filter is expressed as follows:

$$O_n = O_{n-1} + (I_n - O_{n-1})\frac{Ts}{Tc}$$

Ts: 8 ms (sampling time)

Tc: To be set in this parameter (time constant)

15095	Allowable variation in power control speed

[Unit of data] mm/min

[Valid data range] 0 to 255

The calculated feedrate slightly fluctuates also in a steady state, and the data items (specified power, pulse frequency, and pulse duty) calculated based on the feedrate also fluctuate. To prevent these unnecessary fluctuations, specify the width of a dead zone for feedrate fluctuations.

	#7	#6	#5	#4	#3	#2	#1	#0
15096	EGM	SDB	FOV	SQ0		PCD	PCF	PCP

PCP Specifies whether to control the power specification value with the feedrate.

0 : Does not control the value.
1 : Controls the value.

PCF Specifies whether to control the pulse frequency with the feedrate.

0 : Does not control the frequency.
1 : Controls the frequency.

PCD Specifies whether to control the pulse duty with the feedrate.

0 : Does not control the pulse duty.
1 : Controls the pulse duty.

SQ0 If 0 is set as either the specified peak power (S) or specified pulse duty cycle (Q), the power control function is:

0 : Performed.
1 : Stopped.

Whether the specified peak power or pulse duty cycle has become 0 or not is determined by the peak power setting in parameter No. 15040 and the pulse duty cycle setting in parameter No. 15042.
Therefore, even when these parameters are set to 0 by a method other than the specification of addresses S, P, and Q in a machining program, power control is stopped.

FOV When the feedrate override has been changed in the power control mode, the feedrate used as the reference feedrate for power control is:

0 : Not changed.
1 : The feedrate overridden by the new override value.

SDB To set the allowable variation width of power control feedrate (parameter No. 15095):

0 : Specify a feedrate (mm/min) directly.
1 : Set the percentage (0 to 100%) to the specified feedrate used when the system enters the power control mode.

EGM In edge machining mode and start–up machining mode:

0 : Power control mode is canceled.
1 : Power control is stopped only during the return distance movement in edge machining and start–up machining.

Edge machining mode and start–up machining mode refer to the states in which E numbers specifying the respective cutting conditions for edge machining and start–up machining have been selected.

15097	Parameter for calculating the rate of output change (output at a feedrate (F) of 0)

[Data unit] W

[Data range] 0 to 7000

Sets the power specification value corresponding to Mo, i.e., the value assumed when the feedrate (F) is equal to 0. The rate of output change is calculated with this value and the output value specified when power control mode is ON (Mc).

15098	Parameter for calculating the rate of frequency change (frequency at a feedrate (F) of 0)

[Data unit] Hz

[Data range] 5 to 2000

Sets the frequency corresponding to Mo, i.e., the frequency assumed when the feedrate (F) is equal to 0. The rate of output change is calculated with this value and the frequency specified when power control mode is ON (Mc).

15099	Parameter for calculating the rate of duty change (output at a feedrate (F) of 0)

[Data unit] %

[Data range] 0 to 100

Sets the duty corresponding to Mo, i.e., the duty assumed when the feedrate (F) is equal to 0. The rate of duty change is calculated with this value and the duty specified when power control mode is ON (Mc).

Limitations

- **High–precision contour control mode**

If the laser power control function is to be used in high–precision contour control mode (HPCC mode), enable laser power control mode before setting high–precision contour control mode. Similarly, cancel high–precision contour control mode before canceling laser power control mode.

- **Pulse enhanced function**

The laser power control function cannot be used together with the pulse enhanced function. The pulse enhanced function is not effective in power control mode.

- **Use on a rotation axis**

The laser power control function can be used on a linear axis only. It cannot, therefore, be used together with any function that causes rotation about a rotation axis, such as the cylindrical interpolation function.

- **Three–dimensional machining functions**

The laser power control function cannot be used with the three–dimensional machining functions.

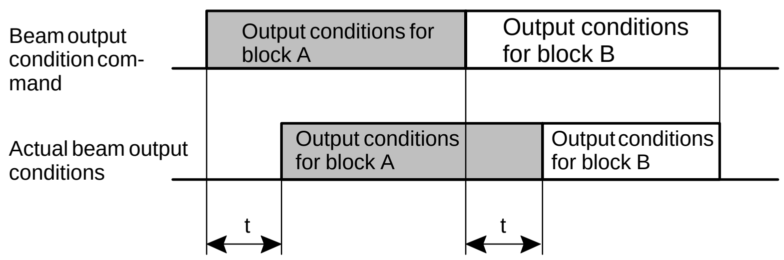
18.4 BEAM OUTPUT CONDITION DELAY FUNCTION

If the laser beam output conditions are to be changed when there are consecutive machining blocks, the output conditions are changed to those for the next block at the deceleration position of the current block. The larger the feedrate, the larger the difference between the block gaps on the programmed beam path and the positions at which the output conditions are actually changed. The timing at which the beam output conditions are to be changed can be delayed in order to compensate for this difference and turn the beam ON/OFF at the optimum positions.

Explanations

- **Control method**

For a given block, a beam control command is issued to the laser oscillator in sync with a move command to the servo. The beam output condition delay function delays the issue of the beam control command to the laser oscillator so that it is issued after the move command to the servo. The output condition delay is set with a parameter. The delay can be 0 to 64 ms in 8 ms units.
If the beam is OFF (the next block is not a machining block), the output conditions are not delayed.



t: Beam output condition delay (parameter)

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
15005	DLY							

DLY

0 : Disables the beam output condition delay function.
1 : Enables the beam output condition delay function.

15219	Beam output condition delay

[Data unit] 8 ms

[Data range] 0 to 8
The change of beam output conditions is delayed by the value set for this parameter.

Limitations

- **High–precision contour control function**

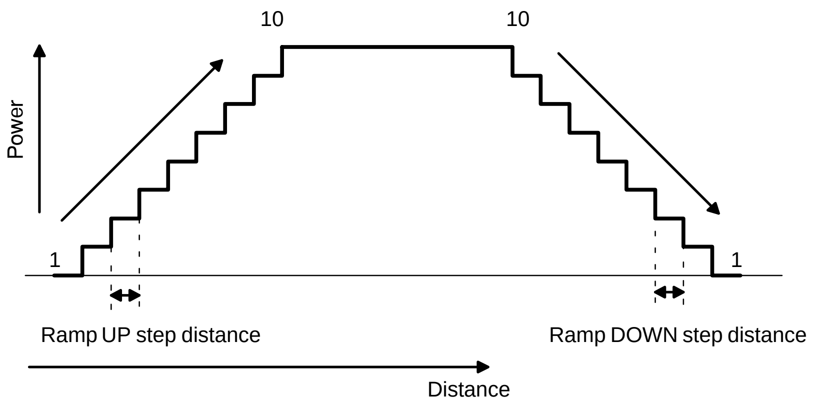
A dedicated parameter (PRM.15410) is provided so that another delay time can be set in high–precision contour control mode (HPCC mode).

- **Using this function together with other functions**

1) Of the functions controlling the output conditions, the following cannot be used together with the beam output condition delay function. (If they are used, the beam output condition delay function is disabled.)
   - Piercing and high–speed piercing functions
   - Edge machining function. (The function is assumed to be used if a machining number is specified.)
   - Startup machining function. (The function is assumed to be used if a machining number is specified.)
2) Beam output condition data using system variables in macros cannot be changed.
3) Beam output condition data using window functions cannot be changed.
4) Beam output condition data using G10 (programmable parameter input) cannot be changed.
5) During automatic operation, beam output condition data cannot be changed on the laser setup screen.

18.5 STEP FUNCTION

The step function controls the laser power in steps for a set distance, starting from the weld start point and for another set distance ending at the weld end point when welding is performed with laser beam machining, to achieve good welding results. The figure below shows an outline of the ramp UP/DOWN distances and power control. The laser power is changed in ten steps over each of the set distances.



The ramp UP step distance multiplied by 10 is called the ramp UP distance, while the ramp DOWN step distance multiplied by 10 is called the ramp DOWN distance.

Explanations

- **Setting the machining conditions**

The data area provided for the step function is capable of storing five data item sets, each set consisting of the data items listed below. E301 to E305 are allocated as the E numbers for specifying the individual data items.

a) Ramp UP step distance	b) Ramp DOWN step distance
Step power 1	Step power 1
Step power 2	Step power 2
Step power 3	Step power 3
Step power 4	Step power 4
Step power 5	Step power 5
Step power 6	Step power 6
Step power 7	Step power 7
Step power 8	Step power 8
Step power 9	Step power 9
Step power 10	Step power 10

One set consists of the items listed under both a) and b).

		Step distance mm	Step power 1	2	3	4	5	6	7	8	9	10
301	Up	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
301	Down	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
302	Up	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
302	Down	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
303	Up	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
303	Down	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
304	Up	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
304	Down	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
305	Up	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000
305	Down	00.000	0000	0000	0000	0000	0000	0000	0000	0000	0000	0000

- **Valid data ranges**

The valid data ranges are as follows:
Ramp UP/DOWN step distance : 0 to 65.000 mm
Step power : 0 to 7000 W
Ramp UP/DOWN operation is performed by measuring the distance of the straight line from the machining start point to the machining end point in a single cutting block.

- **Number of steps**

The number of steps in which the laser power is changed in step control is 10. This number is fixed.

- **Writing to and reading from the data area**

On the CNC, the data area for step control is reserved, but the screen for setting the machining conditions is not provided. Access the data area using C language function of a PMC.

- **Step control mode**

In a machining program, step control mode is set by specifying command E301 to E305. Only the laser power is changed in step control mode; the other machining conditions set with the previously specified E1 to E10 commands remain as is.

- **Step distances**

The ramp UP and DOWN distances cannot extend over multiple blocks.

- **Power override**

Power override is effective for the laser power output in step control.

- **Operation on arcs**

For an arc, step control is possible if the distance of the straight line from the start and end points is longer than the sum of the ramp UP and DOWN distances. A full circle, on which the start and end points are the same, does not satisfy this condition and, therefore, step control is not possible. If, however, the circle is divided in a machining program, step control can be performed on each part of the full circle.

- **Alarm**

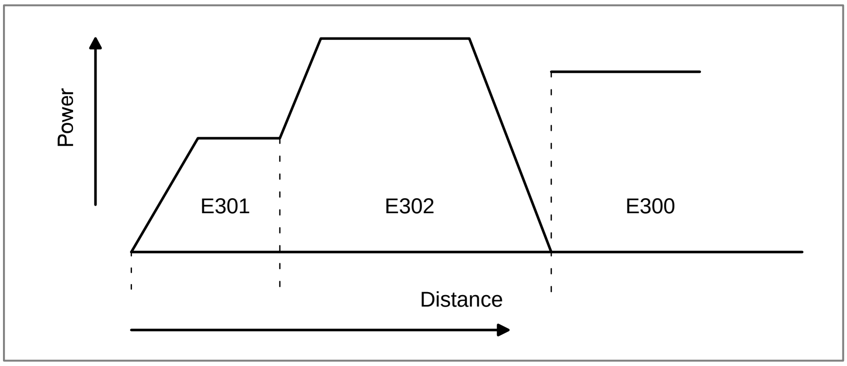
If, in the machining block subject to step control, the remaining amount of movement becomes shorter than the DOWN distance before the movement along the ramp UP distance is completed, an alarm is issued. An alarm is also issued if movement along the ramp UP distance is not completed in the single block.

Alarm No.	Message (Japanese/English)
4131	Invalid step distance/STEP LENGTH

Examples

The following is a machining program example:

```
    O0010;
    G92G90X0Y0;
(1) E1;        . . . Cutting/machining condition definition
(2) E301;      . . . Step control mode & condition definition
(3) G01X100;
(4) E302;      . . . Step control condition definition
(5) G01X300.:
(6) E300;      . . . Step control mode cancel
    G01X400.;
    M30;
```



1) In block (1), the feedrate, assist gas pressure, laser output conditions, etc., are defined.
2) In block (2), the machining conditions in step control mode are defined.
3) In block (4), the machining conditions in step control mode are changed, and machining continues.
4) In block (6), step control mode is canceled and normal cutting feed machining is executed.

In the machining program example, if the conditions in step control are not changed and step control is executed with the same E number (300 to 399) throughout the program, the following is performed automatically:

I) If the subsequent block is a cutting block, ramp DOWN operation is not performed.

II) If the preceding block is a cutting block, ramp UP operation is not performed.

If there are consecutive cutting blocks and the E number changes after each block, as in the program example, step control is executed according to the set values regardless of whether the preceding or subsequent block is a cutting block.

In step control mode, only the laser power is changed according to the set distances; other conditions such as the frequency, duty, and feedrate are not changed. The data effective at that time is inherited. After the 10th step, where ramp UP operation ends, the laser power is held constant. That is, the laser power assumed at the 10th step remains effective until ramp DOWN operation starts. If, during ramp UP or ramp DOWN operation, the amount of movement in one interpolation period in the CNC is larger than the distance for one step, the power may not be changed accurately in ten steps. Should this occur, adjustment is required with the ramp distance per step, feedrate, time constant, or bit 7 of parameter No. 15008.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
15008	RMP							

RMP In the step function,

0 : Measures the movement distance with the UP/DOWN step distances only.

1 : Measures the movement distance with the UP/DOWN step and the specified feed rate.

15730	Distance for one step during ramp UP operation

[Data format] Word

[Data unit] 1/1000 mm

[Data range] 0 to 65000

Sets the distance for one step during ramp UP operation in step control.

15731	Ramp UP step power 1
15732	Ramp UP step power 2
15733	Ramp UP step power 3
15734	Ramp UP step power 4
15735	Ramp UP step power 5
15736	Ramp UP step power 6
15737	Ramp UP step power 7
15738	Ramp UP step power 8
15739	Ramp UP step power 9
15740	Ramp UP step power 10

[Data format] Word

[Data unit] W

[Data range] 0 to 7000

Sets the laser power for each step during ramp UP operation in step control.

15745	Distance for one step during ramp DOWN operation

[Data format] Word

[Data unit] 1/1000 mm

[Data range] 0 to 65000

Sets the distance for one step during ramp DOWN operation in step control.

15746	Ramp DOWN step power 1
15747	Ramp DOWN step power 2
15748	Ramp DOWN step power 3
15749	Ramp DOWN step power 4
15750	Ramp DOWN step power 5
15751	Ramp DOWN step power 6
15752	Ramp DOWN step power 7
15753	Ramp DOWN step power 8
15754	Ramp DOWN step power 9
15755	Ramp DOWN step power 10

[Data format] Word

[Data unit] W

[Data range] 0 to 7000

Sets the laser power for each step during ramp UP operation in step control.

Limitations

- This function is an option function, and requires the following separate functions:
  1) Option function, ”machining condition setting function”
  2) Option function, ”expansion retention type memory add function”
- This function can be used to its full advantage when used with the FANUC C series laser.
- This function cannot be used with the automatic corner override function, edge machining function, and startup machining function. The step function takes precedence.
- This function is not effective when the single block, dry run, or machine lock function is used.
- This function can be used together only with X–Y plane facing.
  It cannot be used together with any other function (such as cylindrical interpolation, normal–direction control, or three–dimensional cutting).
- The sample programs, machining conditions, and other information described in this manual do not represent actual machining conditions.
- The information described in this manual is subject to change without notice.

18.6 ASSIST GAS CONTROL (G32)

The assist gas command (G32) performs assist gas control. Two methods of assist gas control are supported: flow pattern specification and direct gas pressure control specification.
G32 is specified to output, switch, or stop the assist gas.

18.6.1 Flow Pattern Specification

With this method, an assist gas output condition is set for each of three stages: before machining (pre–flow), during machining (machining flow), and after machining (post–flow). Then, the number assigned to a set of these conditions is specified.

Format

G32 P_ Q_;

P_: Assist gas type
Q_: Gas flow pattern

Explanations

- **Assist gas type**

Specify the number assigned to the type of assist gas to be output.
For the correspondence between the assist gas numbers and types of assist gas actually output, refer to the operator's manual provided by the machine tool builder.
When P0 is specified, the assist gas output is stopped.

- **Flow pattern**

Specify the number assigned to the desired assist gas flow pattern to be output.
As a flow pattern, specify the assist gas pressures and output periods applicable before and after machining, and the assist gas pressure applicable during machining on the laser setting screen.

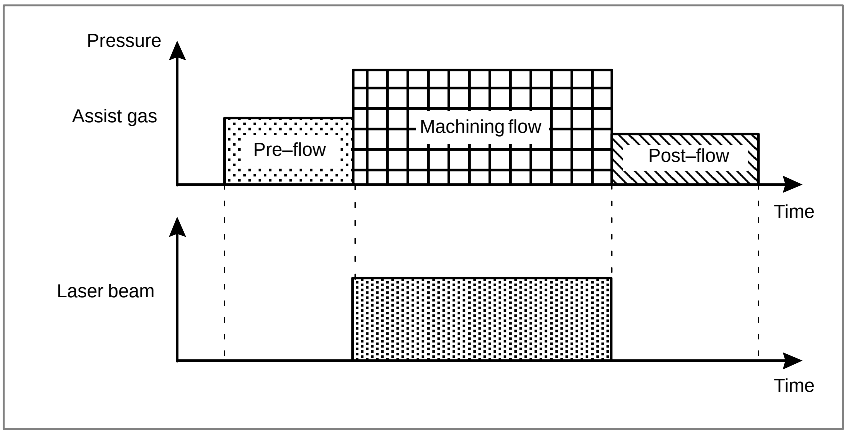


Table 18.6.1 Specifiable Value Range

Command	Specifiable value range
P_	0 to 3, 0 to 7
Q_	0 to 3

* The user can choose between a set of three types of assist gas and a set of seven types of assist gas by setting bit 0 of parameter No. 15001.

- **Shutter control**

When assist gas is output with G32, the shutter is opened. When the output of assist gas is stopped with G32P0, the shutter is closed.

Limitations

- **Switching between flow pattern specification and direct gas pressure control specification**

Whether to use flow pattern specification or direct gas pressure control specification is determined by the specification following G32. However, the method to be used may be ambiguous, depending on the specification. In such a case, the method to be used is determined by the setting of bit 0 of parameter No. 15004.

18.6.2 Direct Assist Gas Control Specification

With this method, assist gas is output without changing the pressure, and machining is started after assist gas has been output for a certain period (gas pressure settling time). When the machining is stopped, the output of the assist gas is also stopped.

Format

G32 P_ T_ R_;

P_: Assist gas type
T_: Assist gas pressure
R_: Assist gas pressure settling time

Explanations

- **Specify the type of assist gas to be output.**

For the correspondence between the assist gas numbers and the types of assist gas that are actually output, refer to the operator's manual provided by the machine tool builder.
When P0 is specified, the output of the assist gas is stopped.

- **Assist gas pressure**

Specify a desired assist gas output pressure.

- **Assist gas settling time**

Specify a settling time for the assist gas. The settling time is the period from the assist gas first being output until a specified assist gas pressure is reached. Machining is started once this settling time has elapsed.

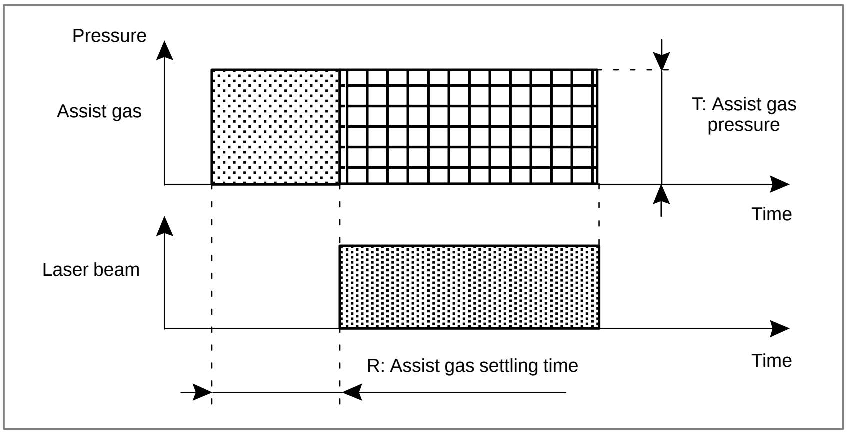


Table 18.6.2 Specifiable Value Range

Command	Specifiable value range	Specifiable value range
P_	0 to 3, 0 to 7	
T_	0 to 255	0.01MPa
R_	0 to 32767	10msec

* The user can choose a set of three types of assist gas or a set of seven types of assist gas by setting bit 0 of parameter No. 15001.

- **Shutter control**

When assist gas is output with G32, the shutter is opened. When the output of assist gas is stopped with G32P0, the shutter is closed.

Limitations

- **Machining condition setting function**

With direct gas pressure control specification, a value specified in the machining program is stored in a parameter, and assist gas is controlled by the parameter value. For assist gas control, a parameter for machining and a parameter for piercing are supported. When the machining condition setting function is not used, only a parameter for machining is used.

- **Switching between flow pattern specification and direct gas pressure control specification**

Whether to use flow pattern specification or direct gas pressure control specification is determined by the specification following G32. However, the method to be used may be ambiguous, depending on the specification. In such a case, the method to be used is determined by the setting of bit 0 of parameter No. 15004.

18.6.3 Direct Gas Pressure Control Specification Using the Machining Condition Setting Function

When the machining condition setting function is used, assist gas specification data for piercing and machining can be registered beforehand. Then, a desired type of assist gas operation can be specified for execution in address L after G32.Format

Format

```
E_;       Machining condition selection
G32 L_;   Assist gas specification

     E_: Machining condition
     L_: Assist gas data
```

Explanations

- **Machining condition**

Select the machining conditions registered with the machining condition setting function. Up to ten conditions for machining and up to three conditions for piercing can be registered. Select one condition for machining and one condition for piercing.

- **Assist gas data**

Select the stop of assist gas, the output of assist gas for machining, or the output of assist gas for piercing.

L0 : Stops assist gas.
L1 : Outputs assist gas for machining.
L2 : Outputs assist gas for piercing.

Table 18.6.3 Specifiable Value Range

Command	Specifiable value range
E_	1 to 10 (for machining), 101 to 103 (for piercing)
L_	0 to 2

- **Shutter control**

When assist gas is output with G32, the shutter is opened. When the output of assist gas is stopped with G32P0, the shutter is closed.

Limitations

- **Machining condition specification**

When assist gas is to be output with G32L_;, specify a desired machining condition with E code before specifying G32L_;.

18.7 MACHINING CONDITION SETTING FUNCTION (E CODE)

Data consisting of a set of items required for laser machining can be numbered and registered in a data area. Then, when a program specifies a data number, the corresponding data is read to perform laser machining. This data area has an enough capacity for holding data required for machining one type of workpiece (material and plate thickness), so conditions required for machining can be set just by specifying a data number. High–speed access to this area is possible, so data can be modified manually and easily even during program execution. Furthermore, modified data can be registered, so machining conditions can be updated easily.

When the option for additional cutting condition registration is specified, the specifications of the conventional machining condition setting function can be expanded.

For details of the expanded specifications, see “OPERATION”.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
15004		ECH						

[Data type] Bit

ECH Upon reset, the active E number is:
0 : Not cleared.
1 : Cleared.

	#7	#6	#5	#4	#3	#2	#1	#0
15008					SOC			

[Data type] Bit

SOC When the machining condition setting function is used, the reference displacement amount of gap control is used as the reference displacement amount of piercing:
0 : Except during machining.
1 : Only during piercing.

15350	M code for reading comments

[Data type] Word

[Unit of data]

[Valid data range] 1 to 999

Set the code number of an auxiliary function for reading comments.

15360	Start address of R area for data for modification

[Data type] Word

[Unit of data]

[Valid data range] 1 to 997

When an address in the R area of the PMC is set, data for modification is assigned with the set address regarded as the start address.
When the modification function is not to be used, set 0.

18.7.1 Piercing Command

By specifying an E code together with G24, the conditions required for machining can be set.

Format

- **When an E code is combined with the G code**
  G24 E__;
- **When an E code is specified alone**
  E__;
  •
  •
  •
  G24;
- **E_: Piercing data set number (101 to 103, 101 to 110)**

Explanations

When G24 is specified, the CNC calls the set of data corresponding to the E code, and piercing is performed according to the data. An E code can also be specified alone.
Once an E code has been specified, G24 not followed by an additional E code performs processing under the same machining conditions.

18.7.2 Cutting Command

By specifying an E code together with a cutting feed command, the conditions required for machining can be set.

Format

- **When an E code is combined with a G code**
  G01 (G02, G03, G12) E__;
- **When an E code is specified alone**
  E__;
  •
  •
  •
  G01 X__Y__;
- **E_: Machining data set number (1 to 10, 1 to 30)**

Explanations

When a machining feed command is executed, the CNC calls the set of data corresponding to the E code, and machining is performed according to the data. An E code can also be specified alone.
Once an E code has been specified, a machining feed command not followed by an additional E code performs processing under the same machining conditions.

18.7.3 Reading a Comment

An M code for transferring a comment is provided.
This code can be set in parameter No. 15350.
Use of this M code allows a comment consisting of up to 24 alphanumeric characters to be posted to the PMC via the window function.
In a comment, the material of a workpiece can be specified.

```
Mxxx (*************);
         └── Within 24 characters including alphabetical
             characters, numeric characters, a decimal point, and
             a sign.
```

where, xxx represents the value set in parameter No. 15350.

Sample programs

```
G92X_Y_Z; :  Sets a coordinate system.
E1;       :  Specifies a machining data set number.
E101;     :  Specifies a piercing data set number.
M200; [Macro call]
   G32L2; :  Opens the assist gas start shutter according to the data of
             E101.
   G24;   :  Performs piercing according to the data of E101.
   G32L1; ;  Switches the assist gas according to the data of E001.
G01X_Y_;  :  Executes start–up machining if an auxiliary data set
             number is set in the item of start–up machining selection
             of E001.
X_Y_;     :  Performs cutting according to the data set of E001.
 •
 •
 •
 •
M201; [Macro call]
   G32L0; :  Stops the output of assist gas, and closes the shutter.
M30;      :  Ends the program.
```

18.7.4 Edge Machining Function

The edge machining function is used to sharpen an edge of a workpiece. This function consists of edge detection, gradual stop control, piercing, and control over the power and feedrate used in the transition from piercing to machining. This function is useful for an edge that is smaller than a specified angle.

Enabling/disabling the edge machining function

When an edge data number is set in the edge selection item of the machining data group on the machining condition setting screen, the edge machining function is enabled. This means that edge machining is applied not to the nozzle path on the program but to the offset nozzle path.
To disable the edge machining function, set 0 in the edge selection item.

Screen

```
No. FEED    PWR FREQ DUTY   GAS   GAS  GAS   DEF        EDGE     APPR POWER CTL

 1 *****  **** **** ***    **.*   **   *.*  -*.*** -*.*** 201  ***     ***
 2 *****  **** **** ***    **.*   **   *.*  -*.*** -*.***   0  ***     ***
 3 *****  **** **** ***    **.*   **   *.*  -*.*** -*.*** 205  ***     ***
 4 *****  **** **** ***    **.*   **   *.*  -*.*** -*.*** 203  ***     ***
 5 *****  **** **** ***    **.*   **   *.*  -*.*** -*.*** 204  ***     ***
```

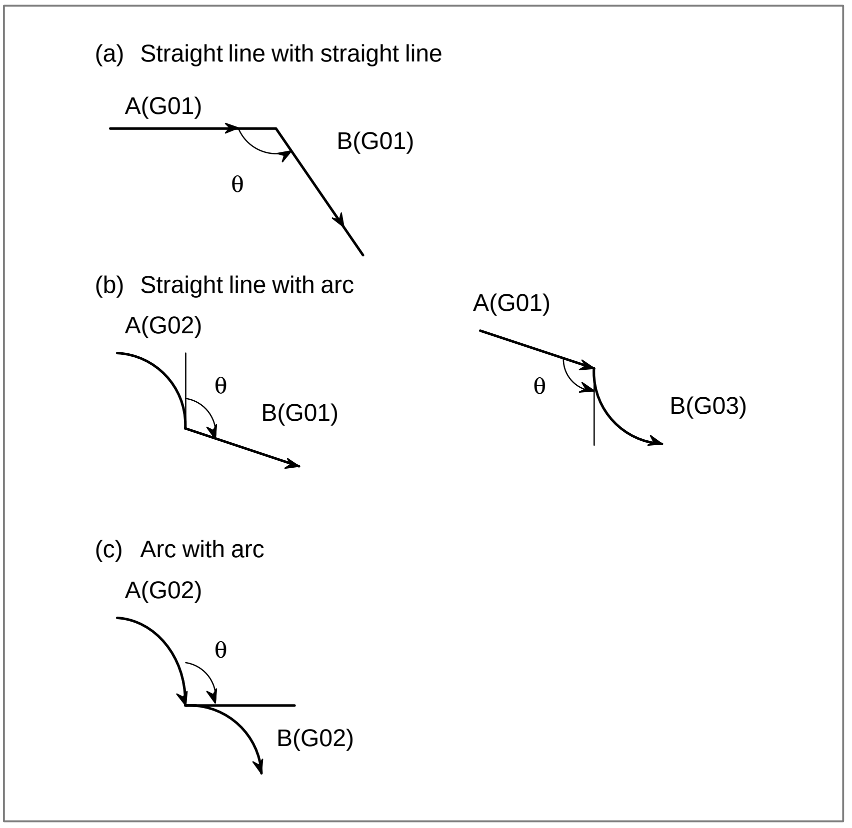
If a machining data set number is specified in address E in the machining program, the edge selection item in the specified machining data is referenced when the program is executed. Then, the edge data set number is read to perform edge machining.

- For an explanation of data setting, see Section III–15.9.

Explanations

• Edge determination

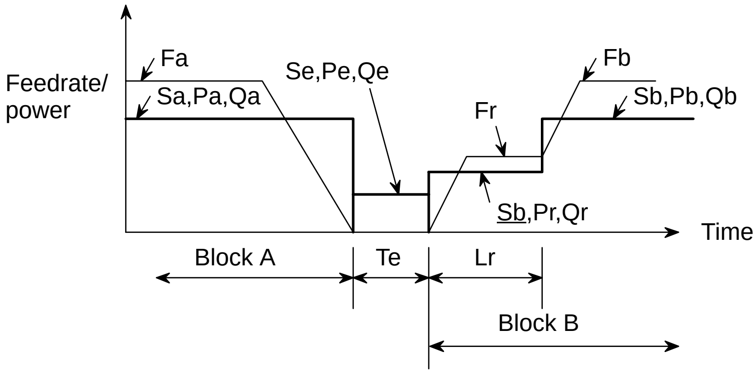
In G64 mode (machining feed), the angle θ of the corner formed by two machining feed blocks (A and B) is calculated.



If a calculated angle is smaller than a set value, a corner is assumed; a gradual stop occurs at the end point of block A, and feedrate and power control is applied.

- **Feedrate and power control**

At the transition from block A to block B, machining is performed as shown below.



Fa	:	Feedrate of block A	Data of block A
Sa	:	Output peak value of block A	
Pa	:	Pulse frequency of block A	
Qa	:	Pulse duty cycle of block A	
Se	:	Piercing output peak value	Edge machining data
Pe	:	Piercing pulse frequency	
Qe	:	Piercing pulse duty cycle	
Te	:	Piercing time	
Lr	:	Return distance	
Fr	:	Return feedrate	
Pr	:	Return frequency	
Qr	:	Return duty cycle	
Fb	:	Feedrate of block B	Data of block B
Sb	:	Output peak value of block B	
Pb	:	Pulse frequency of block B	
Qb	:	Pulse duty cycle of block B	

(1) A gradual stop occurs at the end point of block A.

(2) The output is held until a gradual stop occurs.

(3) Piercing is performed at the top of the corner.

(4) Upon the completion of piercing, a block B movement starts.

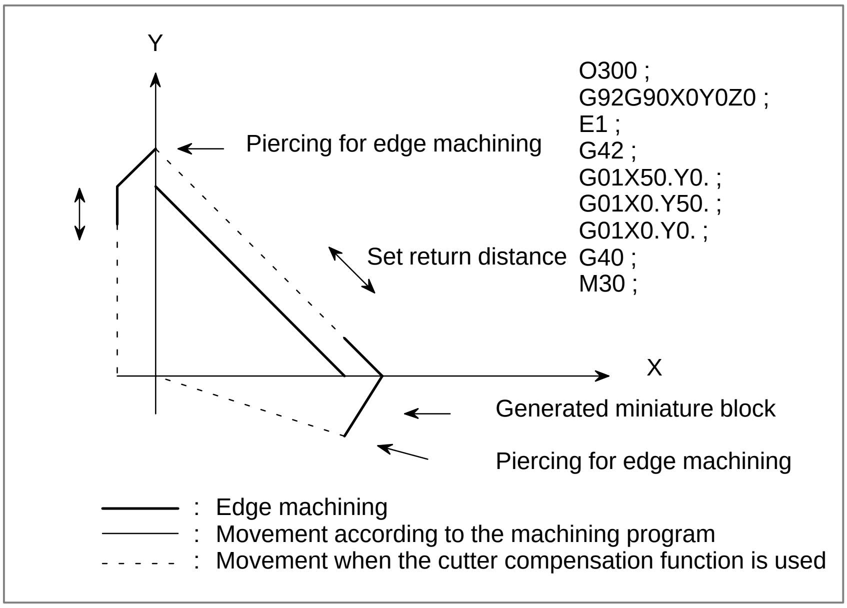
(5) Before switching to the machining condition of block B, the feedrate is changed to the return feedrate until the amount of travel equals the return distance. Then, the output peak value, return frequency, and return duty cycle for block B cutting are set for startup.

(6) When the return distance is moved, machining is continued by switching to the machining condition of block B.

> **⚠ CAUTION**
>
> 1 As the return distance, a value exceeding the length for one block cannot be specified.
>
> 2 Return peak value Sb' can be set in parameter No. 15059. It is set when as output peak value Sb' over distance Lr, a value different from peak value Sb for block B is to be set. When this value is set to 0, machining is performed on the assumption that Sb' equals Sb.

- **Cutter compensation function and edge machining operation**

When the cutter compensation function is used in edge machining mode, a miniature block that does not exist in the machining program may be generated, depending on the mode (G41 or G42). The figure below shows how edge machining is performed at a corner where a miniature block occurs.



(1) Piercing for edge machining is performed before a miniature block.

(2) In a miniature block generated for cutter compensation, cutting feed is executed according to the set return feedrate, return power, and return duty cycle.

(3) Until cutting feed is performed for the set return distance after the operation of (2), the return feedrate, return frequency, and return duty cycle are valid.

(4) After cutting feed is performed for the return distance, the normal feedrate and beam output condition are resumed.

This means that the distance where the return feedrate, return power, return frequency, and return duty cycle are valid becomes longer, so that a miniature block is generated.
When a miniature block is generated by cutter compensation, whether to perform edge machining is selected by determining the angle of a corner not by calculating the angle between the current block and the next in the machining program, but by calculating the angle between the current block and the miniature block.
When the cutter compensation function is used, set an operation angle, considering not only the processing performed by the machining program but also a generated miniature block.
If you want to execute edge cutting on a path specified in the machining program during cutter compensation, set bit 4 of parameter No. 15012 to 1.

- **Combination with conventional commands**

In edge machining also, conventional commands (such as S, P, Q, and F) can be used with a machining program.

- **Assist gas in edge machining**

When assist gas switching is performed for piercing in edge machining, whether to output a beam for the assist gas settling time can be specified by setting bit 4 of parameter No. 15004.

When assist gas switching is not to be performed for edge machining, set the assist gas selection value to 0 or the same value as for machining. If 0 is set as the assist gas selection value for edge machining, the assist gas selection value for machining is used.

In assist gas switching performed at the start of piercing in edge machining, at the start of the subsequent return distance cutting, and at the start of startup machining, when the gas type and gas pressure defined with a specified edge selection number and startup selection number are the same as those for cutting, set bit 3 of parameter No. 15011 to 1 if the set assist gas settling time is to be ignored.

When the assist gas output conditions defined with a specified edge selection number or a startup selection number are to be used as the assist gas output conditions for return distance cutting in edge machining and startup machining, set bit 0 of parameter No. 15011 to 1.

If bit 0 of parameter No. 15011 and bit 3 of parameter No. 15011 are set at the same time, the setting of bit 0 of parameter No. 15011 is assumed. The setting of bit 3 is ignored.

1) Parameter
   When assist gas is switched for edge machining, whether to stop beam output during the assist gas settling time can be specified.

	#7	#6	#5	#4	#3	#2	#1	#0
15004				EDG				

EDG

0 : Stops beam output when assist gas is switched for edge machining.
1 : Does not stop beam output when assist gas is switched for edge machining.

2) Nonselection of assist gas
   When assist gas selection is not required for edge machining, specify 0 or the same value as that used for machining for the assist gas selection on the edge machining condition setting screen.
   With this setting, the execution of a settling time for assist gas switching can be ignored.

- **Override**

The normal override function can be used for the following settings used to perform edge machining:

- Return feedrate
- Return frequency
- Return duty cycle

- **Gap control**

As the reference displacement value and gap value for gap control in edge machining, values set in the edge machining conditions are used. If a set value is 0, the value set for machining becomes effective.

Other settings related to cutting

If portions at or around a corner that are left uncut in edge cutting, setting bit 2 of parameter No. 15007 or bit 7 of parameter No. 15012 to 1 may improve this phenomenon.
If you want to perform edge cutting even in the exact stop mode, set bit 4 of parameter No. 15007 to 1.
If you want to enable the edge cutting function to be used together with the arc radius–based feedrate clamp function, set bit 7 of parameter No. 15010 to 1.

Limitations

(1) The machining condition setting function option must be prepared.

(2) When this function is used, the automatic corner override function cannot be used.

(3) This function and automatic corner deceleration cannot be used at the same time.

(4) In single block and dry run operations, this function does not work.

(5) This function is valid only for facing. This function cannot be specified in circular interpolation, normal direction control, and three–dimensional machining commands.

Signal

• Constant optical path length control start signal STUP<F223#6>

[Classification] Output signal

[Function] This signal indicates that machining is being performed for the return distance.

[Operation] This signal is set to 1 when machining is being performed for the return distance in edge machining and startup machining. After machining for the return distance is completed, the signal is set to 0.

Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
F223		STUP						

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
15004				EDG				

[Data type] Bit

EDG When the assist gas is switched in edge machining, the beam is:
0 : Turned off.
1 : Not turned off.

	#7	#6	#5	#4	#3	#2	#1	#0
15007				XSC	ECK	ESE		

[Data type] Bit

ESE If piercing is to be executed in edge machining, it is:
0 : Executed upon the completion of distribution.
1 : Executed after a smoothing error check is performed upon the completion of distribution.

ECK In edge machining, the angle is judged with:
0 : The actual machining path.
1 : The path in the machining program.

> ⚠ **CAUTION**
> This setting is ignored if bit 4 of parameter No. 15012 is set to 1.

XSC In exact stop mode, the edge machining function is:
0 : Not executed.
1 : Executed.

	#7	#6	#5	#4	#3	#2	#1	#0
15010	OVE							

[Data type] Bit

OVE Edge machining and feedrate clamp by arc radius:
0 : Cannot be used at the same time.
1 : Can be used at the same time.

> ⚠ **CAUTION**
> Set this parameter to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
15011			CSC		LVE			EDS

[Data type] Bit

EDS For machining for the return distance in edge machining and startup machining, the cutting conditions are assumed as follows:
0 : The cutting conditions are set as conventional.
1 : As the laser power, assist gas type, and assist gas pressure, the conditions for piercing in edge machining are used.

> ⚠ **CAUTION**
> This setting takes priority over bit 3 of parameter No. 1511.

LVE Assist gas switching at the start of edge machining and the start of the subsequent return distance cutting is performed as follows:
0 : The conventional specifications are observed.
1 : The settling time is ignored if the gas type and gas pressure remain unchanged.

CAUTION
This setting is ignored if bit 0 of parameter No. 15011 is set to 1.

CSC In the startup machining mode, when S, P, Q, or F is specified at the same time in the first G01, G02, or G03 block that appears after G24:
0 : Startup machining is performed in the conventional manner, regardless of the S, P, Q, or F command.
1 : The startup machining is canceled, and cutting is performed with the specified S, P, Q, or F value.

	#7	#6	#5	#4	#3	#2	#1	#0
15012	EXS			CVA				

[Data type] Bit

CVA The angle in the edge machining function is:
0 : Determined according to the setting of bit 3 of parameter No. 15007.
1 : Determined by the programmed path regardless of the setting of bit 3 of parameter No. 15007.

EXS When the edge machining function is executed, switching to the piercing conditions is performed:
0 : Upon completion of distribution (if bit 2 of parameter No. 15007 is set to 0), or at the end of acceleration/deceleration (if bit 2 of parameter No. 15007 is set to 1).
1 : After the in–position check for all axes has been made.

15059	return peak value

[Data type] Word

[Unit of data] W

[Valid data range] 0 to 7000

At the time of startup from a corner during edge machining, the return feedrate, return frequency, and return duty cycle are used for operation until movement is made over the distance set as the return distance, but the output peak value used for operation is a programmed value.
When a value is set as the return peak value, this setting is used for operation instead of the programmed value.

18.7.5 Startup Machining Function

The edge machining function determines the angle between two cutting blocks to perform machining at the start of the latter block according to the set machining conditions. On the other hand, when movement is made from G24 (piercing) to a cutting block, the startup machining function machines the set return distance according to the set machining conditions.

Startup machining mode selection

When selecting startup machining mode, set a desired number from 201 to 205 in the startup selection item on the machining data group screen of the machining condition setting screen in the same way as in edge machining mode selection. At this time, the same number can be set for both edge machining and startup machining. Moreover, only startup machining mode can be selected without selecting edge machining mode. When startup machining mode need not be selected, set 0.

Explanations

• Machining conditio

For startup machining, the machining conditions set on the edge machining condition setting screen are used, However, the data required for startup machining consists only of a return feedrate, return distance, return power, return frequency, and return duty cycle; the other setting data does not affect the startup machining.

• Combination with conventional commands

In startup machining mode also, conventional commands (such as S, P, Q, and F) can be used with a machining program.

Limitations

- To use this function, the optional machining condition setting function is required.
- This function cannot be used with the single block function or dry run function.
- This function can be used together only with X–Y plane facing.
  It cannot be used together with any other function (such as cylindrical interpolation, normal–direction control, or three–dimensional cutting).
- When this function is used, automatic corner override cannot be used.
- This function and automatic corner deceleration cannot be used at the same time.

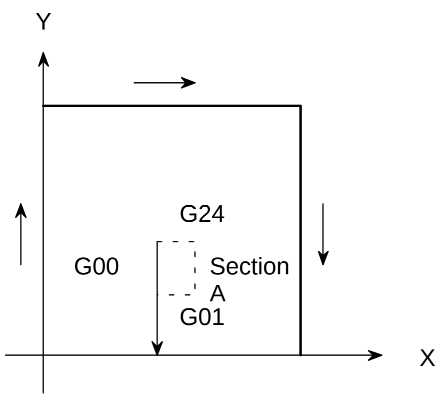
Examples

(1) After assist gas specification using G32, piercing is performed with G24.

(2) Upon the completion of piercing, the assist gas is switched to that used for machining, and cutting is started.

(3) At this time, in section A (set return distance), machining is performed according to the feedrate, power, frequency, and duty cycle set for startup machining.

(4) After machining is performed for the return distance, the normal feedrate and beam output condition are resumed as in the case of edge machining.




```
O301 ;
G92G90X0Y0Z0 ;
E1 ;
E101 ;
G00X50. Y50.
G32L2 ;
G24 ;
G32L1 ;
G01X50.Y. ;
G01X0.Y. ;
G01X0.Y100. ;
G01X100.Y100. ;
G01X100.Y0. ;
G01X50.Y0. ;
G32L0 ;
M30 ;
```

18.8 HIGH–SPEED LASER MACHINING FUNCTION

Outline

The high–speed laser machining function offers the following functions:

(1) RISC control
(2) Beam on/off control in the RISC mode
(3) Beam output condition delay control
(4) Power control function
(5) Parameter switch function for high–speed machining

This function can be used only for a two–dimensional command. If this function is used for a three–dimensional command, no alarm is generated, but this function does not operate normally.
This function is optional.

RISC control

To execute successive miniature blocks accurately at high speed, the following functions are implemented:

(1) Multi–block look–ahead acceleration/deceleration function before interpolation
(2) Automatic feedrate control function

For details, refer to the Operator's Manual (B–63534EN) provided with the FANUC Series 160*i*/160*i*–MB.

Beam on/off control in RISC mode

The beam output condition can be changed by specifying S, P, Q, and E in RISC mode.

(1) In the same way as in the normal mode, specify S (peak power), P (pulse frequency), Q (pulse duty cycle), and E (machining condition number) in a block specifying G01 (G02 or G03).
(2) An E code can be specified singly. The machine decelerates in the block.
(3) The Q command is used to turn the beam on or off in RISC mode. To turn the beam on, specify Q = 1 to 100. To turn the beam off, specify Q = 0.
(4) In RISC mode, the tracing control commands (G13, G14), assist gas control command (G32), piercing command (G24), and power control command (G63) cannot be specified. These commands must be specified before setting RISC mode or after canceling RISC mode.

Beam output delay control

To compensate for the delay between machine operation and beam on/off operation, beam output is delayed for a parameter–set delay from the start of the block.

(1) A delay of 0 to 64 ms, in units of 8 ms, can be set using a dedicated parameter.
(2) This function can be enabled or disabled by a parameter.
(3) This function is enabled in RISC mode.

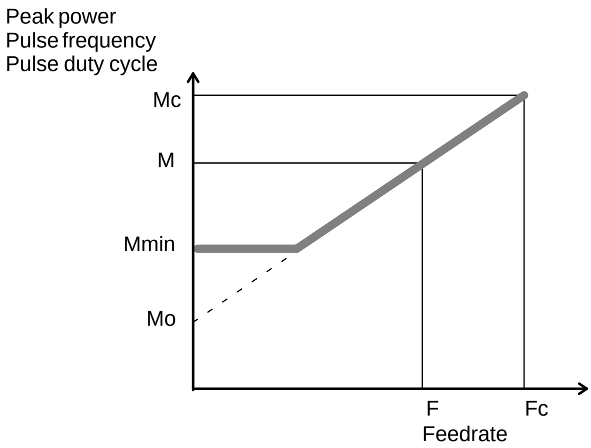
In beam output condition delay control mode, output condition modifications can be made only by using the S, P, Q, and E codes in the program. Output condition modifications cannot be made by using the methods below.

- Method that rewrites parameter settings with G10
- Method that rewrites specified output condition values with the PMC–CNC window
- Method that rewrites specified output condition values with system variables of the macro executor
- Method that rewrites specified output condition values on the laser setting screen

Laser power control function

According to the actual feedrate, the laser power control function controls each of the laser output conditions: peak power, pulse frequency, and pulse duty cycle.
When laser power control mode, described below, is set with a G code or external signal, the power specification, pulse frequency, and pulse duty cycle are controlled according to the following formula:

$$M = Mo + (Mc–Mo)\frac{F}{Fc}$$



Mo	:	Each data item (power, frequency, duty cycle) when feedrate F = 0 To be set by parameters
Mmin	:	Minimum clamp value for each data item (parameter) To be set by parameters
Mc	:	Each specified data item when laser power control mode is set To be set by programming
Fc	:	Specified feedrate when laser power control mode is set To be set by programming

• Power control mode

Laser power control mode can be set with a G code or external signal. When power control mode is set, the power control mode on signal (F224#1) is output.

<u>G code</u>
G63 P1; : Laser power control mode
G63 P0; : Laser power control mode cancel
<u>External signal</u>
G224#1 Power control mode signal

- **Enabling/disabling the function**

Whether to enable or disable the power control function can be set using the laser setting screen or a parameter.
<u>Setting using the laser setting screen</u>
By setting the power control item on the laser setting screen to 1, the power control function can be enabled.
<u>Setting using a parameter</u>
By setting bit 0 of parameter No. 15000 to 1, the power control function can be enabled.

- **Control item selection**

Whether peak power, pulse frequency, or the pulse duty cycle is to be controlled can be freely determined using parameter No. 15096. Multiple items can also be selected.

- **Axis subject to power control**

The power control function uses the feedrate of the nozzle to calculate the value of each laser output element (peak power, pulse frequency, and pulse duty cycle). To determine the feedrate accurately, the axes of the machine that are related to the operation of the nozzle are set using parameter No. 15089.
When this parameter is not set, the specification of the first and second axes is assumed.
A synchronous axis for simple synchronization control is related to the operation of the nozzle, but must be excluded from the setting because it is counted, together with the master axis, as one axis.

- **Dead zone for feedrate fluctuations**

Even in the steady state, the feedrate fluctuates very slightly. So, the value of each data item calculated from the feedrate fluctuates accordingly. To prevent this fluctuation, an allowable feedrate fluctuation width (dead zone) can be set using a parameter.
If the feedrate of a block being executed fluctuates within the value set with a parameter, the specified feedrate is assumed to be satisfied; the value of each data item is not changed.

- **Machine system delay compensation**

The power control function changes the value of each laser output data item in proportion to the specified feedrate. However, laser output changes earlier due to a delay in the machine system (servo system), so that a change in the machine system movement does not match a change in laser output. To compensate for this mismatch, a primary delay filter is used for the feedrate data of the machine system. Thus, a change in the laser output can be delayed accordingly. A primary delay filter is set using a parameter.

- **Laser power control function in RISC mode**

To use the laser power control function in RISC mode, enable power control mode before entering RISC mode. Similarly, before canceling power control mode, exit from RISC mode.

- **Selection of laser conditions to be controlled**

When the laser power control function is used, the laser output conditions to be controlled can be selected by parameters.

- **Combination with the pulse enhanced function**

The laser power control function cannot be used together with the pulse enhanced function. In power control mode, the pulse enhanced function is disabled.

Parameter switch function

Groups of parameters related to the high–speed high–precision contouring function based on RISC can be registered.
From the registered groups of parameters, the optimum group of parameters for a machining state can be selected.

- **Machining mode**

Three modes of laser machining are supported to satisfy different machining needs:

(1) Normal machining mode
This mode does not use the high–speed high–precision contouring function. The G13, G14, G24, G32, and G63 commands can be used.

(2) High–speed high–precision machining mode
This high–speed high–precision machining mode uses the RISC board. In this mode, the G13, G14, G24, G32, and G63 commands cannot be used; these commands must be used in other modes. Beam output can be stopped (for positioning, for example) by setting the laser output condition to 0.

(3) Continuous high–speed machining mode
This mode is basically the same as high–speed high–precision machining mode, but emphasis is placed on parameter setting for enabling a higher machining speed.

In each of these three modes, a different value is set in the parameter for specifying a machining speed. Even in the same mode, a different parameter setting is used when a different machining speed is required. So, parameter settings can be registered with the machining condition setting function to change the parameter settings to satisfy each mode and condition.

- **Parameter groups**

The parameters indicated in Table 18.8 are related to high–speed high–precision contouring mode.

Table 18.8 Parameters Related to High–Speed High–Precision Contouring Mode

Parameter No.	Name	Type	Description
2043	PK1V	Word–axis	Velocity loop integral gain
2048	BLCMP	Word–axis	Backlash acceleration value
2069	VFFLT	Word–axis	Feed–forward factor
2071	PBLCT	Word–axis	Backlash acceleration valid time
2082	BLEND	Word–axis	Backlash acceleration stop timing
2092	ADFF1	Word–axis	Look–ahead feed–forward factor
8451#0	USE	Bit	Enable/disable automatic feedrate control
8475#2	BIP	Bit	Enable/disable corner deceleration
8475#3	CIR	Bit	Enable/disable automatic feedrate control based on acceleration/deceleration during circular interpolation

A group of the parameters above can be registered as machining condition data.
Such a group of parameters can be called as required for operation.
Word–axis type parameters are supported for two axes: Z–axis and Y–axis.
In simple synchronization control, when the master axis is the X–axis or Y–axis, the same data as that for the master axis is automatically set for the slave axis.

- **Specification**

(1) An E code is specified to select a parameter. When an E code is specified, it is stored as an active E code.

(2) An E code from E501 to E506 is used.

(3) When a command for setting RISC mode is executed, parameter switching is performed. So, specify an E code immediately before a command for setting RISC mode.

(4) For parameter switching, an active E code is used. So, if an E code is not specified before a command for setting RISC mode, the number remaining in the active E code is used for operation.

(5) When an E code is specified in RISC mode, the active E code is updated, but parameter switching is not performed. To switch the parameter, first cancel RISC mode, then specify RISC mode again.

Signal

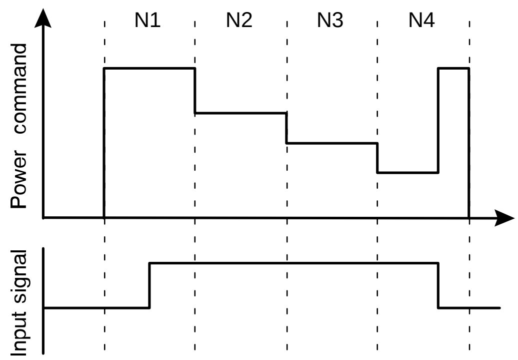
- **Power control mode signal PWCTL <G224#1>**

[Classification] Input signal

[Function] Specifies transition to power control mode.

[Operation] Transition to and return from power control mode are performed as described below.
Transition = (G63P1) OR (power control mode signal on)
When G63P1 is specified in the machining program, or while the power control mode signal is set to 1 by the PMC, power control mode is set.
Return = (G63P0) AND (power control mode signal off)
When the G63P0 state is specified in the machining program, and when the power control mode signal is set to 0, power control mode is canceled.
Usually, the G63P0 state occurs immediately after the CNC is started or upon a reset. So, if G63P1 is not specified, G63P0 need not be specified.
This means that, if a transition to the power control mode is made using an external signal, a return is accomplished by turning off the external signal. If transition to the power control mode is made by G63P1 specified in the machining program, a return is accomplished by specifying G63P0.
Power command changes made when the power control mode signal is used are indicated below (when only the power command is controlled).

```
O0002;
   ..................
  G91 G00 X100.0 Y100.0;
N1 G01 X70.0 S1000 P1000 Q100 F1000;
N2 G01 Y100.0 F750;
N3 G01 X–70.0 F500;
N4 G01 Y–100.0 F300;
  G91 G00 X0 Y0;
  M30;
```



Power control signal and power command

• Power control mode on signal PCMD <F224#1>

[Classification] Output signal

[Function] Notifies the PMC that the power control mode is set.

[Operation] Output when transition to the power control mode is completed.

• Signal address

	#7	#6	#5	#4	#3	#2	#1	#0
G0224							PWCTL	

	#7	#6	#5	#4	#3	#2	#1	#0
G0224							PCMD	

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
15000			FLT					LPC

LPC

0 : Disables the power control function.
1 : Enables the power control function.

When LPC is set to 0, transition to power control mode is disabled. The G code and input signal for setting power control mode are ignored.

FLT

0 : Disables the filter for power control.
1 : Enables the filter for power control.

When FLT is set to 1, parameter No. 15094 is enabled to allow an output command to be delayed.

	#7	#6	#5	#4	#3	#2	#1	#0
15089	LP8	LP7	LP6	LP5	LP4	LPZ	LPY	LPX

LP*

0 : Does not use the *–axis for feedrate calculation.
1 : Uses the *–axis for feedrate calculation.

Each laser output element is calculated using the composite feedrate along the axes selected with this parameter. When no axis is selected, the specification of the first axis (LPX) and the second axis (LPY) is assumed.

15090	Minimum power value

[Unit of data] W

[Valid data range] 0 to 7000

This parameter specifies a minimum power clamp value. If the power value calculated from the current feedrate becomes lower than the value set in this parameter, the power value is clamped to the value set in this parameter.

15091	Minimum pulse frequency

[Unit of data] Hz

[Valid data range] 5 to 2000

This parameter specifies a minimum pulse frequency clamp value. If the frequency calculated from the current feedrate becomes lower than the value set in this parameter, the frequency is clamped to the value set in this parameter.

15092	Minimum pulse duty cycle

[Unit of data] %

[Valid data range] 0 to 100

This parameter specifies a minimum pulse duty cycle clamp value. If the duty cycle calculated from the current feedrate becomes lower than the value set in this parameter, the duty cycle is clamped to the value set in this parameter.

15094	Power control filter time constant

[Unit of data] ms

[Valid data range] 8 to 32767

This parameter specifies a time constant for the primary delay filter used to delay the power command for synchronization with machine feed. The primary delay filter is expressed as follows:

$$O_n = O_{n-1} + (I_n - O_{n-1})\frac{Ts}{Tc}$$

Ts: 8 ms (sampling time)

Tc: To be set in this parameter (time constant)

15095	Allowable variation in power control speed

[Unit of data] mm/min

[Valid data range] 0 to 255

The calculated feedrate slightly fluctuates also in a steady state, and the data items (specified power, pulse frequency, and pulse duty) calculated based on the feedrate also fluctuate. To prevent these unnecessary fluctuations, specify the width of a dead zone for feedrate fluctuations.

	#7	#6	#5	#4	#3	#2	#1	#0
15096	EGM					PCD	PCF	PCP

PCP Power value control based on feedrate is

0 : Not applied.
1 : Applied.

PCF Pulse frequency control based on feedrate is

0 : Not applied.
1 : Applied.

PCD Pulse duty cycle control based on feedrate is

0 : Not applied.
1 : Applied.

EGM In edge machining mode and start–up machining mode:

0 : Power control mode is canceled.
1 : Power control is stopped only during the return distance movement in edge machining and start–up machining.

Edge machining mode and start–up machining mode refer to the states in which E numbers specifying the respective cutting conditions for edge machining and start–up machining have been selected.

15097	Parameter for output change rate calculation (output when feedrate F = 0)

[Unit of data] W

[Valid data range] 0 to 7000

This parameter is used to set a power value for the feedrate F = 0. An output change rate is calculated from the value set in this parameter and the specified output value (Mc) when power control mode is set.

15098	Parameter for frequency change rate calculation (frequency when feedrate F = 0)

[Unit of data] Hz

[Valid data range] 5 to 2000

This parameter is used to set a frequency for the feedrate F = 0. A frequency change rate is calculated from the value set in this parameter and the specified frequency value (Mc) when power control mode is set.

15099	Parameter for duty cycle change rate calculation (duty cycle when feedrate F = 0)

[Unit of data] %

[Valid data range] 0 to 100

This parameter is used to set a duty cycle for feedrate F = 0 corresponding to M0. A duty cycle change rate is calculated from the value set in this parameter and the specified duty cycle value (Mc) when power control mode is set.

Note

NOTE

1 With the parameter switch function, the parameter for the Z–axis cannot be changed.
2 With the parameter switch function, the parameter for the mirror axis used for constant optical path length control cannot be changed.
3 A display unit equivalent to the 10.4–inch LCD is required.
4 This function can be used together only with X–Y plane facing.
  It cannot be used together with any other function (such as cylindrical interpolation, normal–direction control, or three–dimensional cutting).

18.9 GAP CONTROL FUNCTION

Overview

When inputting the signal from a sensor which is installed at the end of the nozzle of the laser processing system to detect the distance from the workpiece, the CNC issues a move command for the Z–axis (or W–axis) to obtain a specified distance between the nozzle and the workpiece.
For details, refer to “FS16*i*–LB Connection Manual” (B–63663EN).

Command format

- **Set gap control mode**
  G13 P__ (R__L__);
- **Cancel gap control mode**
  G14;

P__: Reference displacement value
R__: Gap amount
L__: Gap controlled axis (switching between Z– and W–axes)

Explanation

When G13 is specified, the CNC enters the gap control mode to control the distance between the cutting nozzle and workpiece according to the signal input from the gap sensor so that the distance specified by P or R can be obtained.
When G14 is specified, or when a reset is made, the gap control mode is canceled.

NOTE
In the gap control mode, the CNC performs velocity control instead of position control, which is the control method usually used. Therefore, programmed move commands (G00, G01, and so on) and manual move commands must not be issued for the Z–axis in the gap control mode.

18.10 LASER HIGH–SPEED CONTROL FUNCTION

Overview

This function enables laser output functions in the AI contour control mode and the AI nano contour control mode. For details of the AI contour control mode and the AI nano contour control mode, refer to the FS16*i*–MB Operator's Manual (B–63534EN).

Specification

The following functions can be used in the AI contour control mode and the AI nano contour control mode:

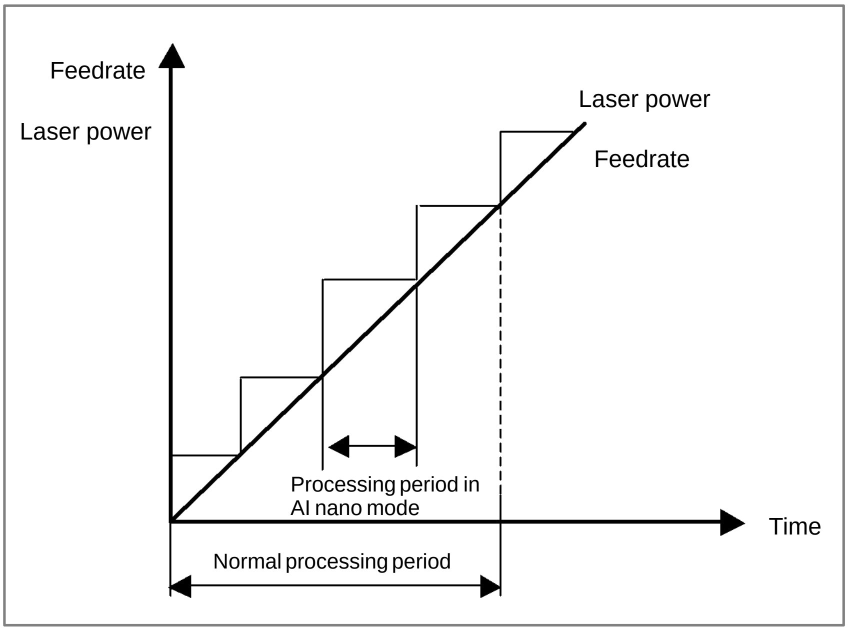
Laser output commands
Power control function
Beam delay control function
Laser output ON/OFF control high–speed processing

- **Laser output commands**

Laser output commands S, P, Q, and E can be specified in the AI contour control mode and AI nano contour control mode. Specify these commands in a block specifying G01 (G02, or G03) in the same way as ordinary laser output commands.

- **Power control function**

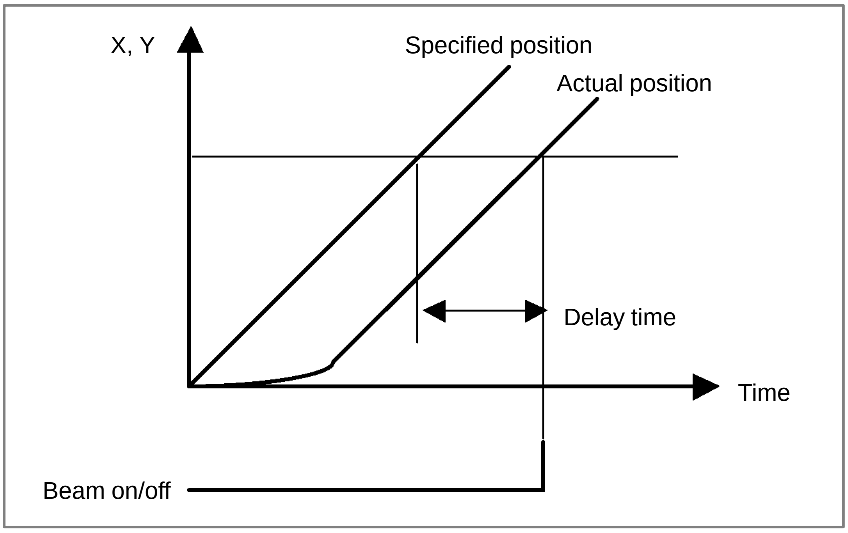
The power control function controls the laser output in proportion to the cutting feedrate. This prevents the laser output from becoming excessively high when the feedrate decreases in corners and other parts, therefore preventing cutting failures.
In the AI contour control mode and AI nano contour control mode, much more appropriate power control is enabled than in the normal mode.
For details of the power control function, see Section 18.3.



Conceptual diagram of power control

- **Beam delay control function**

The beam delay control function controls the laser output timing according to the delay in the machine.
The beam output timing can be delayed according to the delay timing set in parameter No. 15219.
In the AI contour control mode and AI nano contour control mode, control is performed with a shorter processing period than the processing period used in the normal mode to enhance reduction of variation in machining start point.
For details of the beam delay control function, see Section 18.4.



Conceptual diagram of beam delay control

- **Beam on/off control**

In the AI contour control mode and AI nano contour control mode, the execution of programmed laser output commands is controlled at shorter processing intervals than in the normal mode. Even when laser machining is performed at high speed, high precision is attained.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
7050		MI1	MI0					

MI1, MI0 Set the batch transfer quantity in the AI contour control mode and AI nano contour control mode.

MI1	MI0	Batch transfer quantity
0	0	1
0	1	2
1	0	4

	#7	#6	#5	#4	#3	#2	#1	#0
15005	DLY							

DLY

0 : Disables the beam delay function.
1 : Enables the beam delay function.

15219	Beam output condition delay time

[Unit of data] 8msec or 2msec (depending on the batch transfer quantity)

[Valid data range] 0 to 8

Switching of beam output conditions is delayed by the time set in this parameter.

Remarks

- This function is optional.
- Before this function can be used, the AI contour control or AI nano contour control function option is required.
- When using this function, set the batch transfer quantity to 4. If other than 4 is set, high–speed processing is not performed.
- This function is enabled for two–dimensional machining only.
- When using this function, disable AI contour control and AI nano contour control for the Z–axis.

19 THREE–DIMENSIONAL CUTTING FUNCTION

19.1 ATTITUDE CONTROL

Overview

If attitude control is specified with five or more controlled axes specified, nozzles are put under attitude control with the fourth and fifth axes used as rotational axes, thus realizing a three–dimensional machine tool.
The following two attitude control schemes are provided according to the shape of nozzle heads.

1) Attitude control A (zero–offset head)
2) Attitude control B (offset head)

Ask the machine tool builder which scheme your machine tool has.

Limitations

The following limitations apply when the attitude control function is provided. The terms used in the description are defined as follows:

a) The term ”three–dimensional machine tool” refers to a machine tool that performs attitude control.
b) The term “three–dimensional cutting” refers to cutting implemented using linear– and rotary–axis commands.
c) The term “two–dimensional cutting” refers to cutting implemented using linear–axis commands.

1) The following functions cannot be used for three–dimensional cutting. Do not include these functions in three–dimensional programs.
   - Advanced preview control
   - AI contour control/AI nano contour control
   - Laser power control
   - Step control
   - Edge cutting function
   - Start–up cutting function
   - Cutting restart
2) The following functions cannot be used for three–dimensional cutting.
   - High–precision contour control and related functions
   - High–speed laser cutting

19.2 SPATIAL CIRCULAR INTERPOLATION (G12)

Spatial circular interpolation can be performed by specifying midpoints and end points following G12.

Format

```
G12    Xm__Ym__Zm__Am__Bm__;
       Xe__Ye__Ze__Ae__Be__;

       A: Fourth axis
       B: Fifth axis
```

Table 19.2 Explanation of the Command Format

Command	Explanation
G12	Spatial circular interpolation
Xm_	Amount of travel to the midpoint on the X–axis
Ym_	Amount of travel to the midpoint on the Y–axis
Zm_	Amount of travel to the midpoint on the Z–axis
Am_	Amount of travel to the midpoint on the A–axis
Bm_	Amount of travel to the midpoint on the B–axis
Xe_	Amount of travel to the end point on the X–axis
Ye_	Amount of travel to the end point on the Y–axis
Ze_	Amount of travel to the end point on the Z–axis
Ae_	Amount of travel to the end point on the A–axis
Be_	Amount of travel to the end point on the B–axis

Explanations

- **G12**

G12 specifies spatial circular interpolation.
This command is modal and effective until the next G00, G01, G02, or G03 is issued.

- **Amount of travel along an arc**

Specify the amount of travel to the midpoint on the arc for Xm, Ym, Zm, Am, and Bm and the amount of travel to the end point for Xe, Ye, Ze, Ae, and Be.
Specify absolute or incremental values for the amounts according to G90 or G91.
When incremental values are specified, the end point is indicated by the distance from the midpoint.

Limitations

1) For spatial circular interpolation, the path to the end point along an arc can be obtained by specifying the midpoint as well as the end point. The movement can be divided into the following two blocks: a block from the start point to the midpoint and a block from the midpoint to the end point. Therefore, during single–block operation, the nozzle does not move at the first start, but moves from the start point to the end point at the second start.

2) When spatial circular interpolation is specified continuously, the end point of the previous block is used as the start point of the next block. (Always specify the midpoint and end point because the midpoint of the previous block is not used as the start point of the next block.)

3) The G12 command is treated as linear interpolation if it cannot create an arc because:

   (a) Neither a midpoint block nor an end–point block has been specified,

   (b) The start point, midpoint, and end point are on a straight line, or

   (c) Two or more of the start point, midpoint, and end point are identical.

4) The G12 command cannot specify axis movement for any axis other than the first to fifth axes, because alarm 4025 occurs.

5) G12 cannot be issued during coordinate system rotation (G84).

6) G12 cannot be issued during three–dimensional coordinate transform (G68).

7) G12 cannot be issued during cutter compensation. It must be issued with cutter compensation canceled.

19.3 SPATIAL CORNER ROUNDING (G33 AND G34)

Spatial corner rounding automatically inserts an arc of the specified radius into the corner made by two linear movements in space.

Format

```
G33    G01
       X__Y__Z__A__B__R__F__;
       X__Y__Z__A__B__;
       R__;
       X__Y__Z__A__B__;
G34;
```

Table 19.3 Explanation of the Command Format

Command	Explanation
G33	Starts the spatial corner rounding mode.
G34	Stops the spatial corner rounding mode.
G01	Linear interpolation
X_	Amount of travel to the end point on the X–axis
Y_	Amount of travel to the end point on the Y–axis
Z_	Amount of travel to the end point on the Z–axis
A_	Amount of travel to the end point on the A–axis
B_	Amount of travel to the end point on the B–axis
R_	Specifies the radius of an arc.
F_	Feedrate

Explanations

- **Starting and stopping spatial corner rounding mode**

G33 specifies spatial corner rounding mode while G34 cancels the mode. When a one–shot G code is issued in the G33 mode, the spatial corner rounding mode is canceled.

- **Move command in spatial corner rounding mode**

Use G01 to issue a move command in G33 mode.
If a G code in group 01 other than G01 is issued in G33 mode, an alarm occurs.

- **Specifying the radius of an arc**

Use R to specify the radius of an arc to be inserted.
When no R command is specified in the block in which G33 is specified, the nozzle moves linearly.
When only the radius of an arc is specified in a block, an arc is inserted into the corner between the blocks before and after that block.
When an R command is specified following X, Y, and Z commands in a block, an arc is inserted into the corner between that block and the next block.

Limitations

1) When the angle made by the current block with the next block is less than 1° or 180° ± 1° , the nozzle moves linearly at the corner with no arc inserted.
2) During single–block operation, the nozzle moves to the end point of the corner, then stops.
3) During MDI operation, after the first block is entered and the start button is pressed, then the second block is entered and the button is pressed, the nozzle moves along the path specified in the first block to the end point of the corner. When G34 is entered in the third block and the start button is pressed, the nozzle moves to the end point of the path specified in the second block. When a move command is specified in the third block, the nozzle moves in the same way as for the first block.
4) G33 cannot be issued during coordinate system rotation (G84).
5) The following G code commands cannot be issued in the G33 mode, because alarm 4017 occurs:
   G00, G02, G03, G12, G40, G41, G42, G28, G29, G30, G31, and G53
6) If any of the following G codes is issued simultaneously with the G01 move command, no corner rounding is inserted:
   G09, G54 to G59, and G codes in group 00
7) Any command other than the G01 move command cannot be issued immediately after G33 is issued; it is ignored if issued.
8) Do not specify a block that does not involve movement in the G33 mode.
   (The block would be executed before the immediately preceding G01 move command.)
9) If a command that does not involve movement is specified in the same block as the G01 which is in the G33 mode, it is executed simultaneously with the previous G01 move command but one.
10) Do not issue an auxiliary function such as an M or T code during the G33 mode.
    (An auxiliary function issued in the same block as G01 is executed simultaneously with the previous G01 move command but one and output at two points; linear and arc start points.)
11) G33 cannot be used simultaneously with cutter compensation. Issue G33 with cutter compensation canceled.
12) An axis move command for any axis other than the first to fifth axes cannot be issued in the G33 mode, because alarm 4025 occurs.

19.4 THREE–DIMENSIONAL COORDINATE CONVERSION FUNCTION (G68 AND G69)

The origin of coordinate conversion, nozzle direction, and positive X–axis direction can be specified in the block in which the three–dimensional coordinate conversion command (G68) is specified to change the programmed coordinate system, as follows:

Format

```
G68  X_  Y_  Z_  A_  B_  I_   J_  K_ ;      Turn on three–dimensional
                                             coordinate transform.
     G01  X_  Y_  Z_  A_  B_  F_ ;  ┐
     G01  X_  Y_  Z_  A_  B_ ;      │
     G01  X_  Y_  Z_  A_  B_ ;      │
     G02  X_  Y_  I_  J_ ;          ├       Block to be converted
     G03  X_  Y_  I_  J_ ;          │
     G01  X_  Y_  Z_  A_  B_ ;      ┘
G69;                                         End of three–dimensional
                                             coordinate transform
```

Table 19.4 Explanation of the Command Format

Command	Explanation
G68	Three–dimensional coordinate conversion mode
G69	Cancellation of three–dimensional coordinate conversion
X_	X coordinate of the origin of coordinate conversion
Y_	Y coordinate of the origin of coordinate conversion
Z_	Z coordinate of the origin of coordinate conversion
A_	A coordinate in the nozzle direction
B_	B coordinate in the nozzle direction
I_	X element of the vector suggesting the positive X–axis direction
J_	Y element of the vector suggesting the positive X–axis direction
K_	Z element of the vector suggesting the positive X–axis direction

Explanations

- **Three–dimensional coordinate conversion mode/cancellation**

The G68 command specifies three–dimensional coordinate conversion mode.
G69 cancels the mode.
When reset is performed during three–dimensional coordinate conversion, three–dimensional coordinate conversion mode is canceled and the normal coordinate system is restored.

- **Origin of coordinate conversion**

Specify the origin of coordinate conversion with an absolute value.

- **Nozzle direction**

For the nozzle direction, three–dimensional coordinate conversion is not performed. That is, the nozzle direction is assumed to be oriented in the Z–axis direction.

- **Vector suggesting the positive X–axis direction**

Specify the vector with incremental values starting from the origin of coordinate conversion.
When (I,J,K) is omitted, the X–axis is assumed to be parallel to the XY plane before conversion.

Limitations

1) G68 cannot be issued simultaneously with scaling (G51) or coordinate system rotation (G84).
2) Neither G68 nor G69 can be issued if a cutter compensation offset vector is left in the CNC.
3) Three–dimension transform (G98) cannot be issued in the G68 mode.
4) Spatial arc interpolation (G12) cannot be issued in the G68 mode.
5) A machine coordinate system–based move command (G27, G28, G29, or G53) cannot be issued in the G68 mode.
6) If JOG or MDI intervention is required in the G68 mode, an absolute (G90) command must be used to specify movement in the G68 mode.
7) The work coordinate system cannot be changed in the G68 mode.

19.5 PROCESSING HEAD A–AXIS LENGTH COMPENSATION FUNCTION (G71)

The length of the second arm of a three–dimensional offset–type machine may be changed due to replacement of the processing head. In this case, nozzle tip fixing control cannot operate normally without reference position return.
This function enables tip fixing control to be performed with the current position without reference position return by entering the length of the replaced nozzle from a program when a processing head is replaced.

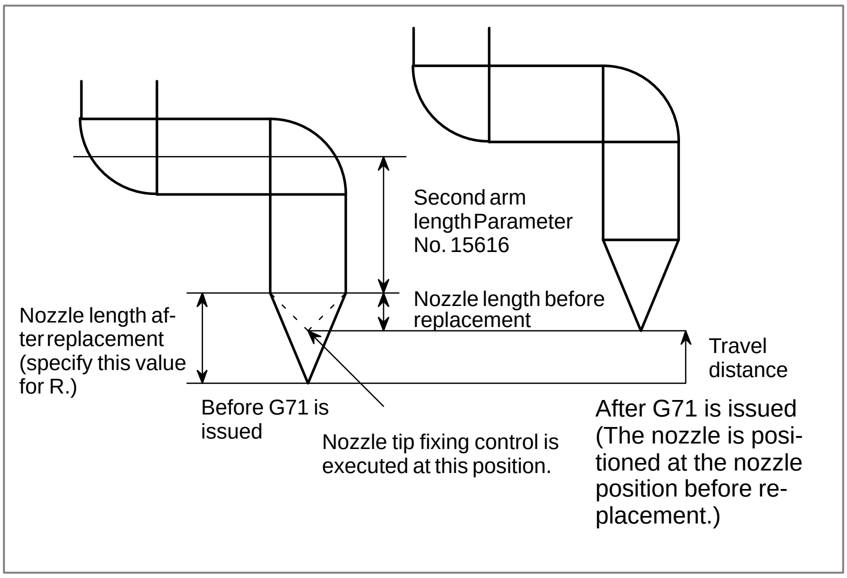
Format

```
G71  R__;

     R__  :    Length of the replaced nozzle
               Unit      :   0.001mm (metric output)
                             0.0001inch (inch output)
```

Explanations

- **Operation**

When G71 is executed, parameter 15619 (nozzle length) is rewritten with the value specified by address R, and the nozzle is positioned to the nozzle tip position before replacement at a rapid traverse rate by movement along the X, Y, and Z axes. At this time, the position along the fourth and fifth axes does not change. Movement by G71 is the same as positioning by G00. The feedrate and other settings specified for G00 are used.
The length of the arm used for tip fixing control is indicated by the value obtained by adding the length of the second arm set for parameter No. 15616.



Parameter

15616	Second arm length

Sets the length of the second arm.

Range of valid settings: 0 to 500000 (metric output)
0 to 196850 (inch output)

Unit : 0.001 mm (metric output)
0.0001 inch (inch output)

15619	Nozzle length

Sets the length of the nozzle.
When G71 is executed, the value specified for address R is set.

Range of valid settings : PRM. 15616 + PRM. 15619 ≦ 500000 (metric output)
PRM. 15616 + PRM. 15619 ≦ 196850 (inch output)

Unit : 0.001 mm (metric output)
0.0001 inch (inch output)

Alarm

Alarm number	Explanation
P/S4021	A negative value is specified as the length of the nozzle. The nozzle length falls outside the specification range. The total of the second arm length and nozzle length must be no more than 500 mm (19.685 inches).

Limitations

1 A change made to the nozzle length, using G71, is retained as a parameter even after the power is turned off. To use it in relation to nozzle compensation, cancel it, using G71R0.
When rewriting parameter 15619 with other than G71, always perform reference position return.

2 Before movement by G71, the stroke limit is checked.

3 G71 is a one–shot G code.

19.6 IMPROVEMENT IN NOZZLE LENGTH COMPENSATION

Overview

Normally, the travel distance in G71R_ is used for executing a positioning command (Fig. 19.6 (a)), but it can be used for interpolation at the same time when the next move command is executed. (Fig. 19.6 (b))

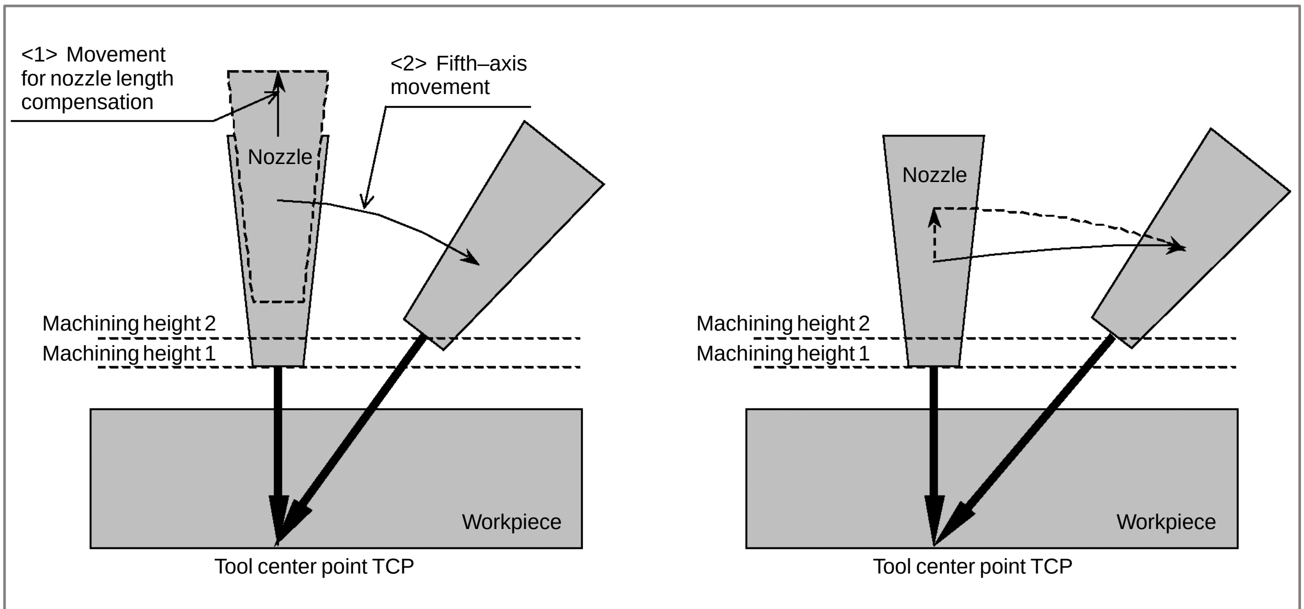


Fig.19.6 (a) Fig.19.6 (b)

The following shows an example in which the nozzle becomes shorter (the machining height decreases) depending on the machining attitude.

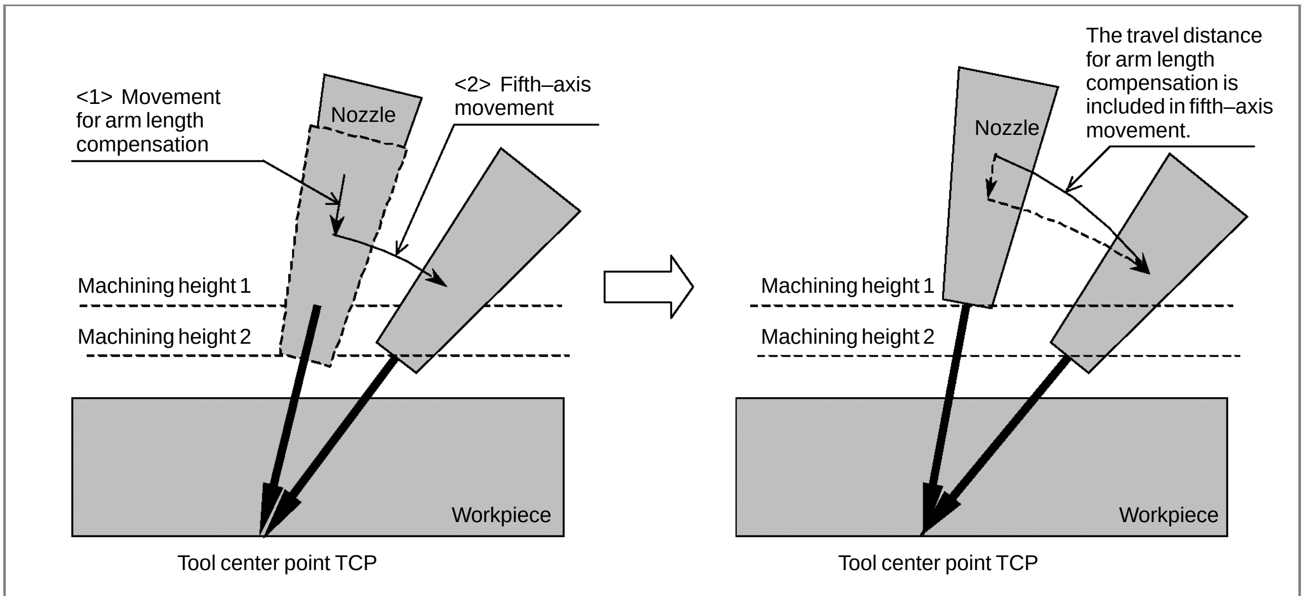
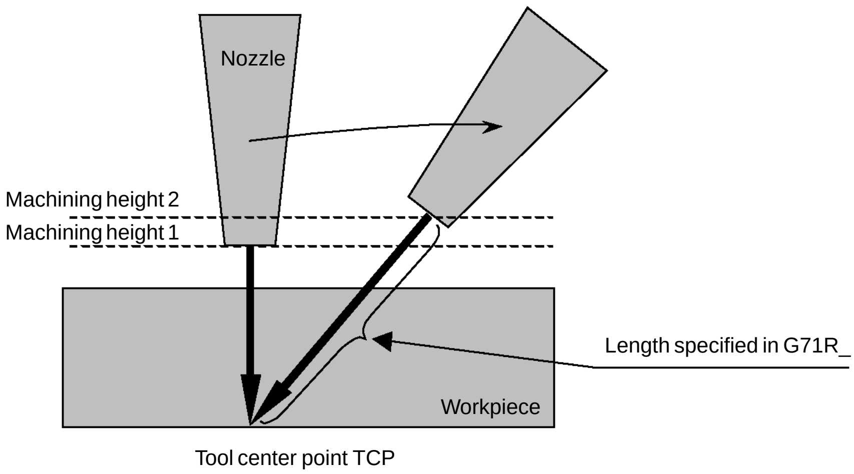


Fig.19.6 (c) Fig.19.6 (d)

Specification

If bit 1 (NZB) of parameter No. 15625 is set to 1, the compensation value in G71R_ is stored in the NC, and it is used for interpolation at the same time when the move command specified in the next or a subsequent block is executed. As the R value specified in G71R_, specify the second arm length compensation value in the attitude for beveling (the angle of the fifth axis). (See the figure below.)
When machining is performed with the fifth axis attitude kept unchanged, the beveling command (G71P1) need not be specified. The beveling command is required when machining is performed by changing the fifth axis attitude while maintaining the current machining height.



Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
15602					OF2			

[Data type] Bit

OF2 Be sure to set this bit to 1 when using this function.

	#7	#6	#5	#4	#3	#2	#1	#0
15625							NZB	

[Data type] Bit

NZB Compensation movement by the G71R_ command causes:

0 : Positioning in that block.
1 : Interpolation at the same time when a block specifying the next interpolation movement command is executed.

Sample program

Bit 1 (NZB) of parameter No. 15625 is set to 1. (A: Fourth axis, B: Fifth axis)

```
   •
   •
N01 G71 R piercing position;   ... 2nd arm length compensation (no axis movement)
N02 G01 X_ Y_ F_;              ... Perform interpolation of compensation value together with
                                   move command
N03 G24 S_ P_ Q_ R_;           ... Piercing
N04 G71 R cutting position;    ... 2nd arm length compensation (no axis movement)
N05 G01 X_ Y_ B_ F_;           ... Perform interpolation of compensation value together with
                               ... move command
   •
N10 G71 P1;                    ... Enable beveling compensation
   •
   •                               (Program for changing beveling angle)
   •
N20 G71 P0;                    ... Disable beveling compensation
   •
```

Notes

(1) In this specification, bit 2 (OF2) of parameter No. 15602 must be set to 1.

(2) If a block specifying the next movement after the G71R_ command is a positioning block, compensation is not performed, and the compensation value is maintained until the next interpolation command is encountered.

(3) If the travel amount of the block specified next to the G71R_ command is 0, compensation is not performed, and the compensation value is maintained until a block specifying the next movement is encountered.

(4) Be sure to specify the G71R_ command in the G71P0 mode. It is impossible to specify the G71P1 command before the compensation operation is completed (for example, when the above compensation operation is maintained).

(5) When the automatic feedrate override function operates during compensation, operation is performed at a feedrate lower than the clamp feedrate set in parameter Nos. 15621 and 15622.

(6) After G71R_ is specified, when a reset is applied during execution of the compensation operation or before the compensation operation, compensation for the rest of the compensation value is performed at a time.

19.7 THREE–DIMENSIONAL TRANSFORM FUNCTION (G98, G99)

When a given point in the coordinate system for the processing program is specified as the base point and a target point corresponding to the base point is also specified, the processing program, transformed in three dimensions, runs. Mirror image transform and three–dimensional movement transform are provided for the three–dimensional transform function.

Format

To specify base and target points (see ”Explanations”) for this function, the following methods are available:

(1) Specifying the points in a program

(a) Mirror image transform

```
G98  P0  X_  Y_  Z_ ;                      Reference point
G98  Q0  X_  Y_  Z_ ;                      Target point
     G01  X_  Y_  Z_  A_  B_  F_ ;  ⎫
     G01  X_  Y_  Z_  A_  B_ ;      ⎬      Block to be subjected to
     G02  X_  Y_  I_  J_ ;          ⎪      transform
     G03  X_  Y_  I_  J_ ;          ⎭
G99;                                       End of three–dimensional
                                           transform
```

(b) Three–dimensional movement transform

```
G98  P1  X_  Y_  Z_ ;                      Reference point 1
G98  P2  X_  Y_  Z_ ;                      Reference point 2
G98  P3  X_  Y_  Z_ ;                      Reference point 3
G98  Q1  X_  Y_  Z_ ;                      Target point 1
G98  Q2  X_  Y_  Z_ ;                      Target point 2
G98  Q3  X_  Y_  Z_ ;                      Target point 3
     G01  X_  Y_  Z_  A_  B_  F_ ;  ⎫
     G01  X_  Y_  Z_  A_  B_ ;      ⎬      Block to be subjected to
     G02  X_  Y_  I_  J_ ;          ⎪      transform
     G03  X_  Y_  I_  J_ ;          ⎭
G99;                                       End of three–dimensional
                                           transform
```

When the program is executed, the coordinates at the base and target points are written into the settings. When Q0 or Q3 is read, transform processing starts according to the values of the settings.

(2) Specifying the points using settings

(a) Mirror image transform

```
G98  Q0 ;                                  Turn on mirror
                                           image transform.
     G01  X_  Y_  Z_  A_  B_  F_ ;  ⎫
     G01  X_  Y_  Z_  A_  B_ ;      ⎬      Block to be subjected to
     G02  X_  Y_  I_  J_ ;          ⎪      transform
     G03  X_  Y_  I_  J_ ;          ⎭
G99;                                       End of three–dimensional
                                           transform
```

(b) Three–dimensional movement transform

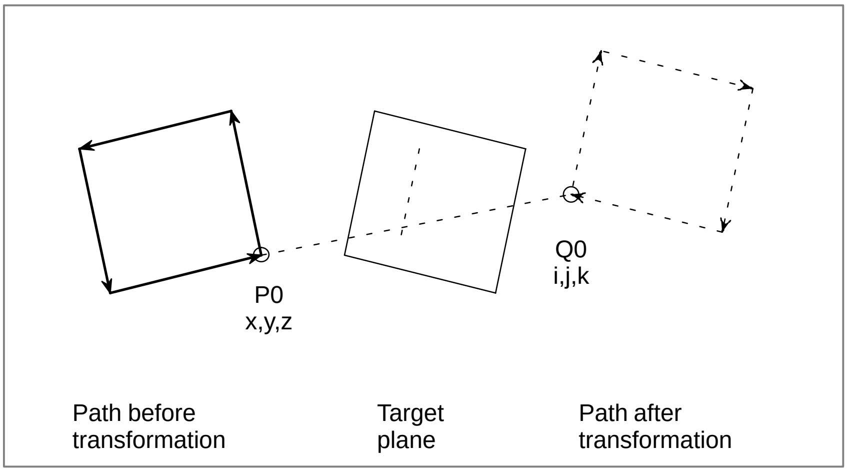
```
G98  Q3 ;                                  Turn on three–
                                           dimensional transform.
     G01  X_  Y_  Z_  A_  B_  F_ ;  ⎫
     G01  X_  Y_  Z_  A_  B_ ;      ⎬      Block to be subjected to
     G02  X_  Y_  I_  J_ ;          ⎪      transform
     G03  X_  Y_  I_  J_ ;          ⎭
G99;                                       End of three–dimensional
                                           transform
```

Set the coordinates at the base and target points on the setting screen (see Part III, "Operation") in advance and specify the above command in the program.
When the program is executed and the above command is read, transform processing starts according to the values of the settings.

Explanations

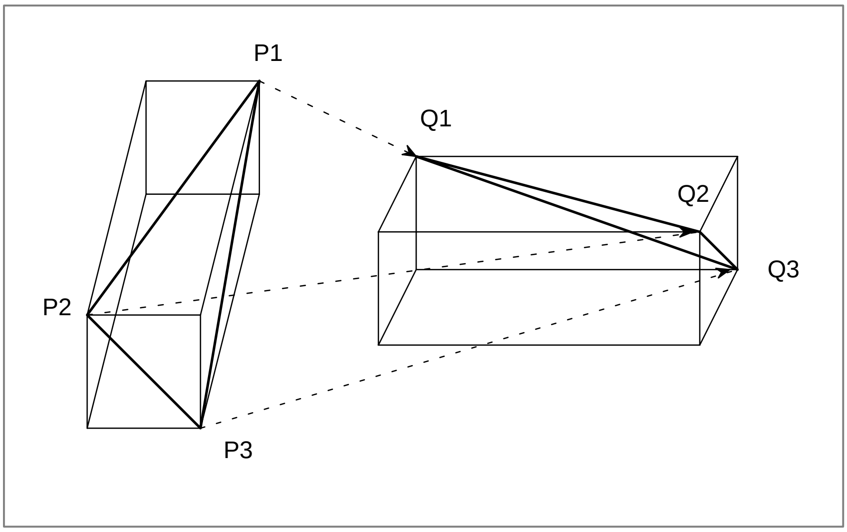
• Mirror image transform

When the base point before transformation, P0 (X,Y,Z), and target point after transformation, Q0 (X,Y,Z), are specified, the plane which perpendicularly bisects the straight line connecting these two points is obtained as the target plane. On the plane, a mirror image transform is performed for the program to be transformed.



• Three–dimensional movement transform

When the three base points before transformation, (P1,P2,P3), and the target points after transformation, (Q1,Q2,Q3), are specified, the coordinate system after transformation is obtained. The program to be transformed moves to the coordinate system.



- **Specifying mirror image transform and movement transform simultaneously**

Mirror image transform and movement transform can be specified simultaneously.
For example, when a movement transform is specified, then a mirror image transform is specified without specifying a cancel command as follows, both transforms are effective.
Cancel command G99 cancels all commands. To cancel multiple transform commands and specify only one of these transform commands, cancel these commands using G99, then respecify the required command.
When a command becomes effective, ”(Selected)” is displayed following the title on the setting screen.

```
 G98Q0 ;
[Move command group]   A mirror image transform is effective.
 G99 ;
 G98 Q3 ;
[Move command group]   A movement transform is effective.
 G98Q0 ;
[Move command group]   A movement transform is effective.
                       A mirror image transform is effective.
 G99 ;
[Move command group]   Normal operation
 G98Q0 ;
[Move command group]   A mirror image transform is effective.
```

Limitations

1) Assume that the following program is executed. When control is transferred from move command group (a) to move command group (b) and from move command group (b) to move command group (c), movement stops and the beam is turned off.

```
[Move command group (a)]  ─┐
 G98Q0 ; or  Q3 ;          │
[Move command group (b)]   │  The three–dimensional
 G99 ;                     │  transform function is effective.
[Move command group (c)]  ─┘
```

2) When specifying a complete circle with G02 and G03 commands in the G98 mode, split it into two blocks.
3) Cutter compensation cannot be performed in the G98 mode. Before specifying cutter compensation, issue three–dimensional coordinate transform (G68) in the G98 mode.
4) G98 cannot be issued simultaneously with scaling (G51) or coordinate system rotation (G84).
5) A machine coordinate system–based move command (G27, G28, G29, or G53) cannot be issued in the G98 mode.
6) If JOG or MDI intervention is required in the G98 mode, an absolute (G90) command must be used to specify movement in the G98 mode.

7) Immediately after G98, three–dimensional transform is performed on nozzle movement based on the current position of the nozzle. So, unexpected changes may occur to the nozzle position. Before issuing G98, therefore, position the fourth and fifth axes in such a point where changes to the nozzle position can be foreseen.
(It is recommended that the fourth and fifth axes be set to the origin before the nozzle is returned to the basic position.)

8) A three–dimensional coordinate transform command (G68) can be issued in the G98 mode.

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
15601					RDC			

[Data type] Bit

RDC In three–dimensional conversion, the rotation around the fourth and fifth axes is:

0 : In a direction in which the distance of movement becomes shorter. When a software OT area is encountered if the axis rotation is performed in the shorter–distance direction, however, the rotation is performed in the opposite direction.

1 : Not the same as usual. For the fourth axis, the direction of travel before the conversion is assumed. If there are two solutions, the one nearer the amount of travel before the conversion is selected. The solution for the fifth axis is determined with the solution selected for the fourth axis.

15623	Angle at which rotation is assumed to be only plane rotation in three–dimensional conversion

[Data type] Two–word

[Unit of data] 0.001 deg

[Valid data range] 0 to 89999

If, in three–dimensional conversion, the difference between the inclination of the plane containing three reference points (P1, P2, and P3) and that of the plane containing three target points (Q1, Q2, and Q3) is equal to or less than the angle specified in this parameter, the inclination difference is ignored and the rotation is assumed to be only rotation in the X–Y plane.
If this parameter is 0, this compensation process is not executed.

19.8 FEEDRATE CLAMP FUNCTION IN POSITION CONTROL B

If the fourth and fifth axes are specified simultaneously with the X–, Y–, and Z–axes, the X–, Y–, and Z–axis feedrates may become very high after position control B compensation is applied. This function is intended to clamp the feedrate specified in the command so that the X–, Y–, and Z–axis feedrates with position control B compensation applied will not exceed the respective rapid traverse rates (parameter No. 1420). It is enabled for G01, G02, G03, G12, and interpolation–type G00 commands.

Explanations

The clamp feedrate for block “G91 G01 Xx Yy Zz Aa Bb Ff” is obtained as explained below.
In this description, the A– and B–axes are specified as the fourth and fifth axes, respectively.
The maximum X–axis feedrate with position control B applied can be estimated using the following expression.

Fxmaz=[f*x/L]+[f*a/L*(π*R/180)]+[f*b/L*(π*P/180)]

$L=\sqrt{(x^2+y^2+z^2)}$: Distance through which the nozzle tip mov

$R=\sqrt{[(\text{Length of the first arm})^2+(\text{Length of the second arm})^2]}$

P; Length of the second arm

f*x/L: X–axis feedrate derived from the Xx command

f*a/L*(π*R180): Maximum X–axis feedrate generated by applying the Aa command compensation

f*b/L*(π*P180): Maximum X–axis feedrate generated by applying the Bb command compensation

Calculate “f” so that Fxmax is not higher than the X–axis traverse rate (parameter No. 1420).

fx=Rpdx/{[x/L]+[a/L*(π*R/180)]+[b/L*(π*P/180)]}

fx: “f” obtained for the X–axis

Rpdx: X–axis rapid traverse rate (parameter No. 1420)

Similarly, obtain the clamp feedrate from the maximum Y–axis feedrate.

Fymax=[f*y/L]+[f*a/L*(π*R/180)]+[f*b/L*(π*P/180)]

fy=Rpdy/{[y/L]+[a/L*(π*R/180)]+[b/L*(π*P/180)]}

fy: “f” obtained for the Y–axis

Rpdy: Y–axis rapid traverse rate (parameter No. 1420)

Because no Aa command–based compensation occurs for the Z–axis,

Fzmax=[f*z/L]+[f*b/L*(π*P/180)]

fz=Rpdz/{[z/L]+[b/L*(π*P/180)]}

fz: “f” obtained for the Z–axis

Rpdz: Z–axis rapid traverse rate (parameter No. 1420)

The smallest of “fx,” “fy,” and “fz” is used as the minimum cutting feedrate clamp value.

Remark) [f*a/L*(π*R/180)] and [f*b/L*(π*P/180)] do not occur at the same time. So, each axis feedrate after clamped is slightly lower than the corresponding rapid traverse rate.

In “G91 G02 Xx Yy Zz Aa Bb Ff”, the maximum feedrates of the X–, Y–, and Z–axes are estimated at:

Fxmax=f+[f*a/L*(π*R/180)]+[f*b/L*(π*P/180)]

Fymax=f+[f*a/L*(π*R/180)]+[f*b/L*(π*P/180)]

Fzmax=[f*y/L]+[f*b/L*(π*P/180)]

Therefore, the following expressions give clamp values for the feedrates.

$$fx=Rpdx/\{1+[a/L*(\pi*R/180)]+[b/L*(\pi*P/180)]\}$$
$$fy=Rpdy/\{1+[a/L*(\pi*R/180)]+[b/L*(\pi*P/180)]\}$$
$$fz=Rpdz/\{[z/L]+[b/L*(\pi*P/180)]\}$$

Parameters

Setting bit 1 of parameter No. 15602 effects clamping with the maximum feedrate after position control taken into account.

	#7	#6	#5	#4	#3	#2	#1	#0
15602		FMX						

FMX If an offset–type nozzle is used, the cutting speed is:

1 : Not clamped.

0 : Clamped by considering the maximum speed after position control.

Example problem

Length of the first arm: 130 mm
Length of the second arm: 145 mm
Rapid traverse rates for individual axes: X = 24000, Y = 24000, Z = 20000, A = 4000, and B = 80000
Upper limit to machining feedrate for the A– and B–axes: A = 4000 and B = 8000
Under the above conditions, let us calculate the feedrate clamp value for the following command block:

G91 G01 X10. Y20. Z30. A45. B90. F5000;

Distance through which the nozzle tip moves is:

$$L=\sqrt{(10^2+20^2+30^2)}=37.416$$

Let f be the feedrate of the nozzle tip after clamped.

The specified X–axis feedrate component is: $F_{XL}=f*10/L=0.267*f$
The specified Y–axis feedrate component is: $F_{YL}=f*20/L=0.535*f$
The specified Z–axis feedrate component is: $F_{ZL}=f*30/L=0.802*f$
The A–axis feedrate is: $F_{AL}=f*45/L=1.203*f$
The B–axis feedrate is: $F_{BL}=f*90/L=2.405*f$

Assuming that the A–axis operates with the B–axis at 90°,

The maximum X–axis feedrate under A–axis position control is:
$F_{XA}=F_{AL}*\pi*\sqrt{(130^2+145^2)}/180=4.089*f$
The maximum Y–axis feedrate under A–axis position control is:
$F_{YA}=F_{AL}*\pi*\sqrt{(130^2+145^2)}/180=4.089*f$
The maximum Z–axis feedrate under A–axis position control is:
$F_{ZA}=0$

Assuming that the B–axis operates with the A–axis at 0°,
The maximum X–axis feedrate under B–axis position control is:
$F_{XB}=F_{BL}*\pi*145/180=6.086*f$

Assuming that the B–axis operates with the A–axis at 90°,
The maximum Y–axis feedrate under B–axis position control is:
$F_{YB}=F_{BL}*\pi*145/180=6.086*f$
The maximum Z–axis feedrate under B–axis position control is:
$F_{XB}=F_{BL}*\pi*145/180=6.086*f$

Summing the specified X–, Y–, and Z–axis feedrate components and the X–, Y–, and Z–axis feedrates under A– and B–axis position control yields the following estimates:

The estimated maximum X–axis feedrate is:

$F_{XMAX}=F_{XL}+F_{XA}+F_{XB}=10.442*f$

The estimated maximum Y–axis feedrate is:

$F_{YMAX}=F_{YL}+F_{YA}+F_{YB}=10.71*f$

The estimated maximum Z–axis feedrate is:

$F_{ZMAX}=F_{ZL}+0+F_{ZB}=6.888*f$

If f is determined so that F_{XMAX}, F_{YMAX}, and F_{ZMAX} do not exceed the respective rapid traverse rates of the X–, Y–, and Z–axes:

f = 2298 because $F_{XMAX} = 10.442*f \leqq 24000$

f = 2241 because $F_{YMAX} = 10.71*f \leqq 24000$

f = 2904 because $F_{ZMAX} = 6.888*f \leqq 20000$

For any of the X–, Y–, and Z–axis feedrates to be lower than the respective rapid traverse rates, the nozzle tip feedrate must be 2241 mm/min.

19.9 AUTOMATIC FEEDRATE OVERRIDE UNDER POSITION CONTROL B

If a specified feedrate output to the motor of each axis exceeds the parameter–specified feedrate, this function applies automatic overriding instantly so that the output feedrate is lowered to within the parameter–specified feedrate. This way, it is possible to clamp, on an execution level, the X–, Y–, and Z–axis feedrate with position control B compensation applied.

Explanations

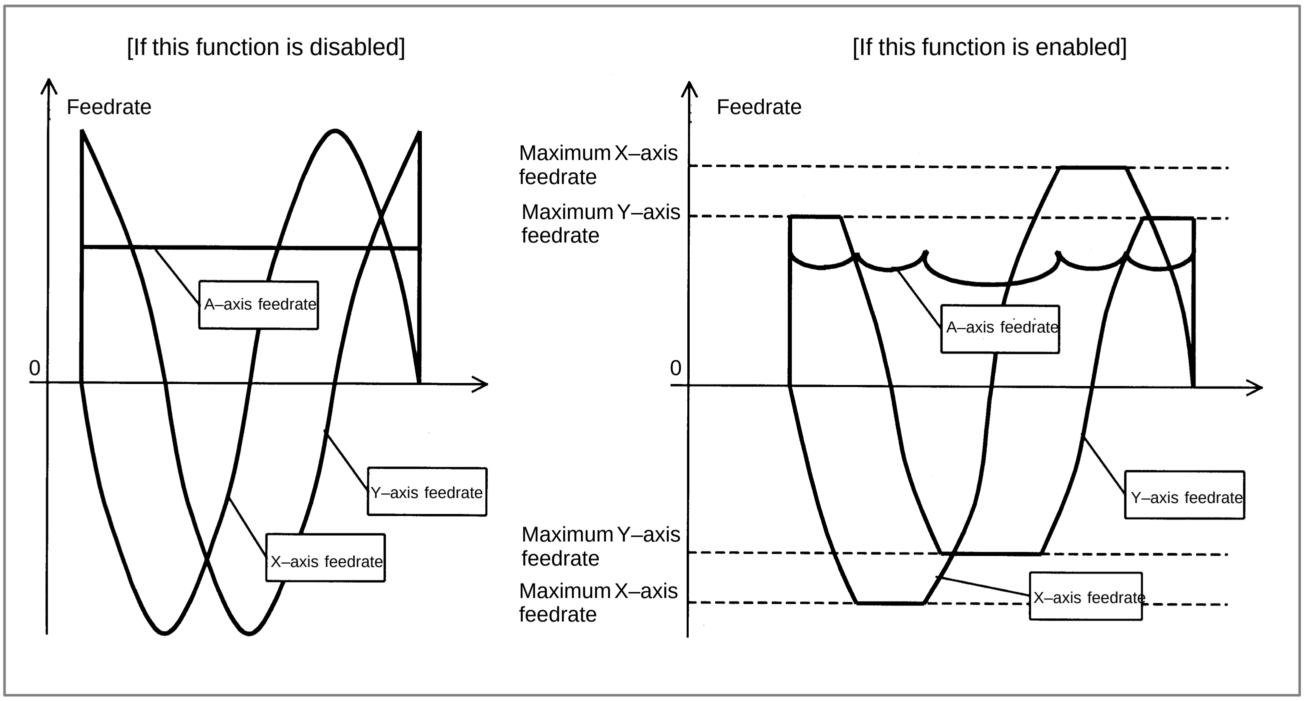
If the feedrate of any axis exceeds the feedrate specified in parameter No. 15621 during axis movement, automatic overriding is applied instantly to lower it to within the parameter–specified feedrate. Because this function works for all axes in common, no deviation from the specified tool path will occur during interpolation. Moreover, the override value can increase automatically so that the actual feedrate will not become lower than necessary. However, the override value will not exceed 100%, so the actual feedrate does not become higher than specified.

This function works in the cycle next to the one in which a feedrate higher than the limit is detected. So, the parameter–specified feedrate may be exceeded a little.

If a command–specified feedrate becomes higher than the feedrate specified in parameter No. 15621, the override value always changes. So, the nozzle tip feedrate does not become constant even within the same command block. Therefore, it is necessary to set parameter No. 15621 with a value that will not cause overriding during usual cutting.

This function works not only in automatic operation but also in all other modes such as jog and handle feed.

Block “G91 G01 X4. A360. F10000” causes overriding shown below. The A–axis is the fourth axis.



Parameters

15621	Maximum speed 1 after position control

[Data type] Two–word axis

[Unit of data, valid data range]

Increment system	Unit of data	IS–A, IS–B	IS–C
Millimeter machine	1 mm/min or 1 deg/min	0 to 240000	0 to 100000
Inch machine	0.1 inch/min or 1 deg/min	0 to 96000	0 to 48000

Specify the maximum speed after position control. The speed on each axis may be overridden so as not to exceed this setting.
The maximum speed check is not performed on an axis for which 0 is set.

15622	Maximum speed 2 after position control

[Data type] Two–word axis

[Unit of data, valid data range]

Increment system	Unit of data	IS–A, IS–B	IS–C
Millimeter machine	1 mm/min or 1 deg/min	0 to 240000	0 to 100000
Inch machine	0.1 inch/min or 1 deg/min	0 to 96000	0 to 48000

Specify the maximum speed after position control. The speed on each axis may be overridden so as not to exceed this setting. This parameter is effective when the position control B speed clamp select signal (G224#7, OTPMX2) is 1. When OTPMX2 is 0, parameter No. 15621 is effective. Parameter No. 15621 is effective to an axis for which 0 is set in this parameter.

Signal

	#7	#6	#5	#4	#3	#2	#1	#0
G224	OTPMX2							

Position control B feedrate clamp switching signal

OTPMX2 Selects which parameter is to be used to specify the maximum value for automatic feedrate overriding under position control B.

1 : Automatic overriding occurs based on parameter No. 15621.
0 : Automatic overriding occurs based on parameter No. 15622.

19.10 TORCH TURNING CONTROL FUNCTION

Overview

When beveling is performed, the bevel torch must always be positioned in a direction normal to a specified straight line or arc. The torch turning control function controls the bevel torch so that it is positioned in the normal direction without stopping the operation of the torch end at a point between adjacent blocks. This function is optional.

Specification

• Definition of a rotary axis

The rotary axis for torch turning control is specified in parameter No. 5490.
The rotation center of the rotary axis is an axis perpendicular to the X–Y plane (that is, the Z–axis).
The angle of the rotary axis is defined on a counterclockwise basis as follows:

0° :+X direction
90° :+Y direction
180° :–X direction
270° :–YX direction
(The angle ranges from 0 to 360°.)

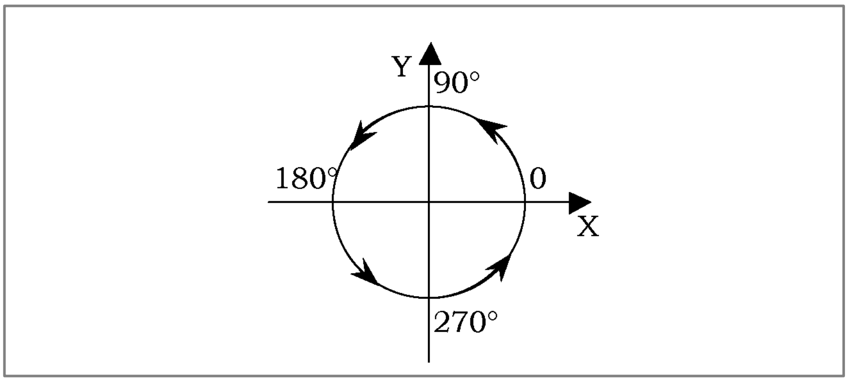


Fig. 19.10 (a) Angle definition

Parameter No. 5490 = Axis number of the rotary axis
The following parameters are set for the axis specified in parameter No. 5490:
Parameter No. 1006#0 = 1
Parameter No. 1008#0 = 1
Parameter No. 1260 = 360000
Parameter No. 1430 = Maximum rotation speed
Parameter No. 7052#0 = 1

For details of parameters, see “Parameter” described later.

- **G codes for torch turning control**

Torch turning control is specified with the following G codes:

G41.1 (G151) : Turn torch left
G42.1 (G152) : Turn torch right
G40.1 (G150) : Cancel torch turning

These G codes are the same as for normal direction control. Whether to use these G codes for torch turning control or normal direction control is selected by the TRTENB (G0201#6) signal.

G0201#6 = 1 : Torch turning control
G0201#6 = 0 : Normal direction control

For details of this signal, see "Signal" described later.

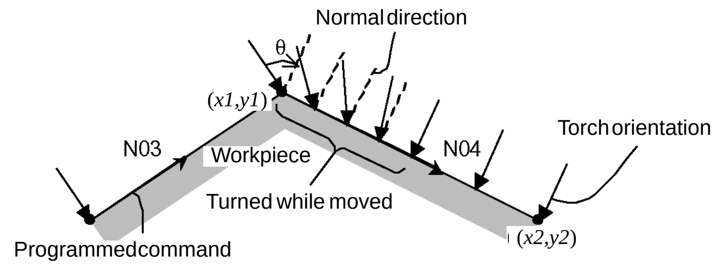
- **Usable interpolation commands**

Torch turning control can be used for the positioning and interpolation commands listed below. For other interpolation commands, torch turning control is not performed.

G00 : Positioning
G01 : Linear interpolation
G02, G03 : Circular interpolation

- **Operation**

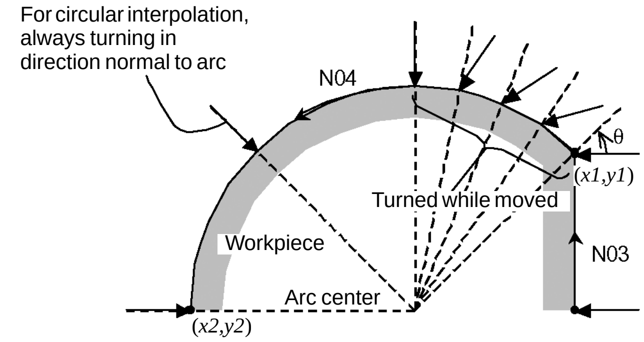
Torch turning control controls the bevel torch so that it is positioned in the normal direction to a specified straight line or arc without stopping the operation of the torch end at a point between adjacent blocks.
For a straight line, while linear interpolation is being performed, the rotary axis of the bevel torch is fed at a constant feedrate to position the rotary axis in the direction normal to the straight line.
For an arc, while circular interpolation is being performed, the rotary axis of the bevel torch is positioned in the normal direction to the arc, which changes from time to time.
In G01, G02, and G03 blocks, positioning to the normal direction in torch turning control is performed at a feedrate set in parameter No. 1430. In a G00 block, the positioning feedrate is set in parameter No. 1420. For positioning, the rotary axis always turns 180° or less (control for the shorter amount).
Examples of torch turning control operation are given in Figs. 19.10 (b) and 19.10 (c). For an acute–angled corner, create a machining program as shown in Fig. 19.10 (c).



(Sample program)

N01 G90
N02 G41.1 ← Turn torch left
N03 G01X$_{x1}$Y$_{y1}$
N04 G01X$_{x2}$Y$_{y2}$

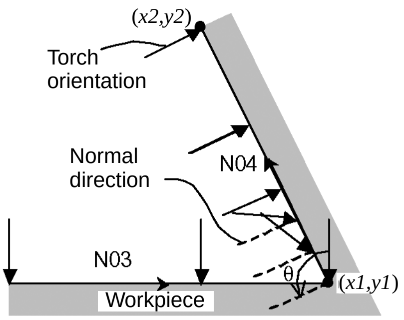
Fig. 19.10 (b) Turning the torch left



(Sample program)

N01 G90
N02 G42.1 ← Turn torch right
N03 G01X_{x1} Y_{y1}
N04 G03X_{x2} Y_{y2} R_{r2}

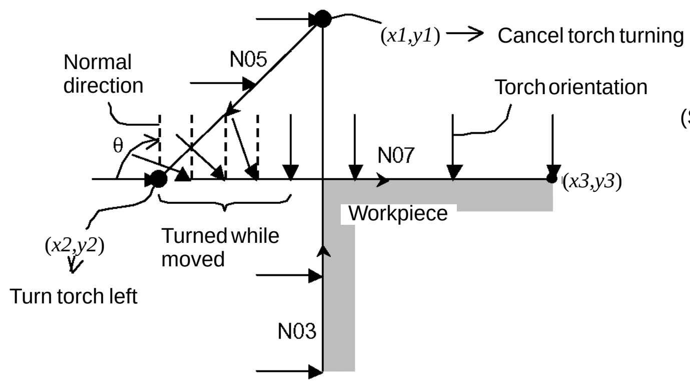
Fig. 19.10 (c) Turning the torch right



(Sample program)

N01 G90
N02 G41.1 ← Turn torch left
N03 G01X_{x1} Y_{y1}
N04 G01X_{x2} Y_{y2}

Fig. 19.10 (d) Turning the torch left (inside a corner)



(Sample program)

N01 G90
N02 G41.1 ← Turn torch left
N03 G01X_{x1} Y_{y1}
N04 G40.1 ← Cancel torch turning
N05 G01X_{x2} Y_{y2}
N06 G41.1 ← Turn torch left
N07 G01X_{x3} Y_{y3}

Fig. 19.10 (d) Turning the torch left (when turning is canceled in the middle)

Parameters

	#7	#6	#5	#4	#3	#2	#1	#0
1006								ROTx

[Data type] Bit axis

ROTx Setting linear or rotary axis.

0 : Linear axis
1 : Rotary axis

	#7	#6	#5	#4	#3	#2	#1	#0
1008								ROAx

[Data type] Bit axis

ROAx The rotary axis roll–over function is:

0 : Invalid
1 : Valid

1260	Amount of a shift per one rotation of a rotary axis

[Data type] 2–word axis

[Unit of data]

Input increment	IS–B	Unit
Unit of data	0.001	deg

[Valid data range] 1000 to 99999999

Set the amount of a shift per one rotation of a rotary asis.

1420	Rapid traverse rate for each axis

[Data type] 2–word axis

[Unit of data, valid data range]

Increment system	Unit of data	Valid data range
		IS–B
Millimeter machine	1 mm/min	30 to 240000
Inch machine	0.1 inch/min	30 to 96000
Rotary axis	1deg/min	30 to 240000

Set the rapid traverse rate when the rapid traverse override is 100% for each axis.

1430	Maximum cutting feedrate for each axis

[Data type] 2–word axis

[Unit of data, valid data range]

Increment system	Unit of data	Valid data range
		IS–B
Rotary axis	1deg/min	6 to 240000

Specify the maximum cutting feedrate for each axis.

A feedrate for each axis is clamped in cutting feed so that it does not exceed the maximum feedrate specified for each axis.

5483	Limit value of movement that is executed at the normal direction angle of a preceding block

[Data type] 2–word

[Unit of data]

Increment system	IS–B	Unit
Millimeter input	0.001	mm
Inch input	0.0001	inch

[Valid data range] 1 to 99999999

This parameter sets the limit value of movement at the normal direction angle of a preceding block.
For details, refer to the parameter manual.

5490	Axis number of a rotary axis for torch turning control

[Data type] Byte

[Unit of data] None

[Valid data range] 1 to the maximum number of controlled axes

Set the axis number of a rotary axis used for torch turning control.

	#7	#6	#5	#4	#3	#2	#1	#0
7052								NMI

[Data type] Bit axis

NMIx Always set to 1.
For the rotary axis used for torch turning control, be sure to set this bit to 1.

	#7	#6	#5	#4	#3	#2	#1	#0
15625					CNZ			

[Data type] Bit

CNZ In normal direction control and torch turning control, the orientation of the C–axis is determined as follows:

0 : The conventional specification is followed.
1 : When the normal direction control or torch turning control mode is entered, the angle of the C–axis is set so that the C–axis is in the normal direction.

Signal

- **Torch turning control enable signal TRTENB<G201#6>**

[Classification]

[Function] When this signal is driven to “1”, specifying G41.1 or G42.1 (G151 or G152) enables torch turning control.
When this signal is “0”, normal direction control is enabled.
This signal status can be changed only in the G40.1 (G150) mode.

	#7	#6	#5	#4	#3	#2	#1	#0
G201		TRTENB						

III. OPERATION

1 GENERAL

1.1 MANUAL OPERATION

Explanations

- **Manual reference position return**

The CNC machine nozzle has a position used to determine the machine position.
This position is called the reference position.
Generally, the nozzle is moved to the reference position immediately after the power is turned on. Moving the nozzle to the reference position using switches and buttons on the machine operator's panel is called manual reference position return. (See Section III–3.1.)

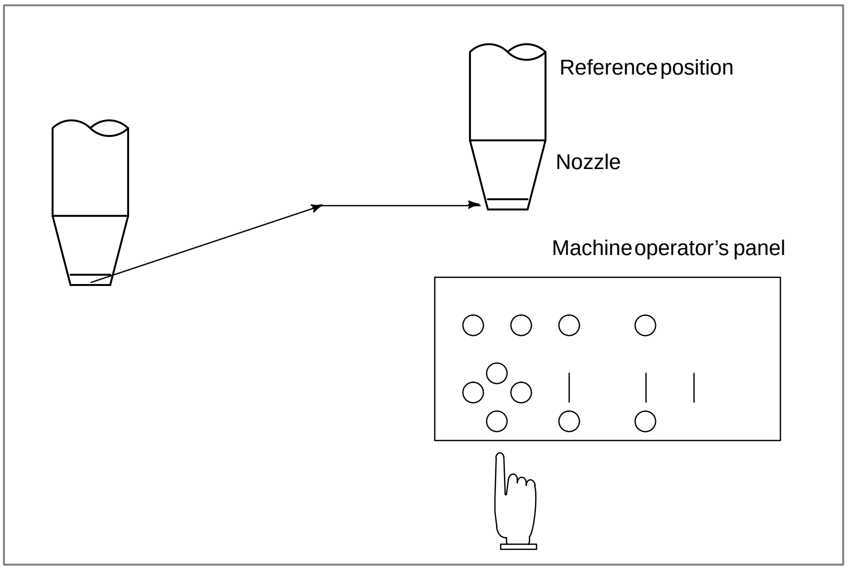


Fig.1.1 (a) Manual Reference Position Return

The nozzle can also be moved to the reference position using program commands.
This operation is called automatic reference position return. (See Section II–6.)

- **Moving the nozzle manually**

The machine operator's panel switches, buttons, and manual handle can be used to move the nozzle along each axis.

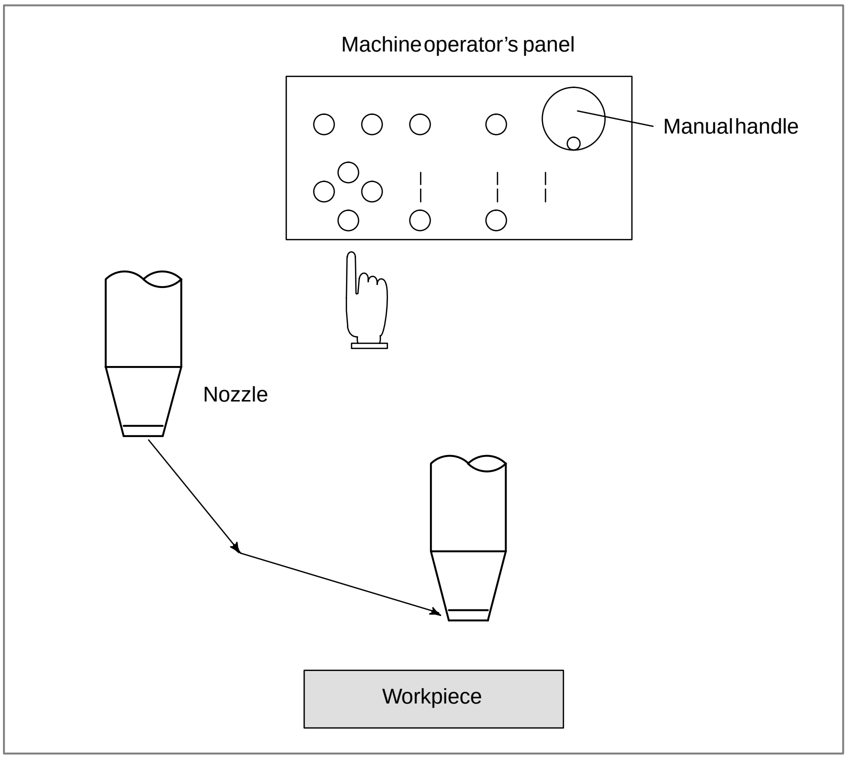


Fig.1.1 (b) Moving the Nozzle Manually

(i) Jog feed (See Section III–3.2.)
The nozzle moves continuously while a button is held down.

(ii) Incremental feed (See Section III–3.3.)
The nozzle moves by the predetermined distance each time a button is pressed.

(iii) Manual handle feed (See Section III–3.4.)
The nozzle moves by the distance corresponding to the degree of handle rotation when the manual handle is rotated.

1.2 MOVING THE NOZZLE USING A PROGRAM– AUTOMATIC OPERATION

Moving the machine according to a created program is called automatic operation.
Automatic operation includes memory and MDI operations. (See Section III–4.)

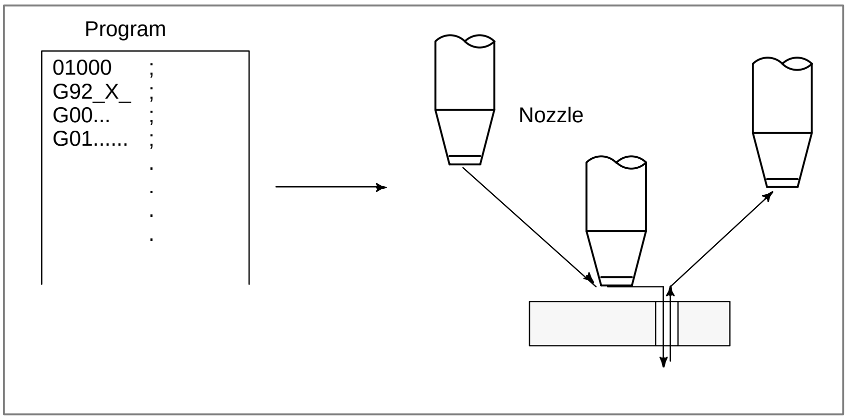


Fig.1.2 (a) Moving the Nozzle Using a Program

Explanations

- **Memory operation**

A program can be stored in CNC memory to enable operation of the machine according to the contents of the memory. This operation is called memory operation.

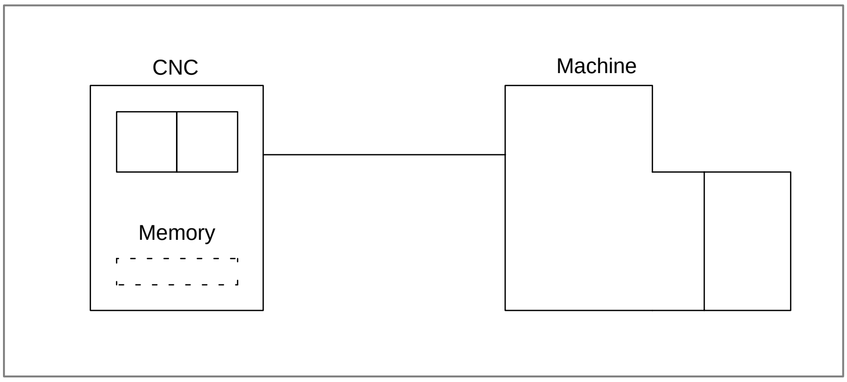


Fig.1.2 (b) Memory Operation

- **MDI operation**

A program can be specified from the MDI keyboard to operate the machine according to the commands in the program. This operation is called MDI operation.

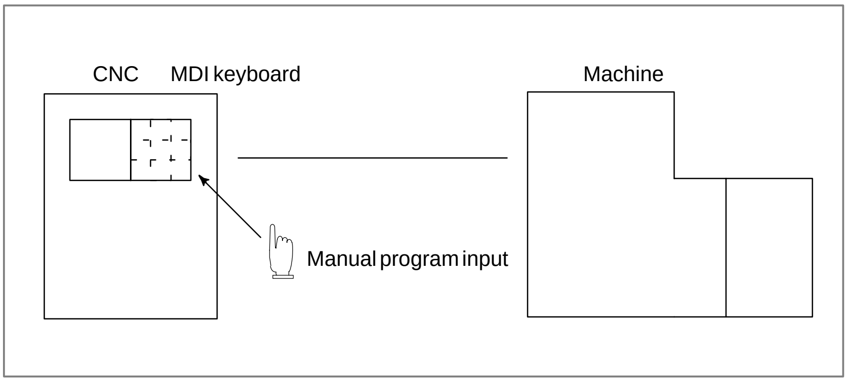


Fig.1.2 (c) MDI Operation

- **DNC operation**

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

1.3 OPERATIONS FOR AUTOMATIC OPERATION

Explanations

- **Selection of a program**

Select a program for the workpiece to be processed. Generally, one program is prepared for one workpiece. When multiple programs are stored on a tape or in memory, search the tape or memory for the program number (see Section III–9.3.) to select the program to be used.

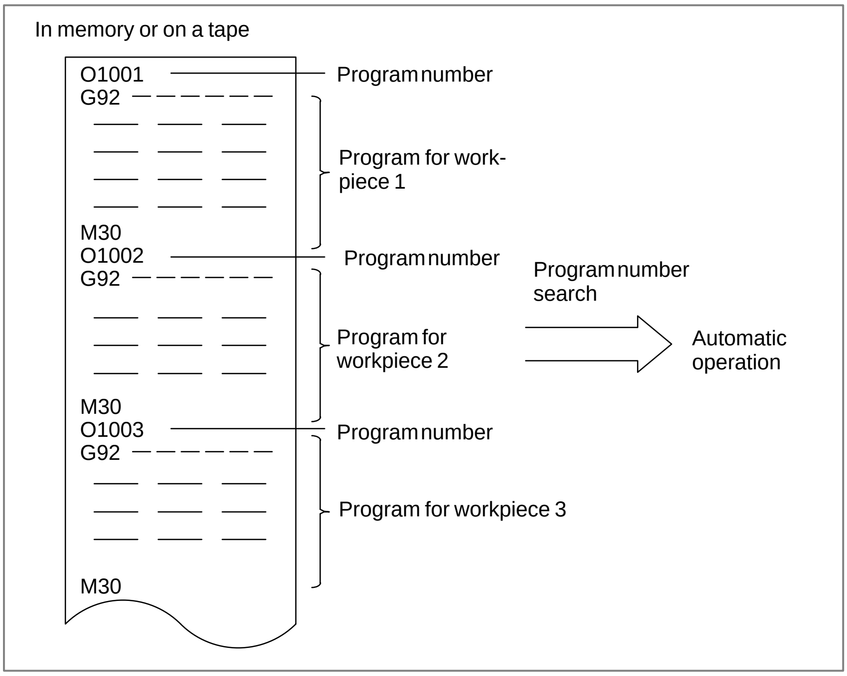


Fig.1.3 (a) Selecting a Program for Automatic Operation

- **Starting and stopping automatic operation**

Pressing the cycle start button starts automatic operation.
Pressing the feed hold or reset button halts or stops automatic operation. When the program stop or program end command is specified in the program, automatic operation stops before it is complete.
Once the process is complete, automatic operation stops. (See Section III–4.)

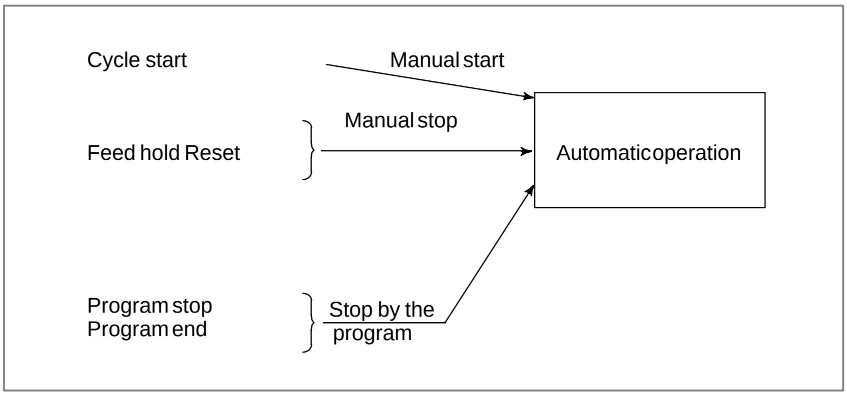


Fig.1.3 (b) Starting and Stopping Automatic Operation

- **Handle interruption**

The manual handle can be rotated during automatic operation to add the manual–feed amount to the automatic–feed amount for the nozzle movement. (See Section III–4.8.)

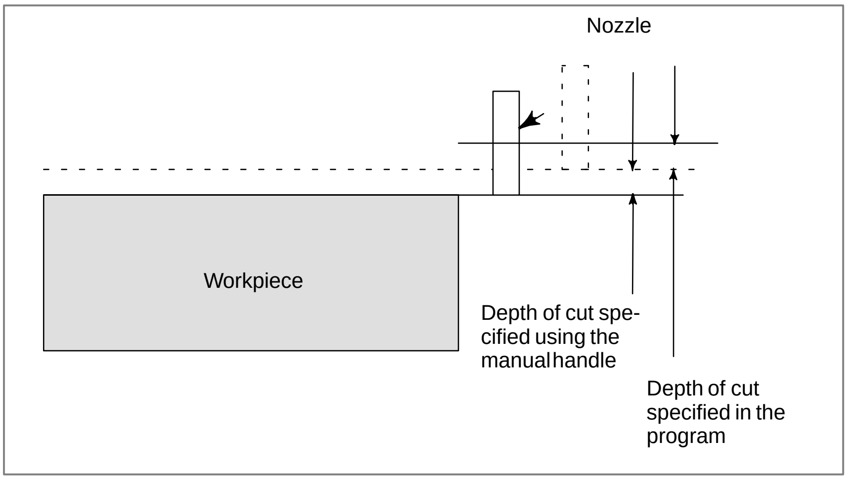


Fig.1.3 (c) Handle Interruption during Automatic Operation

1.4 TEST OPERATION

Before processing is started for a production run, automatic operation may be performed to check the created program to see whether the machine moves as desired.
This check can be made by running the machine or checking the current position display change without actually running the machine. (See Section III–5.)

1.4.1 Check by Running the Machine

Explanations

- **Dry run**

Remove the workpiece and check only the movement of the nozzle. Select the nozzle feedrate using the dial on the machine operator's panel. (See Section III–5.4.)

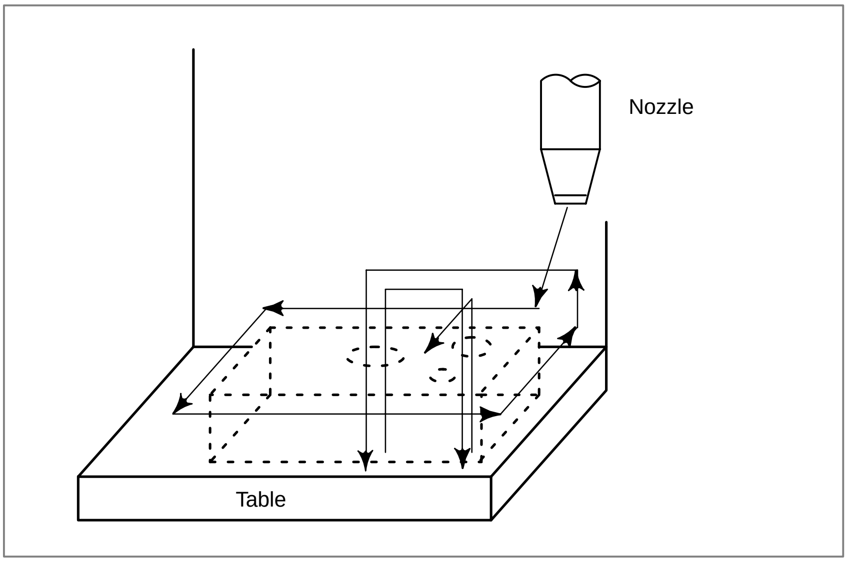


Fig.1.4.1 (a) Dry Run

- **Feedrate override**

Check the program by changing the program command. (See Section III–5.2.)

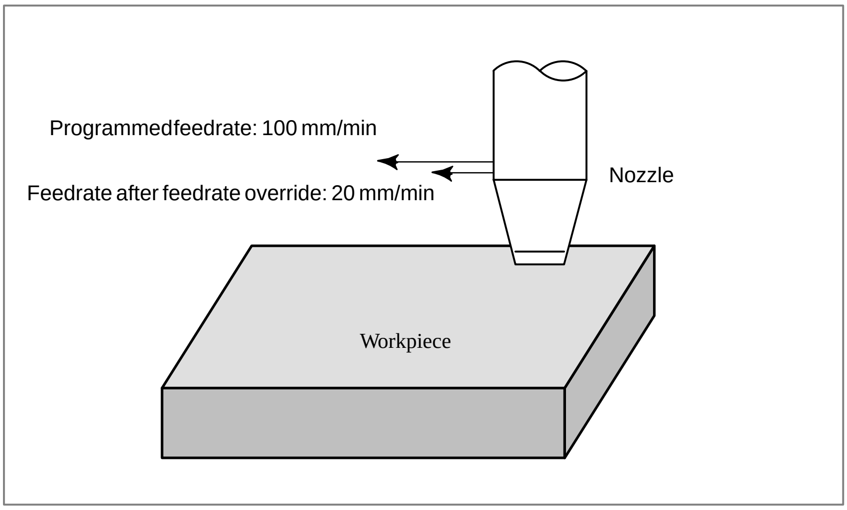


Fig.1.4.1 (b) Feedrate Override

- **Single block**

When the cycle start button is pressed, the nozzle performs one operation, then stops. When the cycle start button is pressed again, the nozzle performs the next operation, then stops. The program is checked in this way. (See Section III–5.5.)

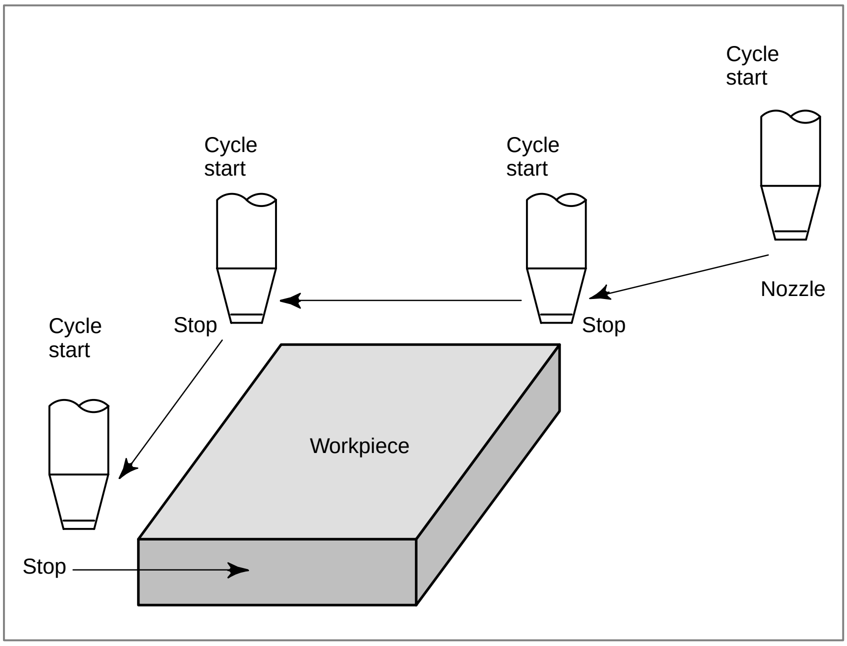


Fig.1.4.1 (c) Single Block

1.4.2 Checking the Position Display Change Without Running the Machine

Explanations

- **Machine lock**

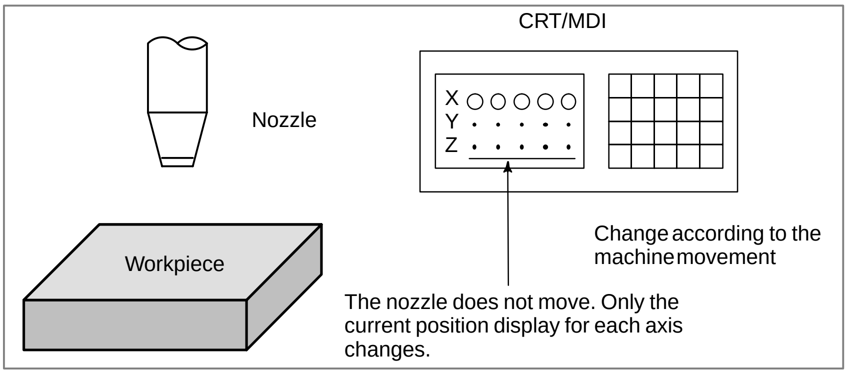


Fig.1.4.2 (a) Machine Lock

- **Auxiliary function lock**

When automatic operation is performed in auxiliary function lock mode as well as in machine lock mode (see Section III–5.1), all auxiliary functions are disabled. (See Section III–5.1.)

1.5 EDITING A PROGRAM

After a created program has been stored into memory, it can be corrected or modified from the MDI panel (see Section III–9).
This operation can be performed using the part program storage and edit function.

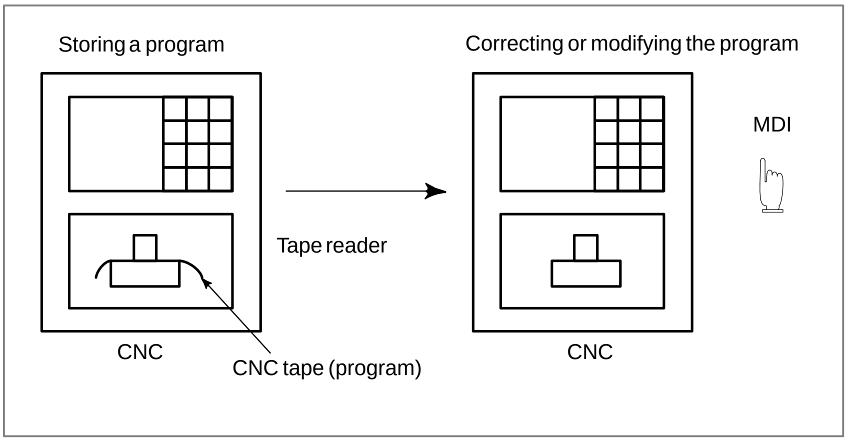


Fig.1.5 Editing a Program

1.6 DISPLAYING AND SETTING DATA

A new value can be set for the data stored in CNC internal memory on the screen by key operation, and memory data can be displayed on the screen. (See Chapter III–11.)

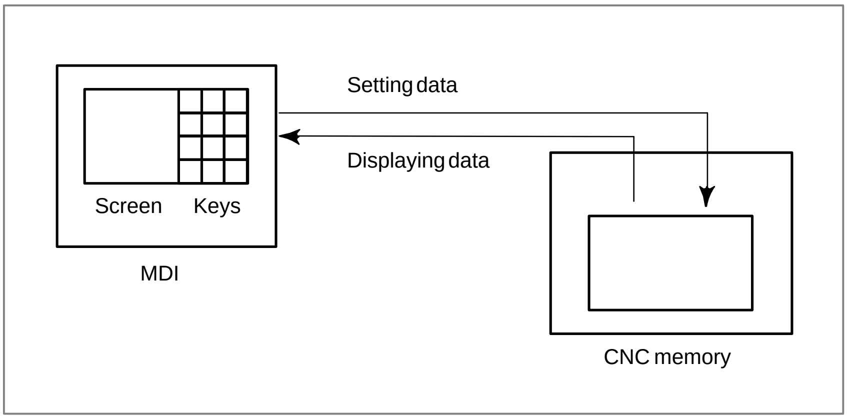


Fig.1.6 (a) Displaying and Setting Data

Explanations

- **Offset value**

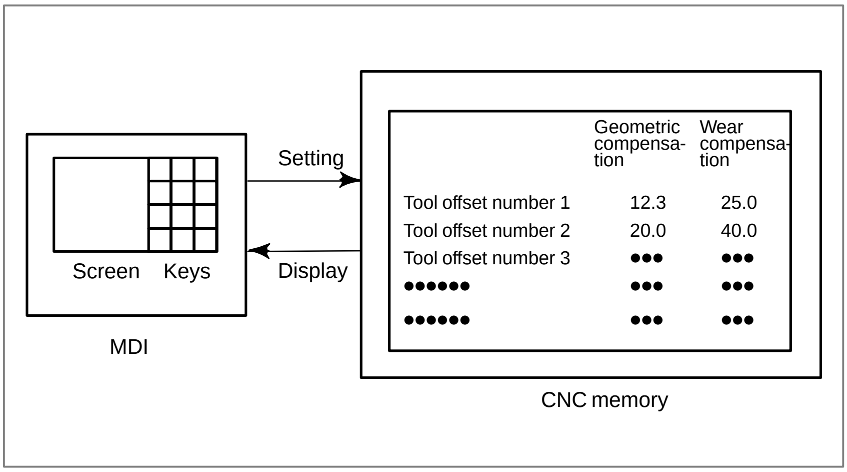


Fig.1.6 (b) Displaying and Setting Offset Values

The nozzle has nozzle dimensions (beam diameter and so on).
When a workpiece is processed into a given figure, the nozzle path depends on the processing conditions. By setting data related to the nozzle dimensions in CNC memory in advance, the beam path which allows each nozzle to be used for processing a workpiece into the figure specified in the program can automatically be generated internally for the same program under differing processing conditions.
Data related to nozzle dimensions is called the offset value. (See Section III–11.4.1.)

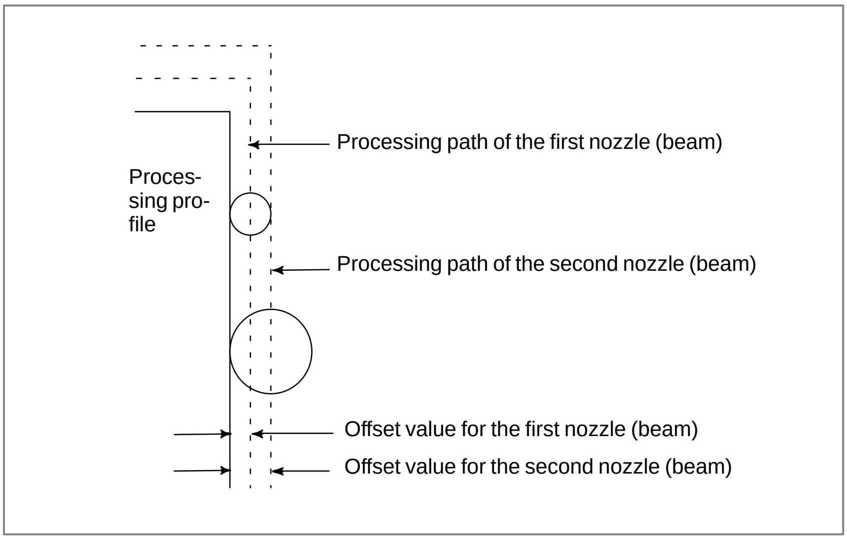


Fig.1.6 (c) Offset Values

- **Displaying and setting settings**

Aside from parameters, data is set by the operator during machine operation as required to change the machine characteristics.
For example, the following data can be set:

- Inch/metric conversion
- Data related to I/O devices
- Mirror image processing on/off

The above data is referred to as the settings. (See Section III–11.4.2.)

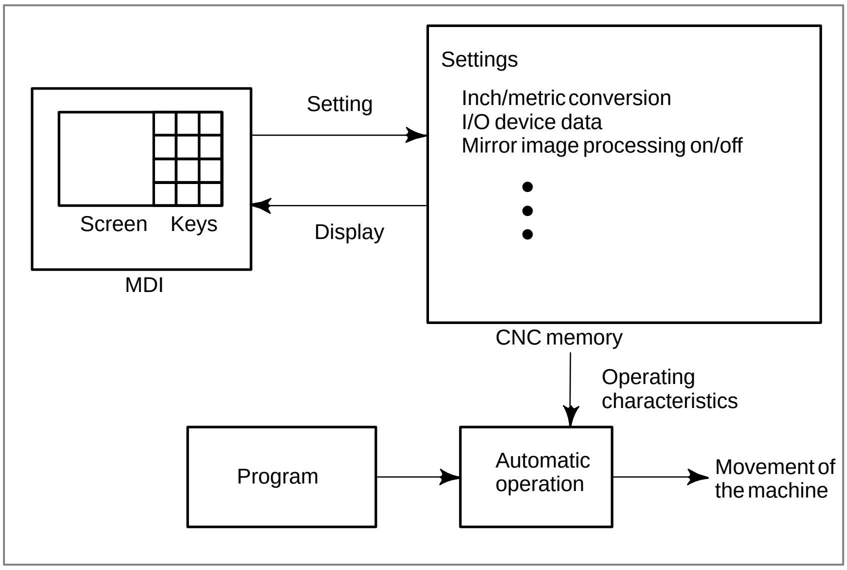


Fig.1.6 (d) Displaying and Setting Settings

- **Displaying and setting parameters**

The CNC functions are compatible with the characteristics of a wide variety of machines.
For example, the following items can be specified:

- The rapid traverse rate to be used for each axis
- Whether the metric or inch system is to be used for the least command increment.
- How command multiplication (CMR) and detected multiplication (DMR) is set

The data which determines the above items is referred to as parameters (see Section III–11.5.1).
Parameters are machine–specific and differ depending on the machine.

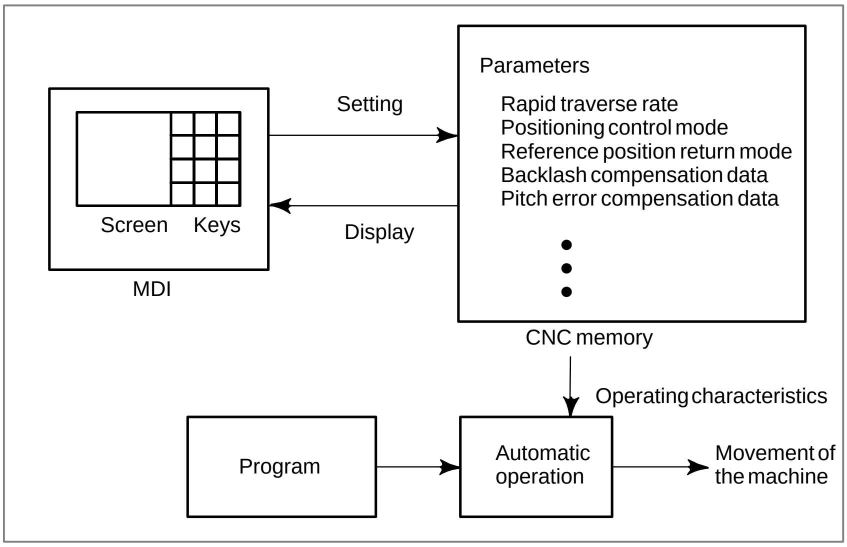


Fig.1.6 (e) Displaying and Setting Parameters

- **Data protection key**

A key can be set so that a program, offset value, parameter, or setting cannot be stored, modified, or deleted unintentionally.
This key is called a data protection key. (See Section III–11.)

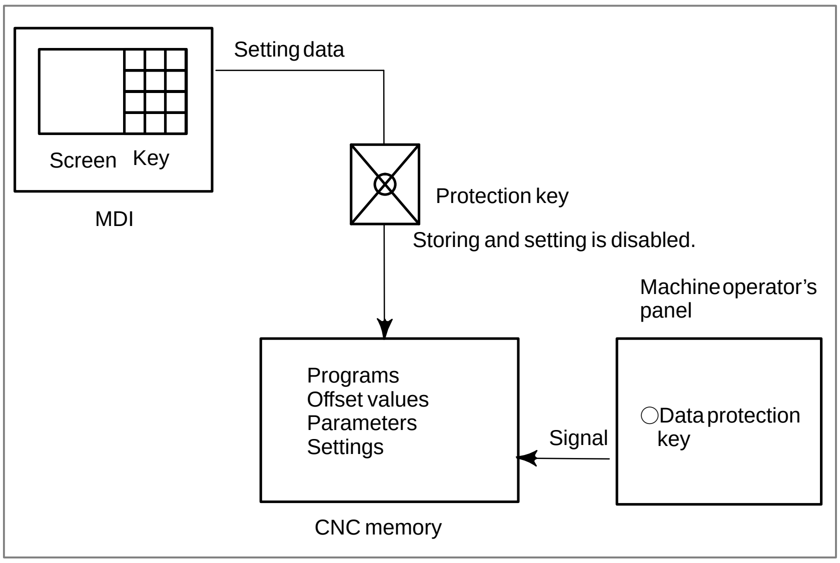
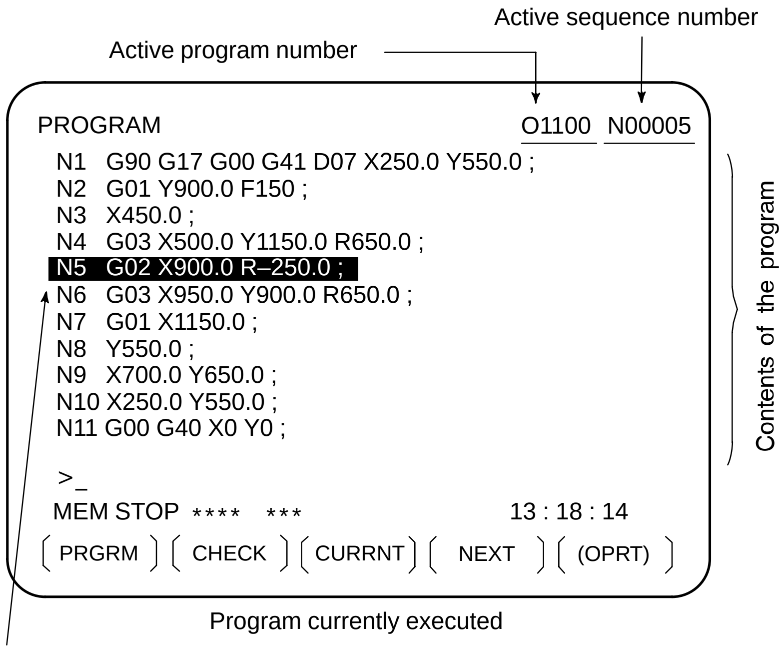


Fig.1.6 (f) Data Protection Key

1.7 DISPLAY

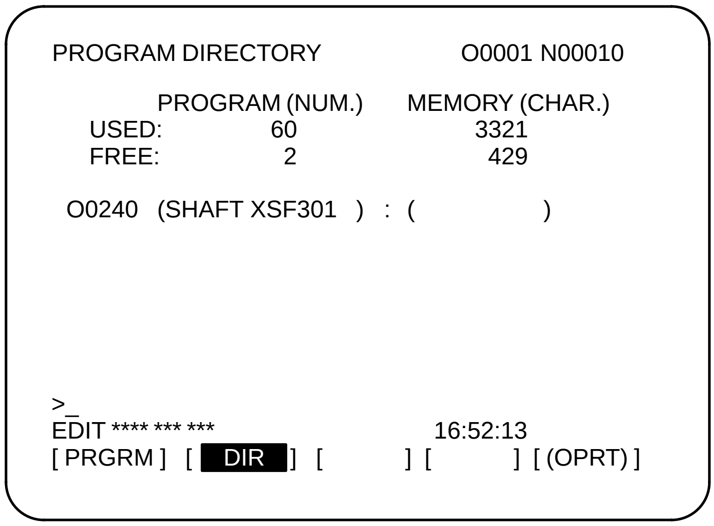
1.7.1 Program Display

The contents of the program currently being executed are displayed. The program to be executed next and a list of programs are also displayed. (See Section III–11.2.1.)



Program currently executed

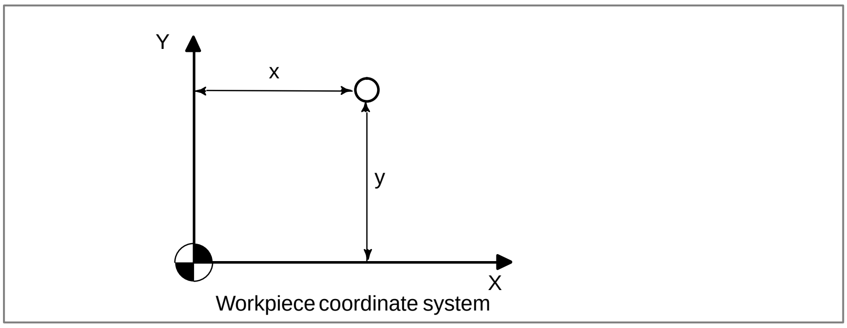
The cursor indicates the block that is currently being executed.

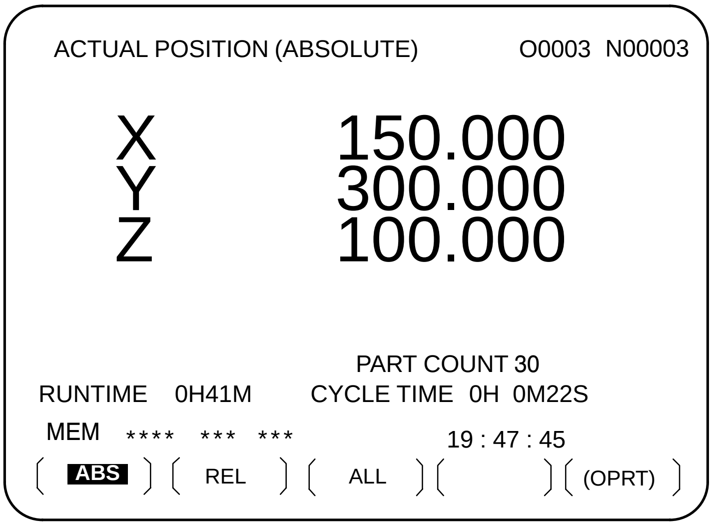


Program dilectory

1.7.2 Current Position Display

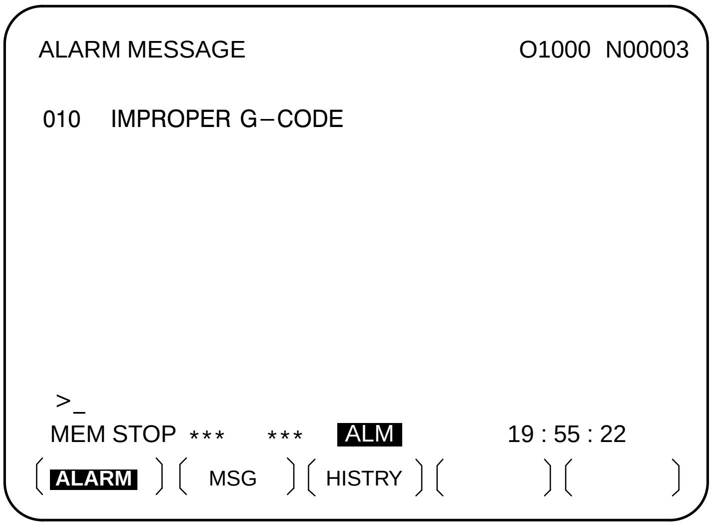
The current position of the nozzle is displayed with the coordinates in each coordinate system.
The distance from the current position to the target position can also be displayed. (See Sections III–11.1.1 to III–11.1.3.)





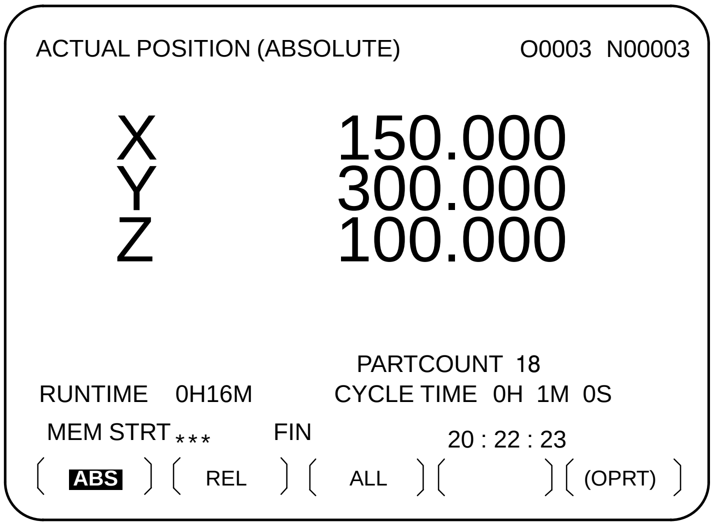
1.7.3 Alarm Display

If a problem occurs during the operation, the corresponding error code and alarm message are displayed on the screen. (See Section III–7.1.)
See Appendix G for details of the error codes.



1.7.4
Part Count Display and Run Time Display

When this option is selected, the following two items are displayed on the screen: Run time and part count. (See Section III–11.4.4.)



1.7.5
Graphic Display

Figures can be displayed on the following planes (see Section III–12):

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

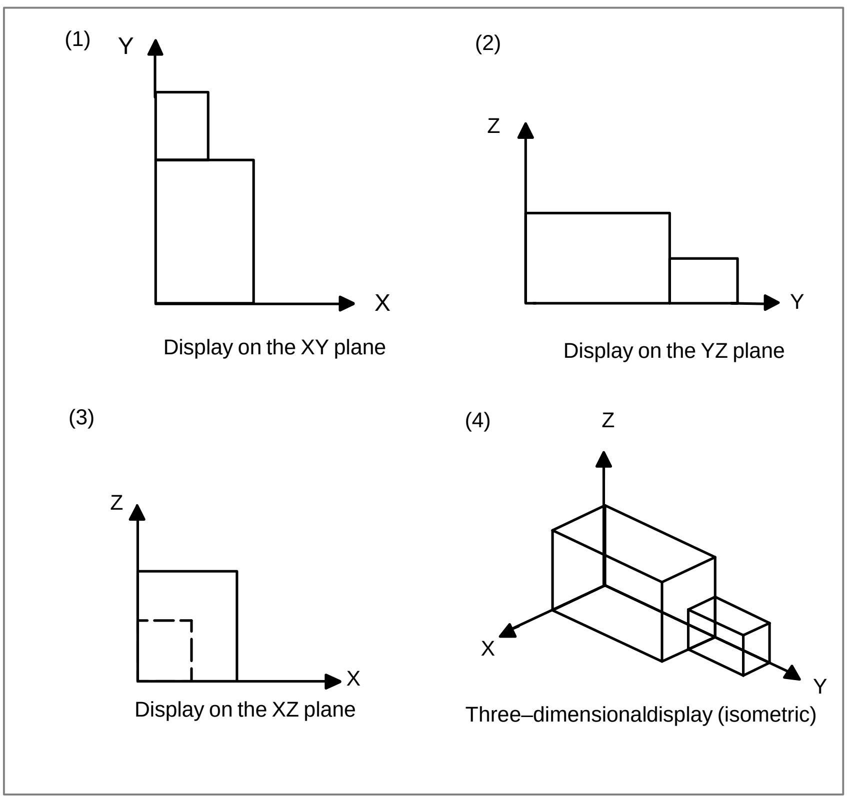


Fig.1.7.5 Graphic Display

1.8 DATA INPUT/OUTPUT

Programs, offset values, parameters, etc. input in CNC memory can be output to paper tape, cassette, or a floppy disk for saving. After once output to a medium, the data can be input into CNC memory.
(Refer to III–8)

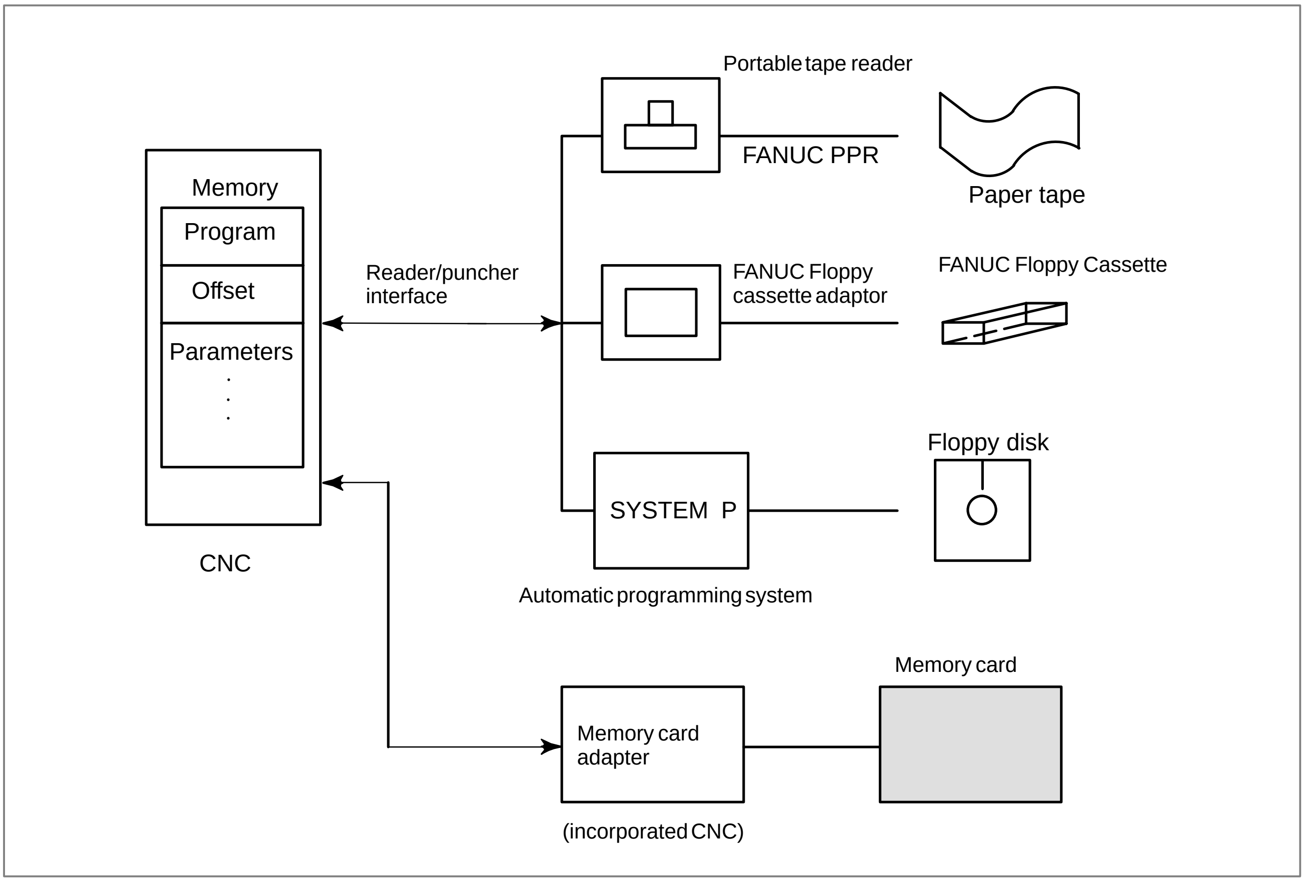


Fig.1.8 Data Output

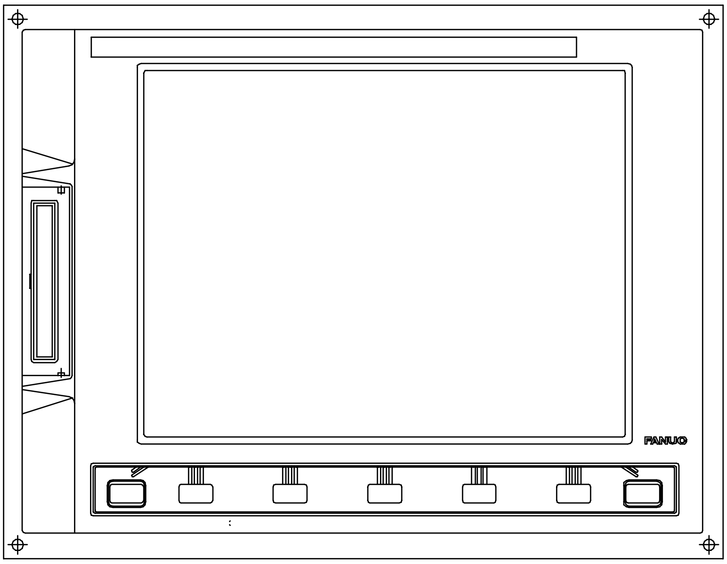
2 OPERATIONAL DEVICES

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

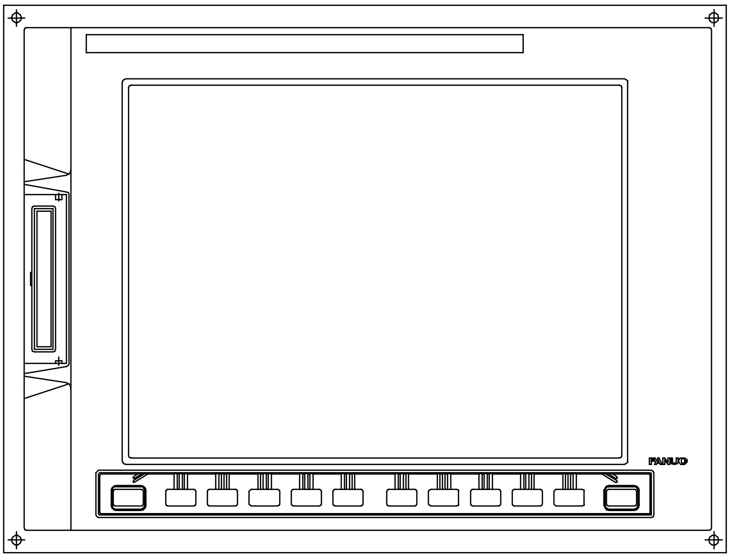
2.1 SETTING AND DISPLAY UNITS

The setting and display units are shown in Subsections 2.1.1 to 2.1.5 of Part III.

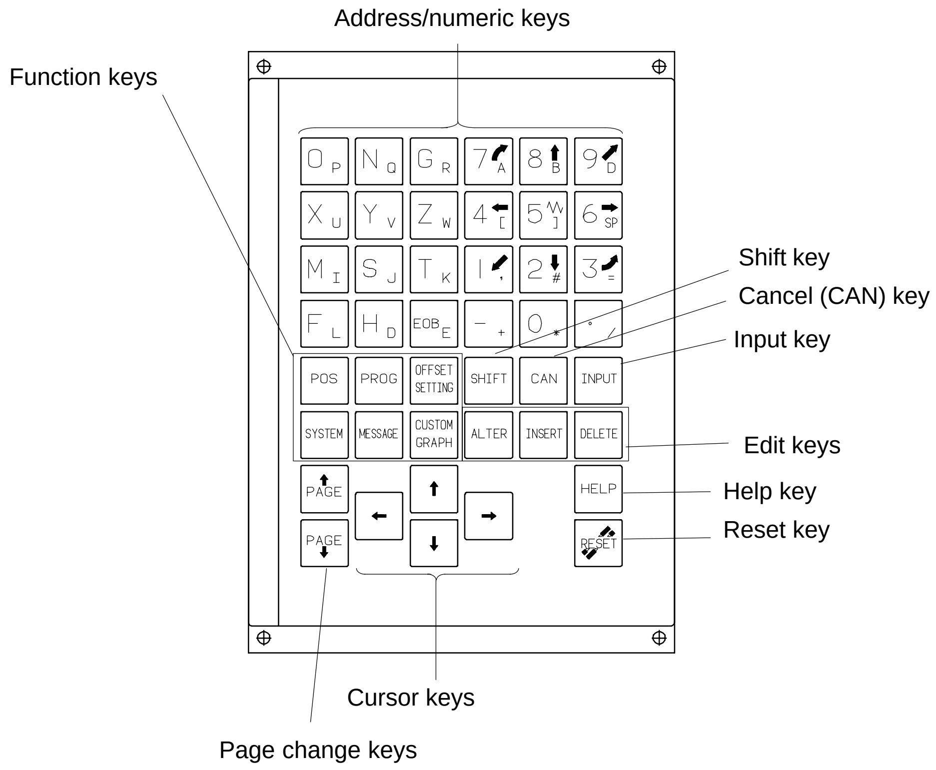
2.1.1 7.2″/8.4″ LCD–mounted Type CNC Control Unit



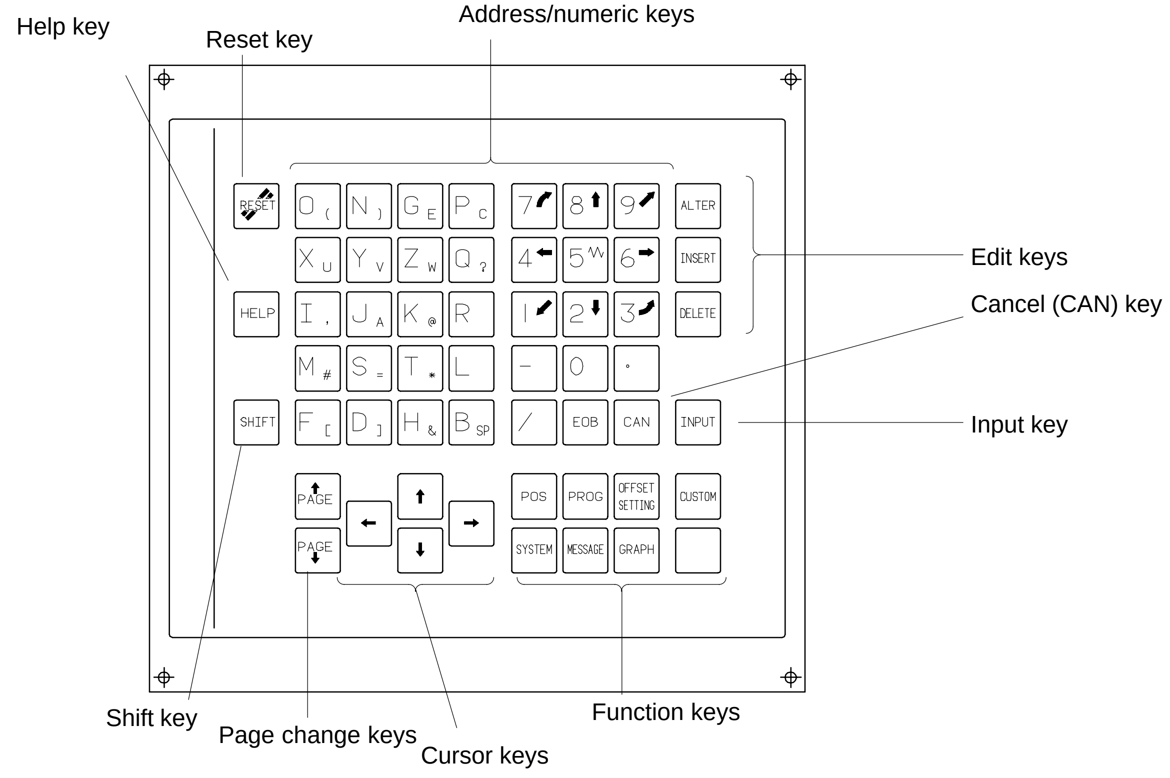
2.1.2 9.5″/10.4″ LCD–mounted Type CNC Control Unit



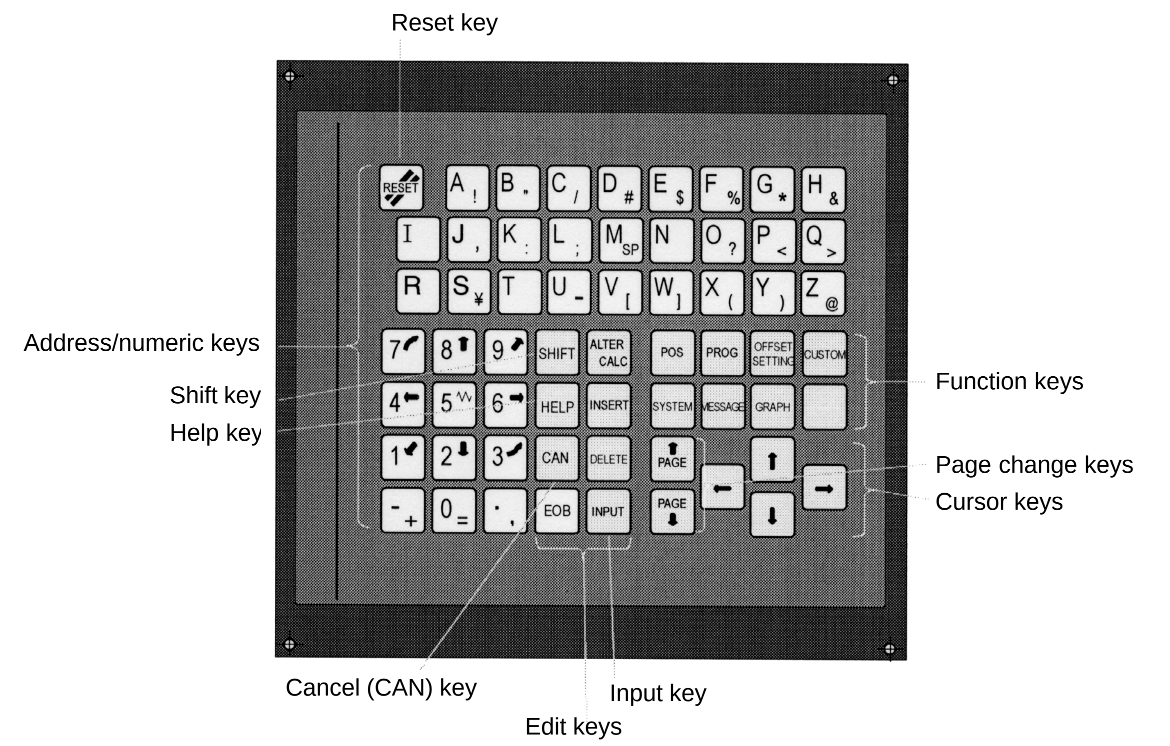
2.1.3 Stand–alone Type Small MDI Unit



2.1.4
Stand–alone Type Standard MDI Unit



2.1.5 Stand–alone Type 61 Fullkey MDI Unit



2.2 EXPLANATION OF THE KEYBOARD

Table 2.2 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). In case of 160*i*, this key is assigned to ”Esc” key of the personal computer.
3	Soft keys	The soft keys have various functions, according to the Applications. The soft key functions are displayed at the bottom of the screen.
4	Address and numeric keys 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 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 (In case of 160*i*, this key is assigned to ”Tab” key of the personal computer.) INSERT : Insertion DELETE : Deletion
9	Function keys POS PROG …	Press theses keys to switch display screens for each function. See III – 2.3 for detailas of the function keys.

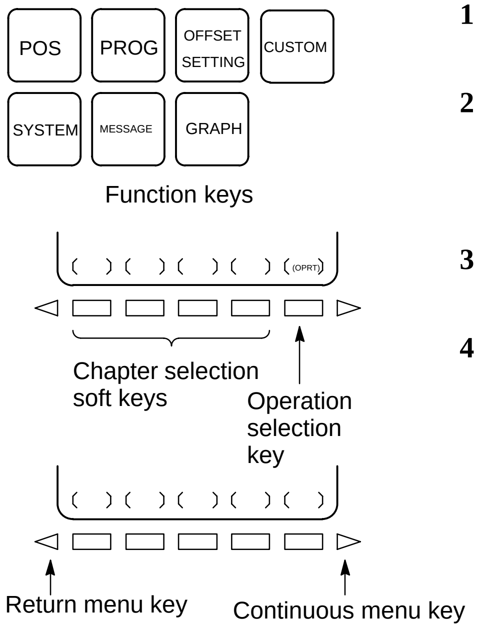
Table 2.2 Explanation of the MDI keyboard

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 screen in the forward direction. ↑ PAGE : This key is used to changeover the page on the screen in the reverse direction.

2.3 FUNCTION KEYS AND SOFT KEYS

The function keys are used to select the type of screen (function) to be displayed. When a soft key (section select soft key) is pressed immediately after a function key, the screen (section) corresponding to the selected function can be selected.

2.3.1 General Screen Operations



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

2 Press one of the chapter selection soft keys. The screen for the selected chapter appears. 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.

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.

2.3.2 Function Keys

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

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 and laser setting screen**.

SYSTEM

Press this key to display the **system screen**.

MESSAGE

Press this key to display the **message screen.**

GRAPH

Press this key to display the **graphics screen**.

CUSTOM

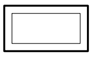
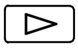
Press this key to display the **custom screen (conversational macro screen)**.
In case of 160*i*, this key is assigned to “Ctrl” key of the personal computer.

In case of 160*i*, this key is assigned to “Alt” key of the personal computer.

2.3.3 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 a soft key displayed in green.
	: Indicates the continuous menu key (rightmost soft key)(*3).

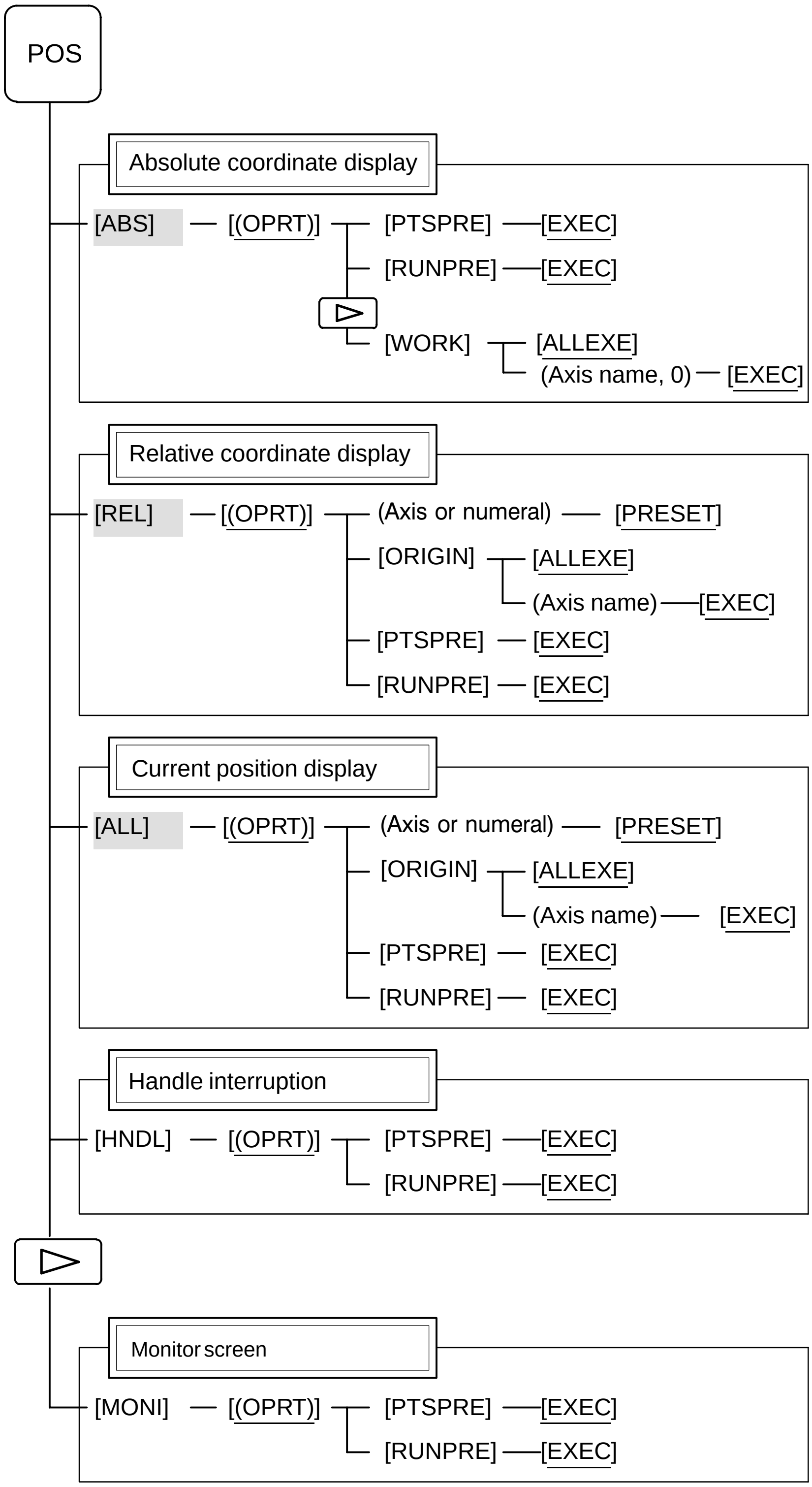
*1 Press function keys to switch between screens that are used frequently.

*2 Some soft keys are not displayed depending on the option configuration.

*3 In some cases, the continuous menu key is omitted when the 12 soft keys display unit is used.

POSITION SCREEN

Soft key transition triggered by the function key POS



PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MEM mode

1/2

PROG

Program display screen

- [PRGRM] (1) — [(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]
    - (N number) — [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]
    - (N number) — [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] — (number) — [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

1/2

PROG

Program display

- [PRGRM] — [(OPRT)]
  - [BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
  - (O number) — [O SRH]
  - (Address) — [SRH↓]
  - (Address) — [SRH↑]
  - [REWIND]
  - ▷
  - [F SRH]
    - [CAN]
    - (N number) — [EXEC]
  - [READ]
    - [CHAIN] (The cursor moves to the end of a program.)
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [PUNCH]
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [DELETE]
    - [CAN]
    - (N number) — [EXEC]
  - [EX–EDT]
    - [COPY]
      - [CRSR~] / [~CRSR] / [~BTTM] / [ALL] — (O number) — [EXEC]
    - [MOVE]
      - [CRSR~] / [~CRSR] / [~BTTM] / [ALL] — (O number) — [EXEC]
    - [MERGE]
      - [~CRSR] / [~BTTM] — (O number) — [EXEC]
    - [CHANGE] — (Address) — [BEFORE] — (Address) — [AFTER]
      - [SKIP]
      - [1–EXEC]
      - [EXEC]

(1)(Continued on the next page)

2/2

(1)

Program directory display

- [LIB] — [(OPRT)]
  - [BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
  - (O number) — [O SRH] ⇒ Return to the program
  - ▷
  - [READ]
    - [CHAIN]
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [PUNCH]
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]

▷

Floppy directory display

- [FLOPPY]
  - [PRGRM] ⇒ Return to the program
  - [DIR] — [(OPRT)]
    - [F SRH]
      - (number) — [F SET]
      - [CAN]
      - [EXEC]
    - [READ]
      - (number) — [F SET]
      - (O number) — [O SET]
      - [STOP]
      - [CAN]
      - [EXEC]
    - [PUNCH]
      - (number) — [F SET]
      - (O number) — [O SET]
      - [STOP]
      - [CAN]
      - [EXEC]
    - [DELETE]
      - (number) — [F SET]
      - [CAN]
      - [EXEC]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MDI 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"
- (Address) — [SRH↓]
- (Address) — [SRH↑]
- [REWIND]

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 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”
- (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)

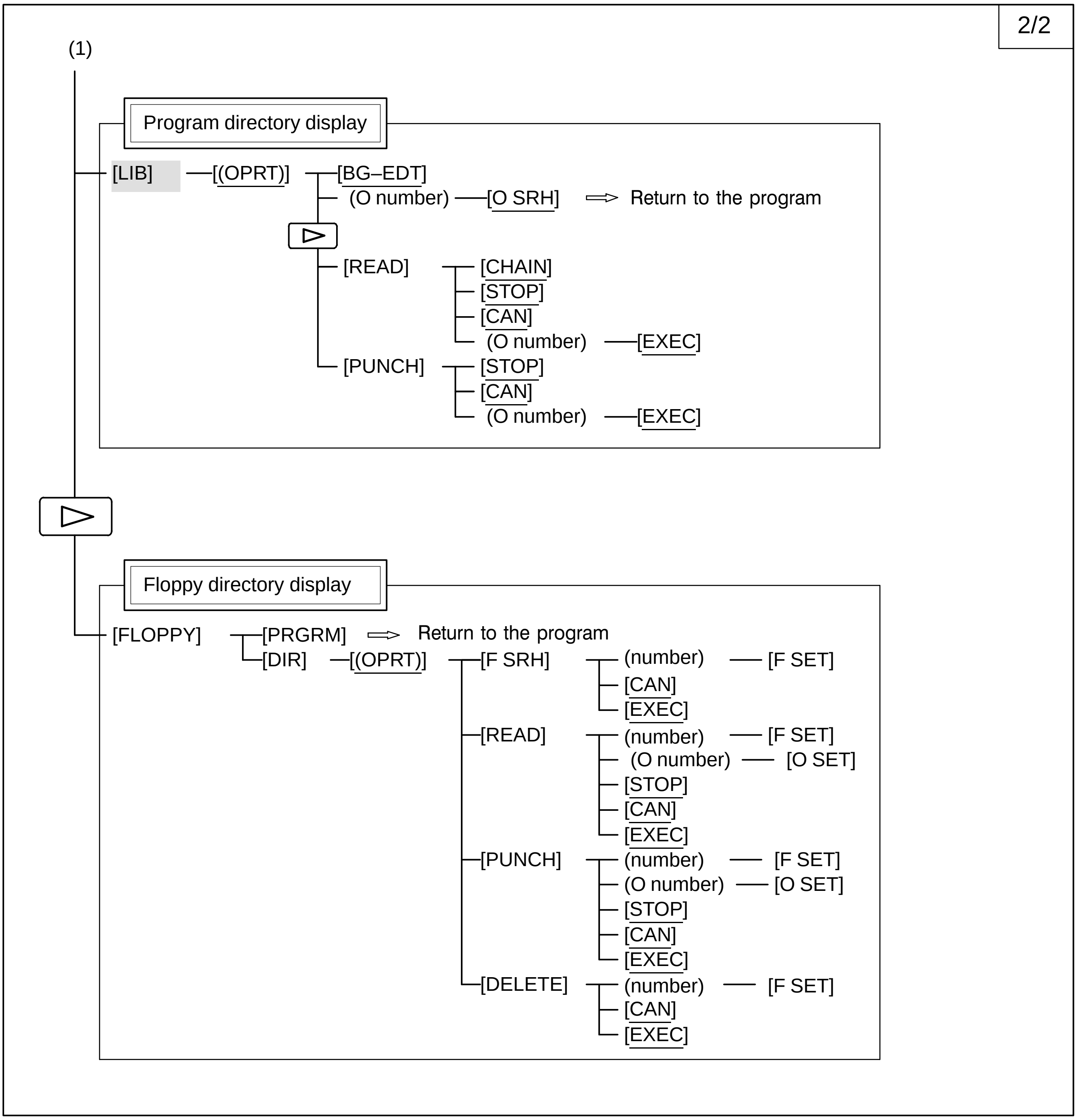
1/2

PROG

Program display

- [PRGRM] — [(OPRT)]
  - [BG–END]
  - (O number) — [O SRH]
  - (Address) — [SRH↓]
  - (Address) — [SRH↑]
  - [REWIND]
  - ▷
  - [F SRH]
    - [CAN]
    - (N number) — [EXEC]
  - [READ]
    - [CHAIN] (The cursor moves to the end of a program.)
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [PUNCH]
    - [STOP]
    - [CAN]
    - (O number) — [EXEC]
  - [DELETE]
    - [CAN]
    - (N number) — [EXEC]
  - [EX–EDT]
    - [COPY]
      - [CRSR~] / [~CRSR] / [~BTTM] / [ALL] — (O number) — [EXEC]
    - [MOVE]
      - [CRSR~] / [~CRSR] / [~BTTM] / [ALL] — (O number) — [EXEC]
    - [MERGE]
      - [~CRSR] / [~BTTM] — (O number) — [EXEC]
    - [CHANGE] — (Address) — [BEFORE] — (Address) — [AFTER]
      - [SKIP]
      - [1–EXEC]
      - [EXEC]

(1)(Continued on the next page)


2/2
(1)
Program directory display
[LIB]
[(OPRT)]
[BG–EDT]
(O number)
[O SRH]
Return to the program
[READ]
[CHAIN]
[STOP]
[CAN]
(O number)
[EXEC]
[PUNCH]
[STOP]
[CAN]
(O number)
[EXEC]
Floppy directory display
[FLOPPY]
[PRGRM]
Return to the program
[DIR]
[(OPRT)]
[F SRH]
(number)
[F SET]
[CAN]
[EXEC]
[READ]
(number)
[F SET]
(O number)
[O SET]
[STOP]
[CAN]
[EXEC]
[PUNCH]
(number)
[F SET]
(O number)
[O SET]
[STOP]
[CAN]
[EXEC]
[DELETE]
(number)
[F SET]
[CAN]
[EXEC]

OFFSET/SETTING SCREEN

Soft key transition triggered by the function key OFFSET SETTING

1/2

OFFSET SETTING

Tool offset screen

- [OFFSET] — [(OPRT)]
  - (Number) — [NO SRH]
  - (Axis name) — [INP.C.]
  - (Numeral) — [+INPUT]
  - (Numeral) — [INPUT]
  - ▷
  - [CLEAR]
    - [ALL]
    - [WEAR]
    - [GEOM]
  - [READ]
    - [CAN]
    - [WEAR]
  - [PUNCH]
    - [CAN]
    - [EXEC]

Setting screen

- [SETING] — [(OPRT)]
  - (Number) — [NO SRH]
  - [ON:1]
  - [OFF:0]
  - (Numeral) — [+INPUT]
  - (Numeral) — [INPUT]

Workpiece coordinate system setting screen

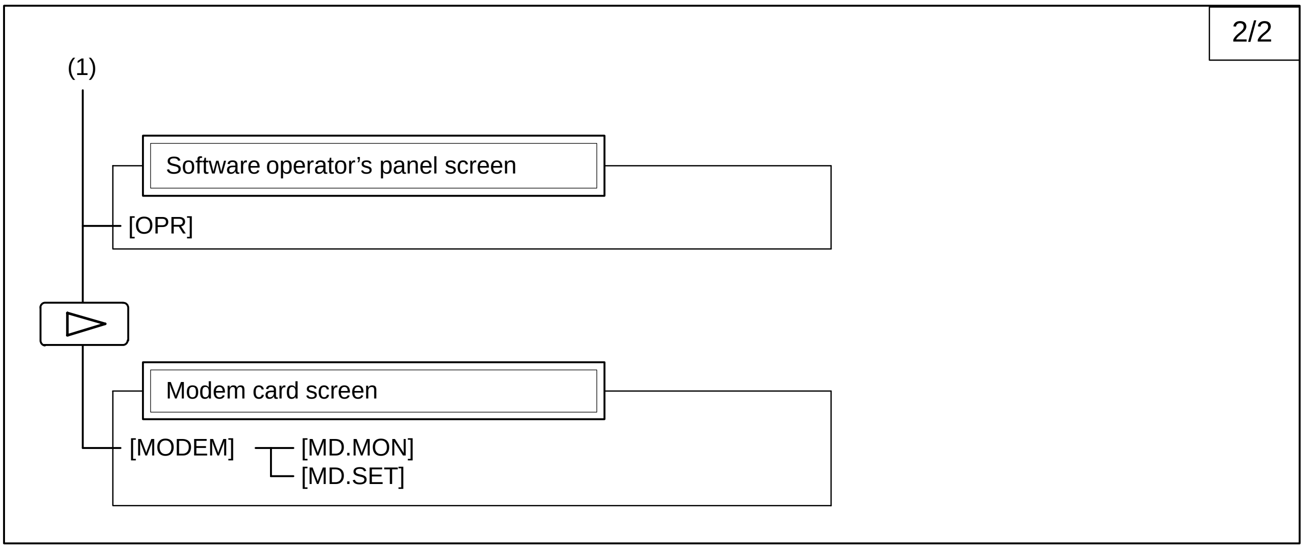
- [WORK] — [(OPRT)]
  - (Number) — [NO SRH]
  - (Axis, Numeral) — [MEASUR]
  - (Numeral) — [INPUT]
  - (Numeral) — [INPUT]

▷

Macro variables display screen

- [MACRO] — [(OPRT)]
  - (Number) — [NO SRH]
  - (Axis name) — [INP.C.]
  - (Numeral) — [INPUT]
  - ▷
  - [PUNCH]
    - [CAN]
    - [EXEC]

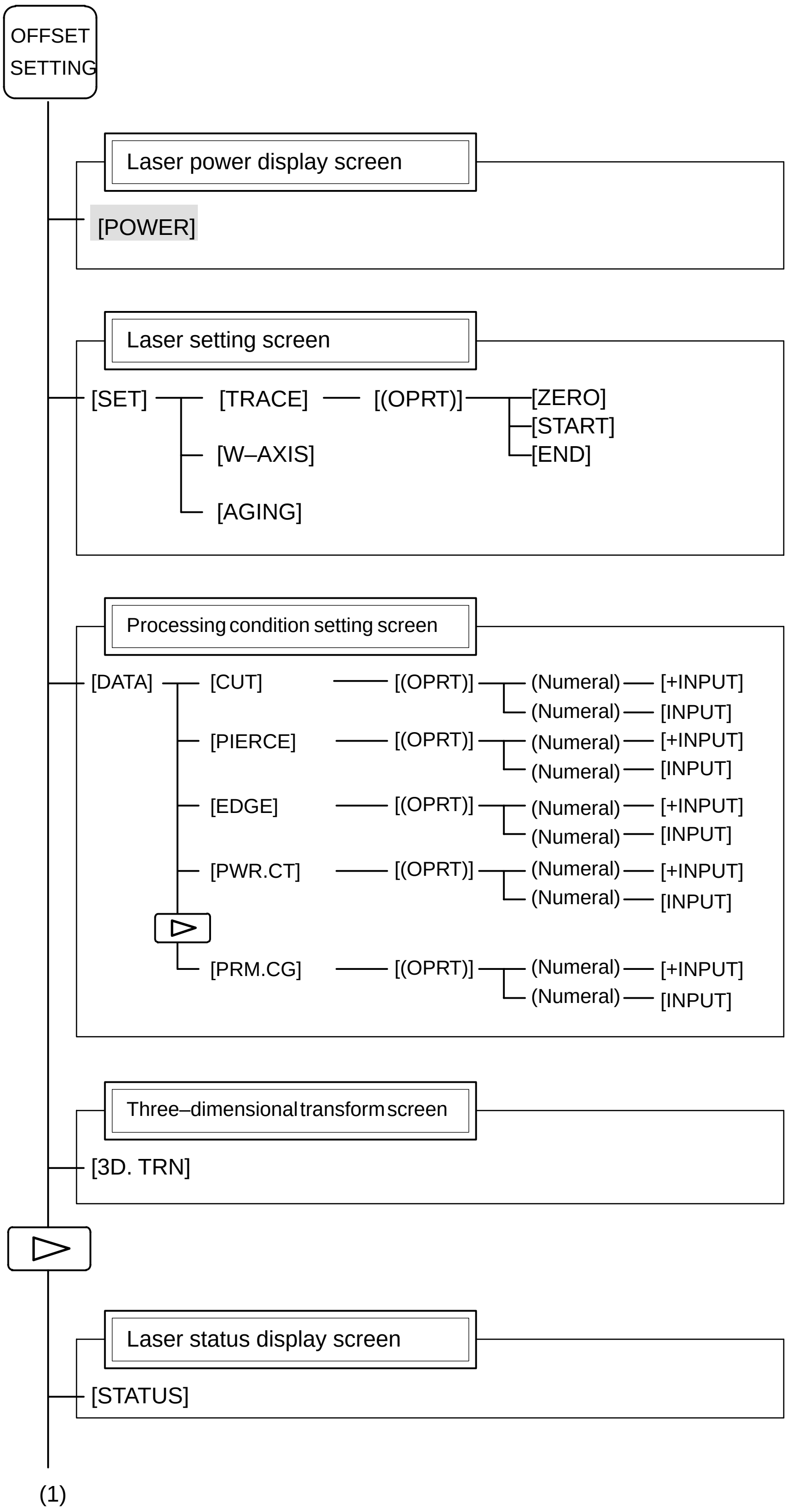
(1)

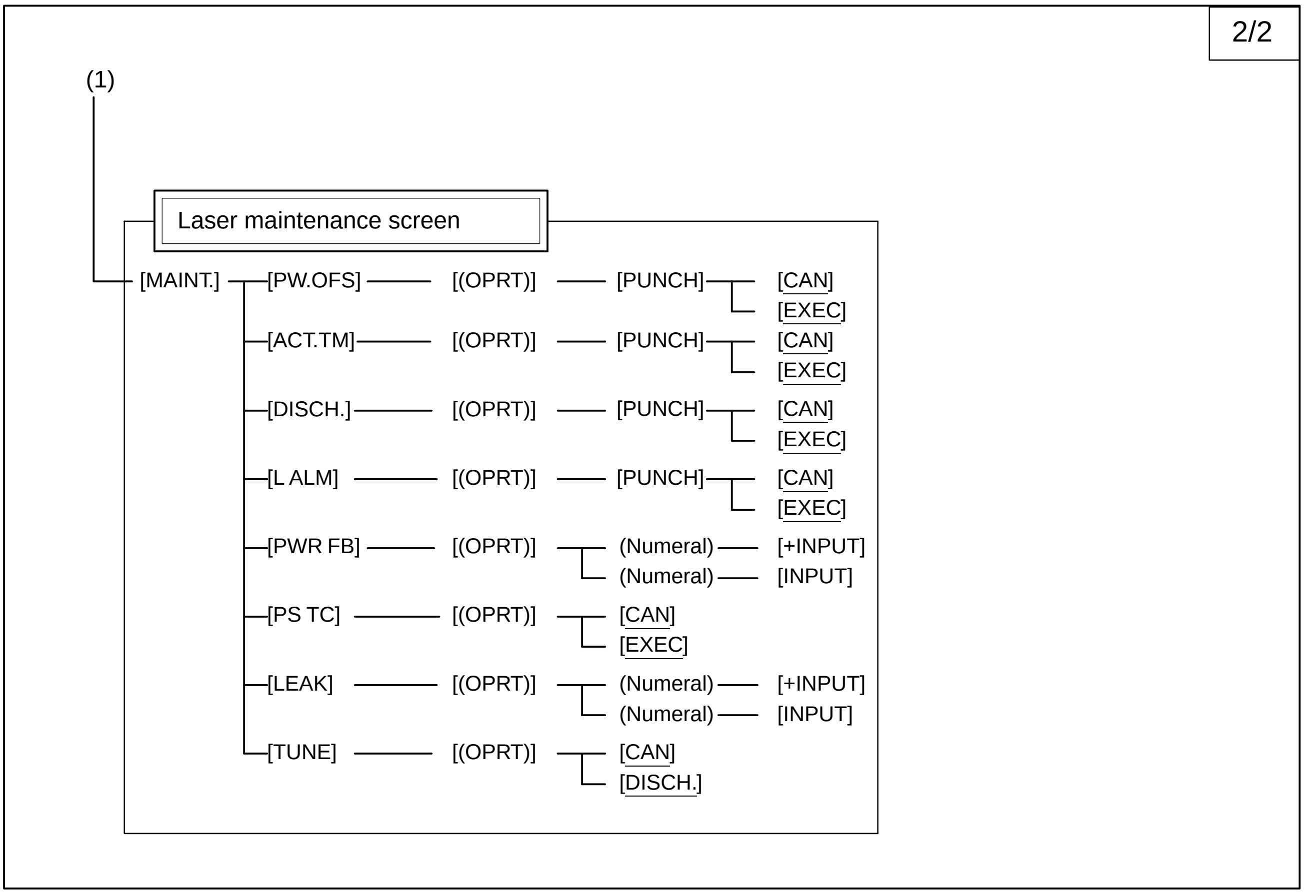

2/2
(1)
Software operator’s panel screen
[OPR]
Modem card screen
[MODEM]
[MD.MON]
[MD.SET]

LASER SETTING SCREEN

Soft key OFFSET SETTING transition triggered by the function key

1/2




2/2
(1)
Laser maintenance screen
[MAINT.]
[PW.OFS]
[(OPRT)]
[PUNCH]
[CAN]
[EXEC]
[ACT.TM]
[(OPRT)]
[PUNCH]
[CAN]
[EXEC]
[DISCH.]
[(OPRT)]
[PUNCH]
[CAN]
[EXEC]
[L ALM]
[(OPRT)]
[PUNCH]
[CAN]
[EXEC]
[PWR FB]
[(OPRT)]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]
[PS TC]
[(OPRT)]
[CAN]
[EXEC]
[LEAK]
[(OPRT)]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]
[TUNE]
[(OPRT)]
[CAN]
[DISCH.]

SYSTEM SCREEN Soft key transition triggered by the function key SYSTEM

1/2

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]

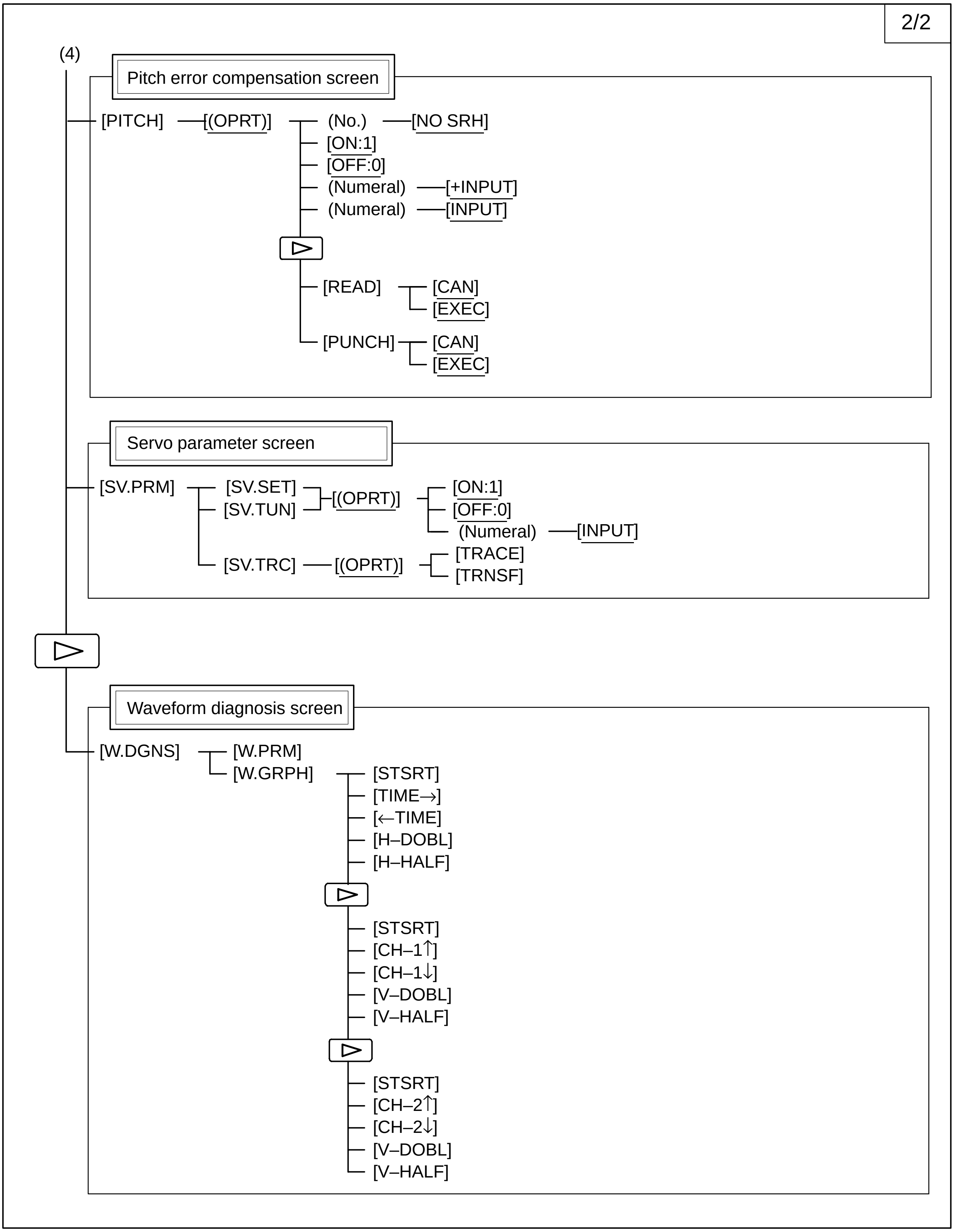
System configuration screen

- [SYSTEM]

▷

(4)

(Continued on the next page)


2/2
(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]
[SV.TRC]
[(OPRT)]
[TRACE]
[TRNSF]
Waveform diagnosis screen
[W.DGNS]
[W.PRM]
[W.GRPH]
[STSRT]
[TIME→]
[←TIME]
[H–DOBL]
[H–HALF]
[STSRT]
[CH–1↑]
[CH–1↓]
[V–DOBL]
[V–HALF]
[STSRT]
[CH–2↑]
[CH–2↓]
[V–DOBL]
[V–HALF]

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

[ALARM] —[(OPRT)] —[SELECT]

Operation method screen

[OPRAT] —[(OPRT)] —[SELECT]

Parameter table screen

[PARAM]

GRAPHIC SCREEN

Soft key transition triggered by the function key GRAPH

Tool path graphics

GRAPH

Beam path graphics

- [PARAM]
- [EXEC] —— [(OPRT)]
  - [AUTO]
  - [STSRT]
  - [STOP]
  - [REWIND]
  - [CLEAR]
- [ZOOM] —— [(OPRT)]
  - [EXEC]
  - [←]
  - [→]
  - [↑]
  - [↓]
- [POS]

Solid graphics

GRAPH

Solid graphics

- [PARAM]
- [BLANK] —— [(OPRT)]
  - [ANEW]
  - [+ROT]
  - [–ROT]
  - [+TILT]
  - [–TILT]
- [EXEC] —— [(OPRT)]
  - [A.ST]
  - [F.ST]
  - [STOP]
  - [REWIND]
- [REVIEW] —— [(OPRT)]
  - [ANEW]
  - [+ROT]
  - [–ROT]
  - [+TILT]
  - [–TILT]

▷

- [PARAM]
- [3–PLN] —— [(OPRT)]
  - [∩]
  - [←]
  - [→]
  - [↑]
  - [↓]

2.3.4 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.

```
Key input buffer display  --->  > N001X100Z_
                                EDIT             ALM 12:35:45
                                [      ] [      ] [      ] [      ] [      ]
```

Fig.2.3.4 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.

2.3.5 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.

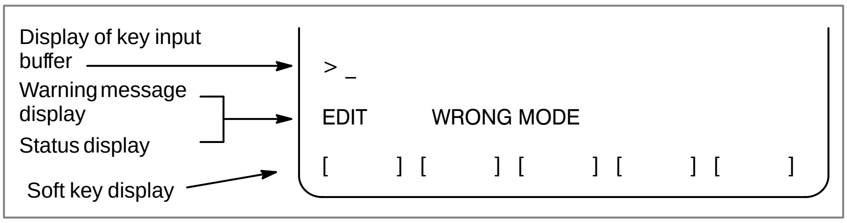


Fig.2.3.5 Warning message display

Table 2.3.5 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 input value 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.

2.3.6 Soft Key Configuration

There are 12 soft keys in the 10.4″LCD/MDI or 9.5″LCD/MDI. As illustrated below, the 5 soft keys on the right and those on the right and left edges operate in the same way as the 7.2″LCD or 8.4″ LCD, whereas the 5 keys on the left hand side are expansion keys dedicated to the 10.4″LCD or 9.5″LCD.

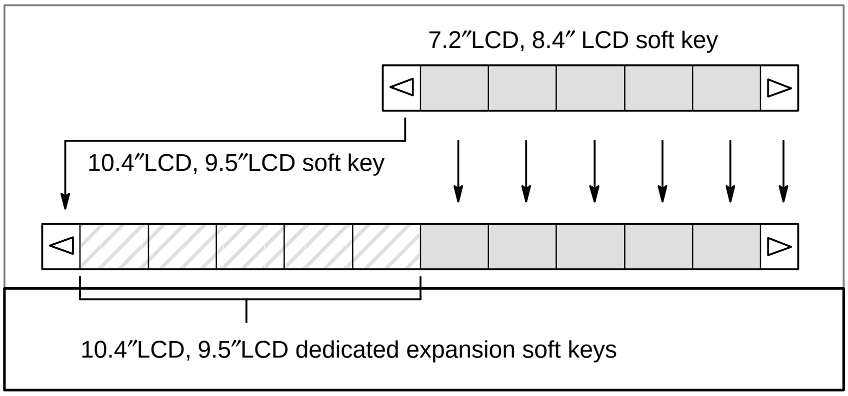
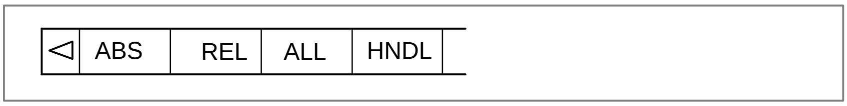


Fig.2.3.6 LCD soft key configuration

Whenever a position display appears in the left half of the screen after a function key other than POS is pressed, the soft keys on the left half of the soft key display area are displayed as follows:



The soft key corresponding to the position display is indicated in reverse video.
This manual may refer to 10.4″and 9.5″LCD display units as 12 soft key types, and 7.2″ and 8.4″ LCD display units as 7 soft key types.

2.4 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 2.4 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

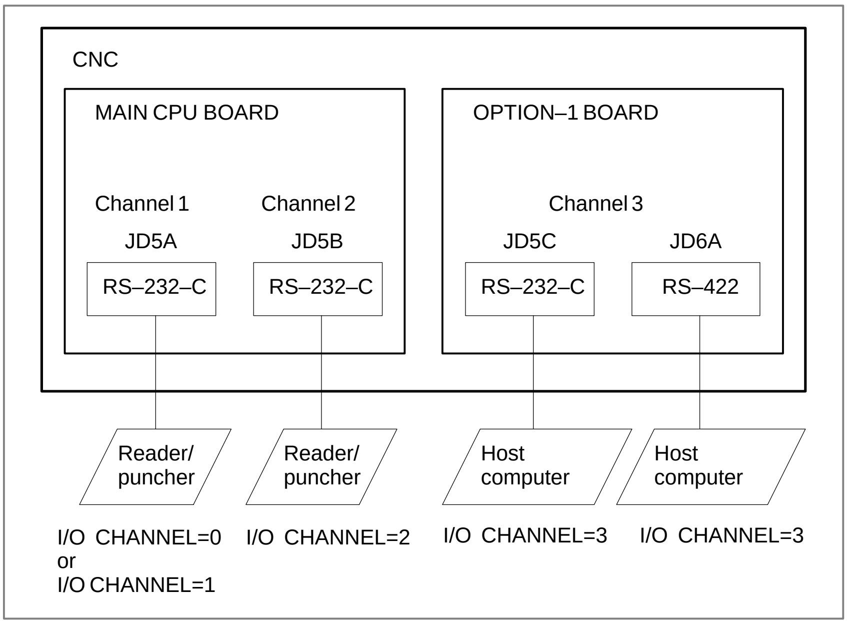
The following data can be input/output to or from external input/output devices:

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

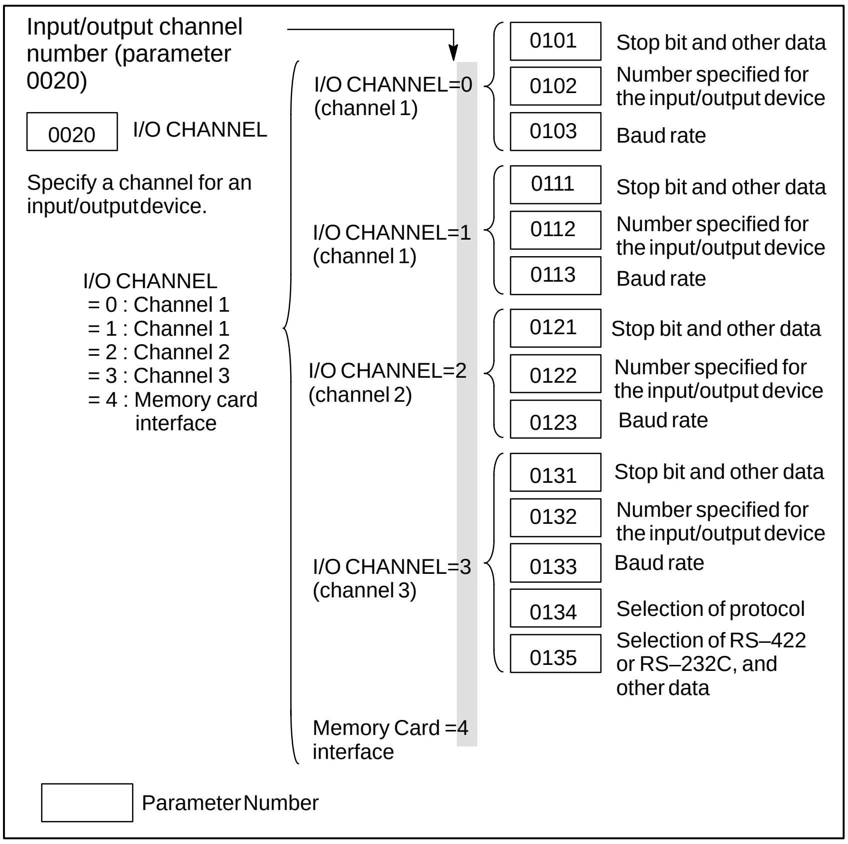
For how data is input and output, see III–8.

Parameter

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

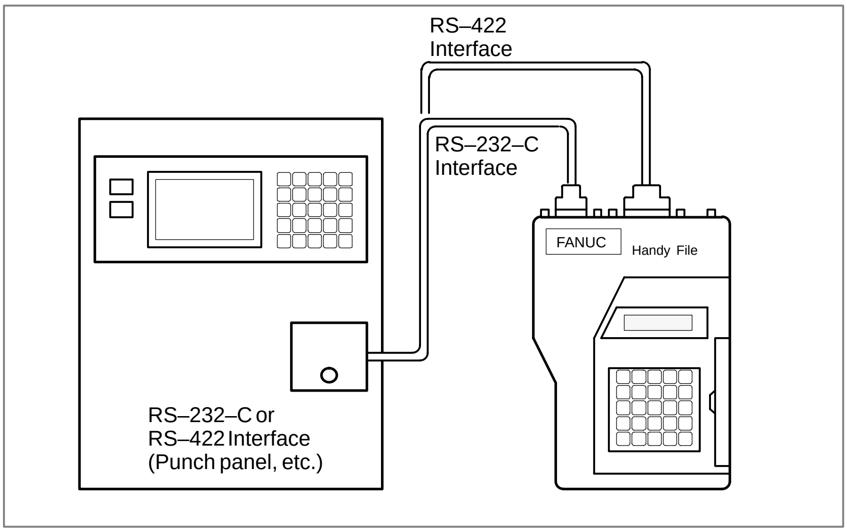


CNC has three channels of reader/punch interfaces. The input/output device to be used is specified by setting the channel connected to that device in setting parameter I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel 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.



2.4.1 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.



2.5 POWER ON/OFF

2.5.1 Turning on the Power

Procedure of turning on the power

Procedure

1 Check that the appearance of the CNC machine tool is normal. (For example, check that front door and rear door are closed.)

2 Turn on the power according to the manual issued by the machine tool builder.

3 After the power is turned on, check that the position screen is displayed. An alarm screen is displayed if an alarm occurs upon power–on. If the screen shown in Section III–2.5.2 is displayed, a system failure may have occurred.

```
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  ]
```

Current position display (7 soft keys type)

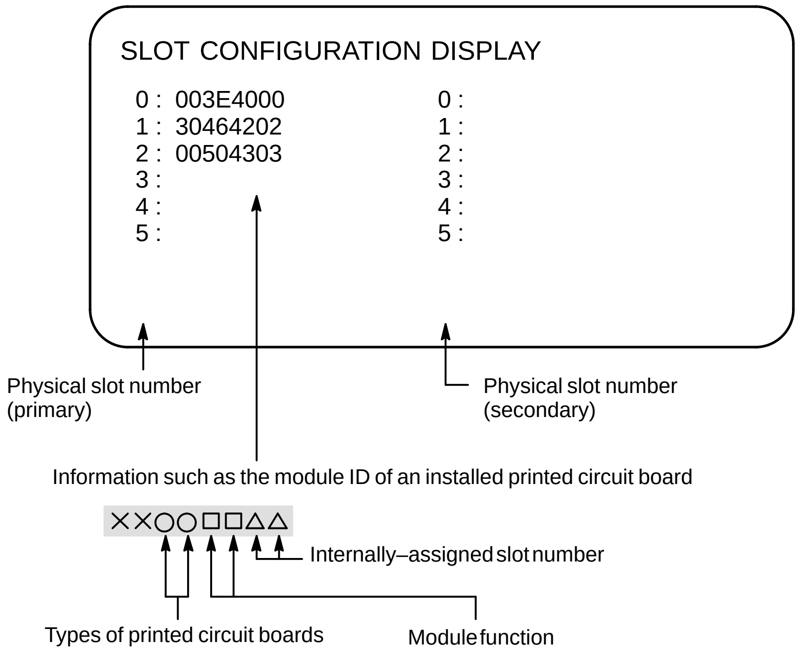
4 Check that the fan motor is rotating.

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

2.5.2 Screen Displayed at Power–on

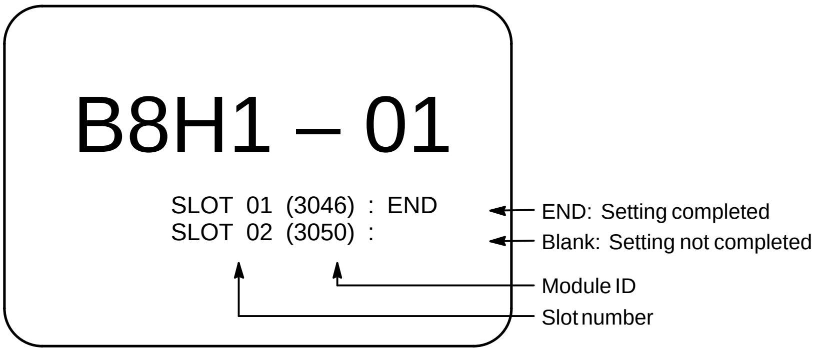
If a hardware failure or installation error occurs, the system displays one of the following three types of screens then stops.
Information such as the type of printed circuit board installed in each slot is indicated. This information and the LED states are useful for failure recovery.

Slot status display

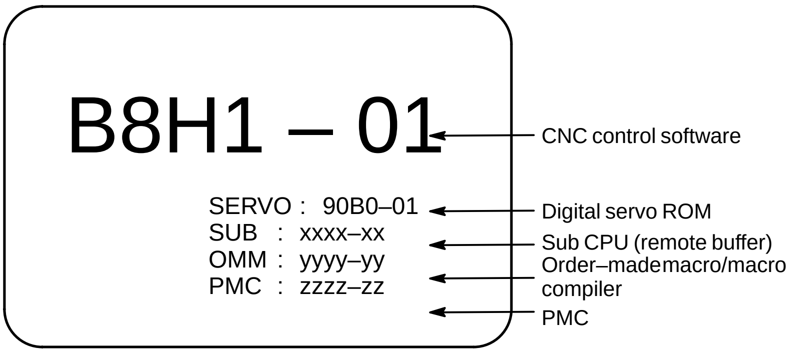


For more information about the types of printed circuit boards and module functions, refer to the Maintenance manual (B–63525EN).

Screen indicating module setting status



Display of software configuration



The software configuration can be displayed on the system configuration screen also.
Refer to the Maintenance manual (B–63665EN) for the system configuration screen.

2.5.3 Power Disconnection

Power Disconnection

Procedure

1 Check that the LED indicating the cycle start is off on the operator's panel.

2 Check that all movable parts of the CNC machine beam 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 beam builder's manual for turning off the power to the machine.

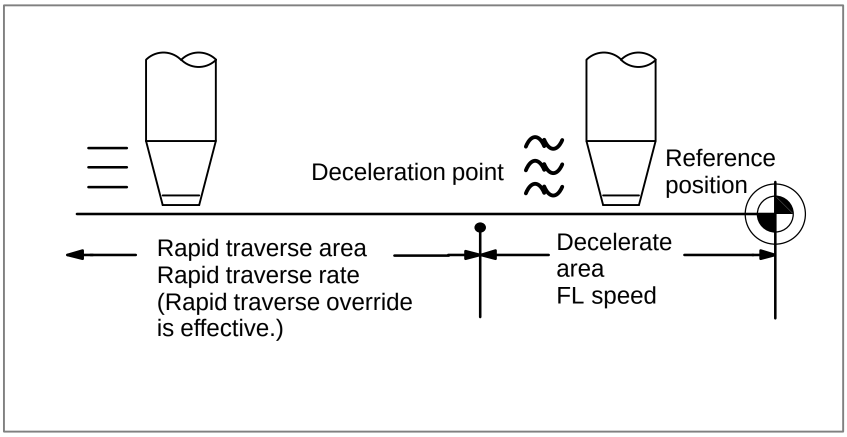
3 MANUAL OPERATION

MANUAL OPERATION are six kinds as follows :

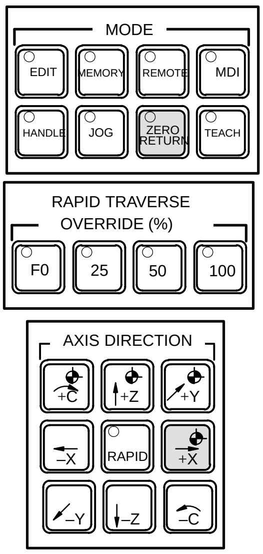
3.1 Manual reference position return
3.2 Jog feed
3.3 Incremental feed
3.4 Manual handle feed
3.5 Manual absolute on and off

3.1 MANUAL REFERENCE POSITION RETURN

The nozzle is returned to the reference position as follows :
The nozzle 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 nozzle moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 1420,1421, and 1425).
Fourstep rapid traverse override is effective during rapid traverse.
When the nozzle has returned to the reference position, the reference position return completion LED goes on. The nozzle 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



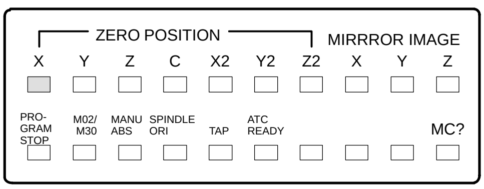
Procedure

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 nozzle 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 nozzle returns to the reference position. The nozzle can be moved along three axes simultaneously when specified so in an appropriate parameter setting. The nozzle 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 nozzle builder for the actual operations.



Explanations

- **Automatically setting the coordinate system**

Bit 0 (ZPR) of parameter No. 1201 is used for automatically setting the coordinate system. When ZPR is set, the coordinate system is automatically determined when manual reference position return is performed.
When α, β and γ are set in parameter 1250, the workpiece coordinate system is determined so that reference point the tip of the reference nozzle is X= α, Y = β, Z = γ when reference position return is performed. This has the same effect as specifying the following command for reference position return:
G92X<u>α</u>Y<u>β</u>Z<u>γ</u>;
However, when options of the workpiece coordinate system is selected, it is not able to use.

Restrictions

- **Moving the tool again**

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

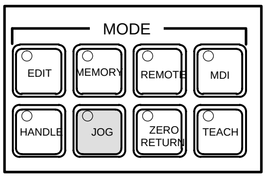
- **Reference position return completion LED**

The REFERENCE POSITION RETURN COMPLETION LED is extinguished by either of the following operations:
– Moving from the reference position.
– Entering an emergency stop state.

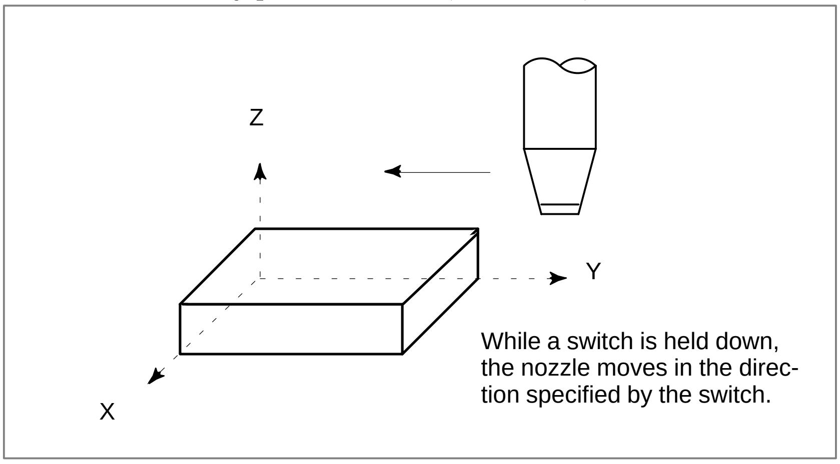
- **The distance to return to reference position**

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

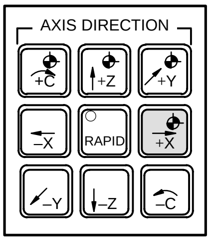
3.2 JOG FEED

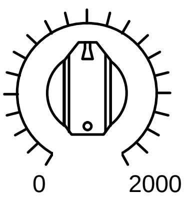


In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the nozzle along the selected axis in the selected direction.
The jog feedrate is specified in a parameter (No.1423)
The jog feedrate can be adjusted with the jog feedrate override dial.
Pressing the rapid traverse switch moves the nozzle at the rapid traverse feedrate (No. 1424) regardless of the postiotion of the jog feedrate override dial. This function is called the manual rapid traverse.
Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter JAX (No.1002#0).

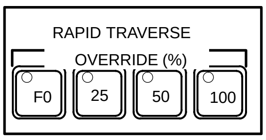


Procedure for JOG feed





JOG FEED RATE OVERRIDE



Procedure

1 Press the jog switch, one of the mode selection switches.

2 Press the feed axis and direction selection switch corresponding to the axis and direction the nozzle is to be moved. While the switch is pressed, the nozzle moves at the feedrate specified in a parameter (No. 1423). The nozzle 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 nozzle 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/decelera– tion for rapid traverse**

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

- **Change of modes**

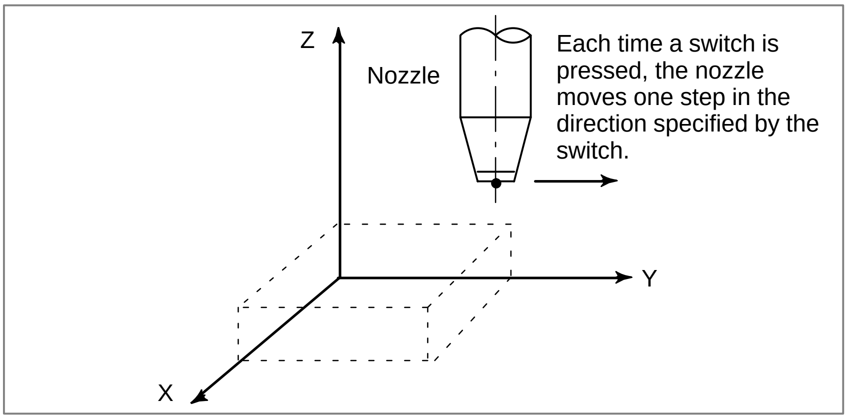
Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

- **Rapid traverse prior to reference position return**

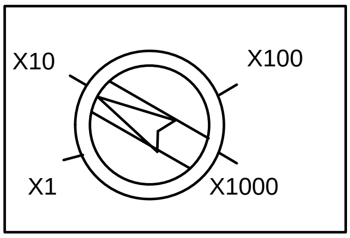
If reference position return is not performed after power–on, pushing RAPID TRAVERSE button does not actuate the rapid traverse but the remains at the JOG feedrate. This function can be disabled by setting parameter RPD (No.1401#01).

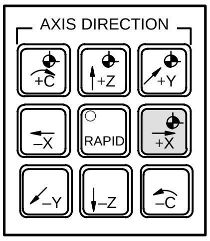
3.3 INCREMENTAL FEED

In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the nozzle one step along the selected axis in the selected direction. The minimum distance the nozzle is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.
This mode is effective when a manual pulse generator is not connected.



Procedure for Incremental Feed





1 Press the INC switch, one of the mode selection switches.

2 Select the distance to be moved for each step with the magnification dial.

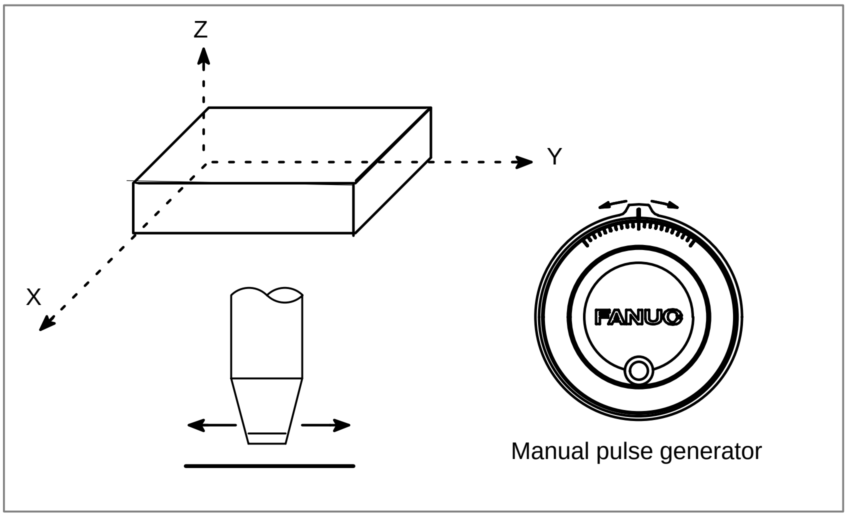
3 Press the feed axis and direction selection switch corresponding to the axis and direction the nozzle is to be moved. Each time a switch is pressed, the nozzle moves one step. The feedrate is the same as the jog feedrate.

4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the nozzle at the rapid traverse rate. Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

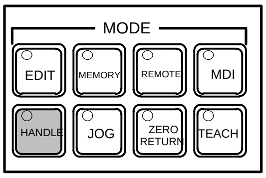
The above is an example. Refer to the appropriate manual provided by the machine nozzle builder for the actual operations.

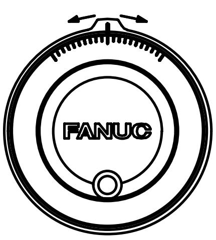
3.4 MANUAL HANDLE FEED

In the handle mode, the nozzle can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the nozzle is to be moved with the handle feed axis selection switches.
The minimum distance the nozzle is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. Or the distance the nozzle 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).



Procedure for Manual Handle Feed





Manual pulse generator

1 Press the HANDLE switch, one of the mode selection switches.

2 Select the axis along which the nozzle is to be moved by pressing a handle feed axis selection switch.

3 Select the magnification for the distance the nozzle is to be moved by pressing a handle feed magnification switch. The minimum distance the nozzle is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.

4 Move the nozzle along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the nozzle 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.

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 (bit 4 of No. 7100) or (No. 7117) specifies as follows:

- Parameter HPF (bit 4 of No. 7100)

Set value 0 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored.(The distance the nozzle 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.
(No longer rotating the handle does not immediately stop the nozzle. The nozzle is moved by the pulses accumulated in the CNC before it stops.)

- Parameter HPF (No. 7117) (It is available when parameter HPF is 0.)

Set value 0 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored.(The distance the nozzle is moved may not match the graduations on the manual pulse generator.)

Other than 0 : 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 nozzle. The nozzle is moved by the pulses accumulated in the CNC before it stops.)

- **Movement direction of an axis to the rotation of MPG (HNG_X)**

Parameter HNGx (No. 7102 #0) switches the direction of MPG in which the nozzle 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 three manual pulse generators can be connected, one for each axis. The three manual pulse generators can be simultaneously operated.

> ⚠ **WARNING**
> Rotating the handle quickly with a large magnification such as x100 moves the nozzle too fast. The feedrate is clamped at the rapid traverse feedrate.

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

3.5 MANUAL ABSOLUTE ON AND OFF

Whether the distance the nozzle 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 nozzle is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the nozzle is moved by manual operation is not added to the coordinates.

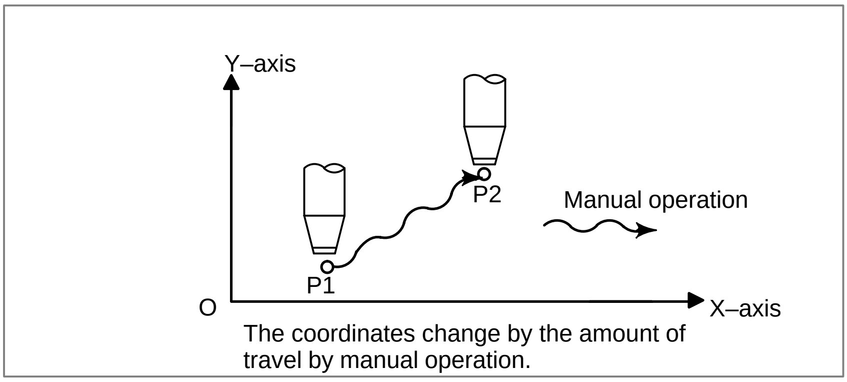


Fig.3.5 (a) Coordinates with the switch ON

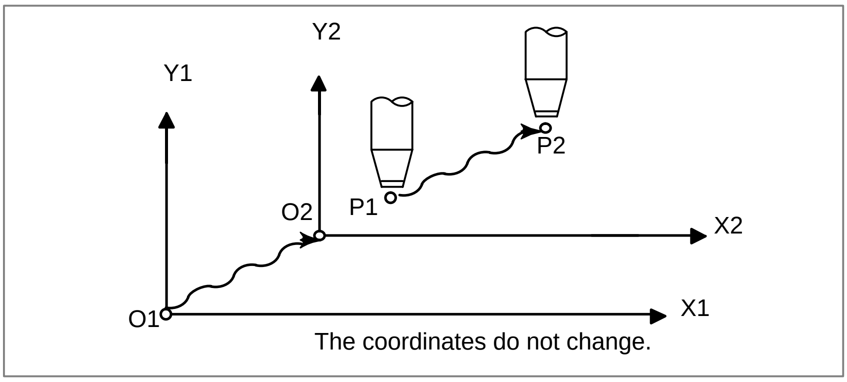


Fig.3.5 (b) Coordinates with the switch OFF

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   ; [1]
         X200.0Y150.0       ; [2]
         X300.0Y200.0       ; [3]
```

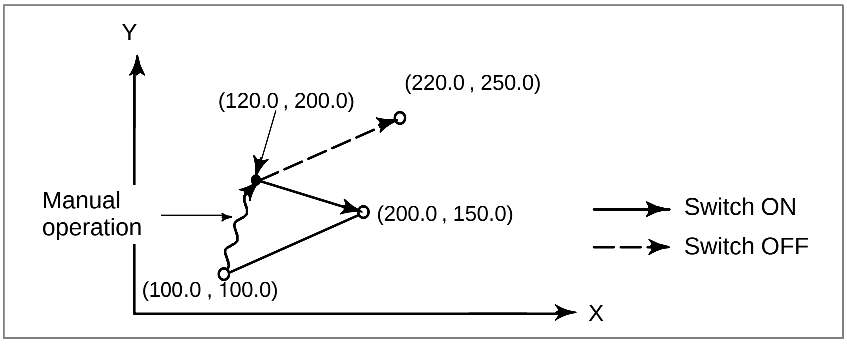
The subsequent figures use the following notation:

⟶ Movement of the nozzle when the switch is on
– – ⟶ Movement of the nozzle when the switch is off

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

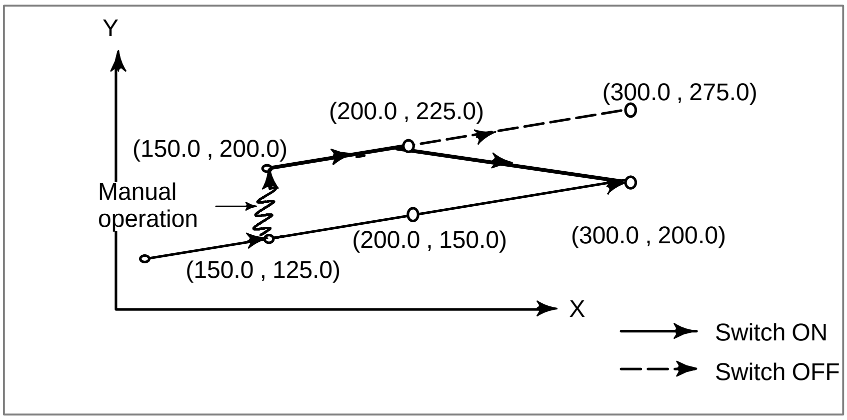
• Manual operation after the end of block

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.



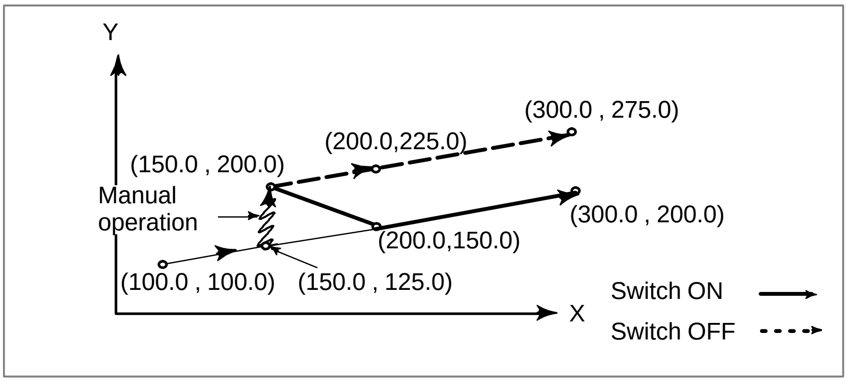
• Manual operation after a feed hold

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.



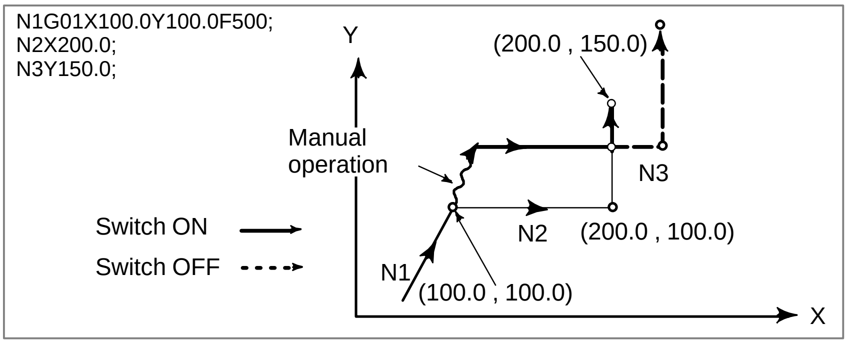
- **When reset after a manual operation following a feed hold**

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.

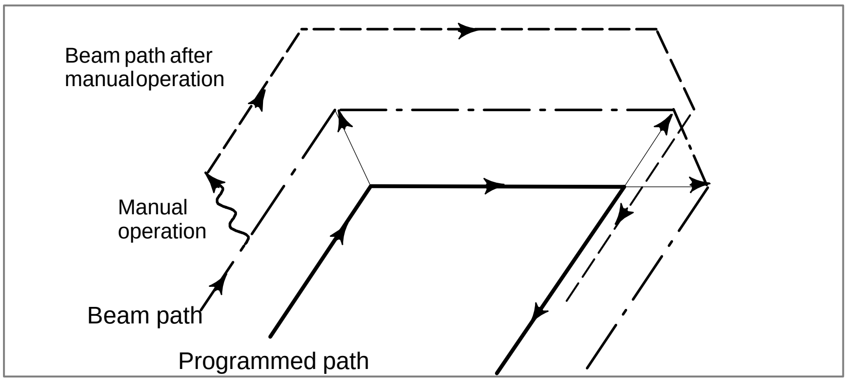


- **When the next move block is an incremental**

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 nozzle 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.

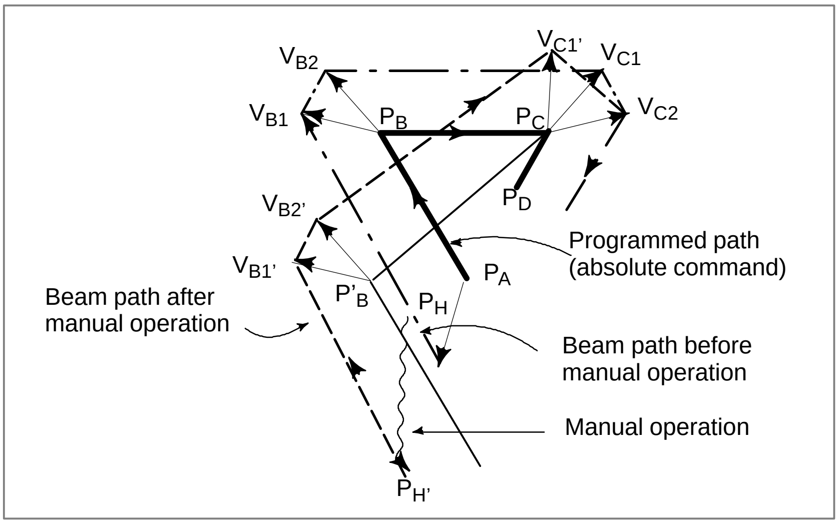


When the switch is ON during cutter compensation

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.

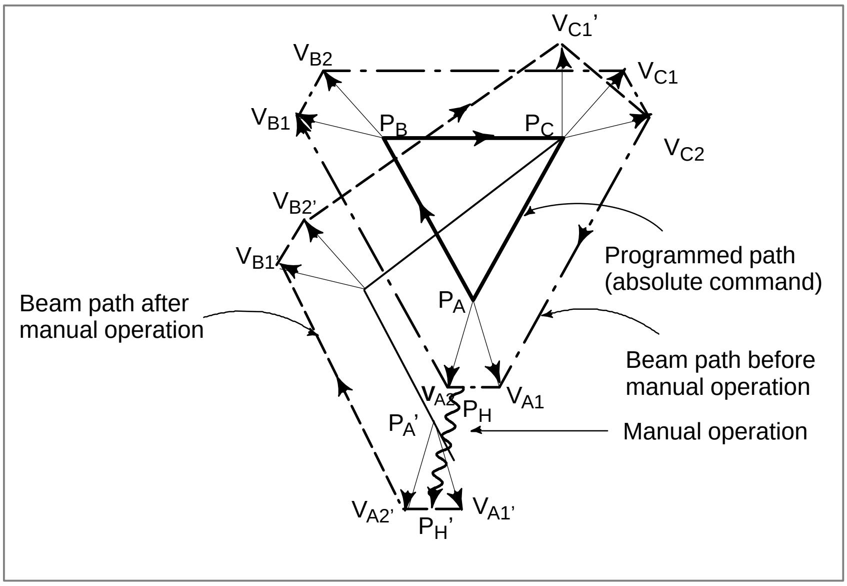
Manual operation performed in other than cornering

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 nozzle 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

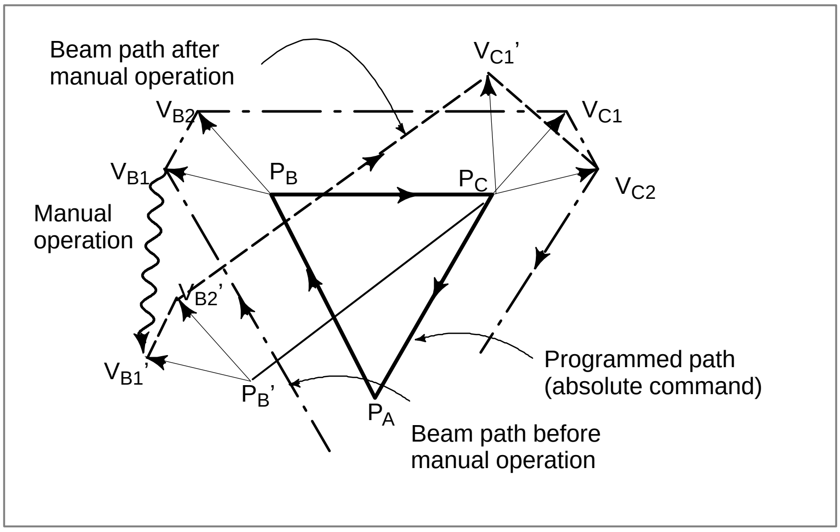
This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$, and $V_{B2'}$ are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct cutter compensation is performed 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**
  Operation by executing a program entered from the MDI panel
- **DNC operation**
  Operation while reading a program from an input/output device
- **SIMULTANEOUS INPUT/OUTPUT**
  Program execution and memory registration can be performed simultaneously.
- **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
- **RETRACE FUNCTION**
  Function for moving the nozzle in the reverse direction to retrace the path followed, and for moving the nozzle in the forward direction again along the retraced path.
- **MANUAL INTERVENTION AND RETURN**
  Function restarting automatic operation by returning the nozzle to the position where manual intervention was started during automatic operation
- **Retreat and retry functions**
  This function enables machining to be restarted from the start block.

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 key on the 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 Press the **MEMORY** mode selection switch.

2 Select a program from the registered programs. To do this, follow the steps below.

2–1 Press PROG to display the program screen.

2–2 Press address O .

2–3 Enter a program number using the numeric keys.

2–4 Press the **[O SRH]** soft key.

3 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.

4 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, T was being executed, the operation is stopped after M, 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 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 nozzle 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 MDI panel or external reset signal.

When reset operation is applied to the system during a nozzle 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 4.7.

4.2 MDI OPERATION

In the **MDI** mode, a program consisting of up to 10 lines can be created in the same format as normal programs and executed from the MDI panel. MDI operation is used for simple test operations.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Procedure for MDI Operation

Procedure

1 Press the **MDI** mode selection switch.

2 Press the PROG function key on the MDI panel to select the program screen. The following screen appears:

```
PROGRAM ( MDI )                          O0010  N00002

 O0000;



 G00  G90  G94  G40  G80  G50  G54  G69
 G17  G22  G21  G49  G98  G67  G64  G15
                B              H    M
      T                        D
      F                   S

 >_
  MDI   ****   ***  ***          20 : 40 : 05
[ PRGRM ][  MDI  ][ CURRNT ][  NEXT  ][ (OPRT) ]
```

Program number O0000 is entered automatically.

3 Prepare a program to be executed by an operation similar to normal program editing. M99 specified in the last block can return control to the beginning of the program after operation ends. Word insertion, modification, deletion, word search, address search, and program search are available for programs created in the MDI mode. For program editing, see III–9.

4 To entirely erase a program created in MDI mode,use one of the following methods:

a. Enter address O, then press the DELETE key on the MDI panel.

b. Alternatively, press the RESET key. In this case, set bit 7 of parameter MCL No. 3203 to 1 in advance.

5 To execute a program, set the cursor on the head of the program. (Start from an intermediate point is possible.) Push Cycle Start button on the operator's panel. By this action, the prepared program will start. When the program end (M02, M30) or ER(%) is executed, the prepared program will be automatically erased and the operation will end.
By command of M99, control returns to the head of the prepared program.

```
PROGRAM ( MDI )                                   O0001  N00003
O0000  G00  X100.0  Y200. ;
 M03  ;
 G01  Z120.0  F500  ;
 M93  P9010  ;
 G00  Z0.0  ;
 %

 G00   G90   G94   G40   G80   G50   G54   G69
 G17   G22   G21   G49   G98   G67   G64   G15
                   B           H     M
       T                       D
       F                 S
 >_
 MDI    ****   ***  ***              12 : 42 : 39
( PRGRM )(   MDI   )( CURRNT )( NEXT  )( (OPRT) )
```

6 To stop or terminate MDI operation in midway through, follow the steps below.

a. Stopping MDI 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, T was being executed, the operation is stopped after M, T is finished.

When the cycle start switch on the machine operator's panel is pressed, machine operation restarts.

b. Terminating MDI operation

Press the RESET key on the MDI panel.

Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Explanation

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**

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

- **Editing a program during MDI operation**

A program can be edited during MDI operation. The editing of a program, however, is disabled until the CNC is reset, when bit 5 (MIE) of parameter No. 3203 is set accordingly.

Limitations

- **Program registration**

Programs created in MDI mode cannot be registered.

- **Number of lines in a program**

A program can have as many lines as can fit on one page of the 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).

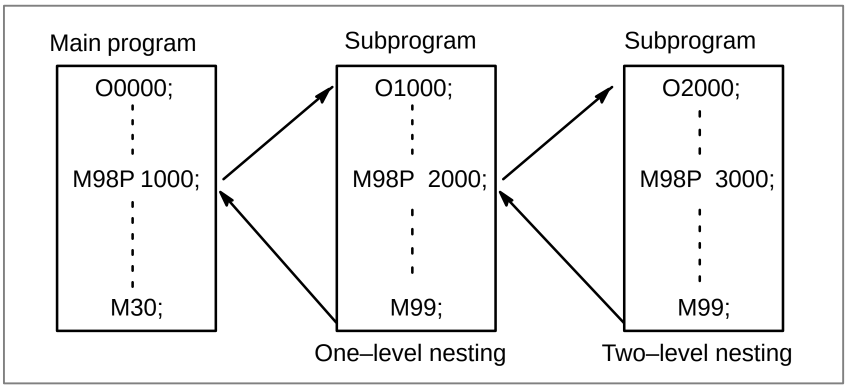


Fig.4.2 Nesting Level of Subprograms Called from the MDI Program

- **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.

4.3 DNC OPERATION

By activating automatic operation during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read in via reader/puncher interface, or remote buffer.

If the floppy cassette directory display option is available, it is possible to select files (programs) saved in an external input/output unit of a floppy format (Handy File, FLOPPY CASSETTE, or FA card) and specify (schedule) the sequence and frequency of execution for automatic operation.

(see III–4.4)

To use the DNC operation function, it is necessary to set the parameters related to the reader/punch interface, and remote buffer in advance.

The following procedure is an example. For actual operation, see the manual provided by the machine tool builder.

DNC OPERATION

Procedure

1 Search for the program (file) to be executed.

2 Press the REMOTE switch on the machine operator's panel to set RMT mode, then press the cycle start switch. The selected file is executed. For details of the use of the REMOTE switch, refer to the relevant manual supplied by the machine tool builder.

- **Program check screen (Seven soft keys type)**

```
PROGRAM CHECK                                O0001 N00020

N020 X100.0 Z100.0 (DNC–PROG) ;
N030 X200.0 Z200.0 ;
N050 X400.0 Z400.0 ;

 (RELATIVE)   (DIST TO GO)   G00  G17  G90
 X  100.000   X      0.000   G22  G94  G21
 Y  100.000   Y      0.000   G41  G49  G80
 Z    0.000   Z      0.000   G98  G50  G67
 A    0.000   A      0.000             B
 C    0.000   C      0.000   H         M
HD.T        NX.T             D         M
   F                    S              M
 ACT.F            SACT            REPEAT
 RMT  STRT  MTN ***          21:20:05
[  ABS  ]   [  REL  ]   [       ]  [       ]  [ (OPRT) ]
```

- **Program screen (Seven soft keys type)**

```
PROGRAM                                            O0001 N00020

N020 X100.0 Z100.0 (DNC–PROG) ;
N030    X200.0    Z200.0 ;
N040    X300.0    Z300.0 ;
N050    X400.0    Z400.0 ;
N060    X500.0    Z500.0 ;
N070    X600.0    Z600.0 ;
N080    X700.0    Z400.0 ;
N090    X800.0    Z400.0 ;
N100    x900.0    z400.0 ;
N110    x1000.0   z1000.0 ;
N120    x800.0    z800.0 ;

 RMT  STRT  MTN  ***            21:20:05
[ PRGRM ]  [ CHECK ]  [        ]  [        ]  [ (OPRT) ]
```

- **Program screen (Twelve soft keys type)**

```
PROGRAM                                F0001 N00020

N020 X100.0 (DNC–PROG) ;               N180 Z50.0 ;
N030 X90.0 ;                           N190 Z40.0 ;
N040 X80.0 ;                           N200 Z30.0 ;
N050 X70.0 ;                           N210 Z20.0 ;
N060 X60.0 ;                           N220 Z10.0 ;
N070 X50.0 ;                           N230 Z0.0 ;
N080 X40.0 ;                           N240 M02 ;
N090 X30.0 ;                           %
N100 X20.0 ;
N110 X10.0 ;
N120 X0.0 ;
N130 Z100.0 ;
N140 Z90.0 ;
N150 Z80.0 ;
N160 Z70.0 ;
N170 Z60.0 ;

                                       RMT STRT MTN ***    22:23:24

                                       PRGRM CHECK    (OPRT) +
```

During DNC operation, the program currently being executed is displayed on the program check screen and program screen.
The number of displayed program blocks depends on the program being executed.
Any comment enclosed between a control–out mark (() and control–in mark ()) within a block is also displayed.

Explanations

- During DNC operation, sub programs stored in memory can be called.
- During DNC operation, macro programs stored in memory can be called.

Limitations

- **Limit on number of characters**

In program display, no more than 256 characters can be displayed. Accordingly, character display may be truncated in the middle of a block.

- **M198 (command for calling a program from within an external input/output unit)**

In DNC operation, M198 cannot be executed. If M198 is executed, P/S alarm No. 210 is issued.

- **Custom macro**

In DNC operation, custom macros can be specified, but no repeat instruction and branch instruction can be programmed. If a repeat instruction or branch instruction is executed, P/S alarm No. 123 is issued. When reserved words (such as IF, WHILE, COS, and NE) used with custom macros in DNC operation are displayed during program display, a blank is inserted between adjacent characters.

Example

		[During DNC operation]
#102=SIN[#100] ;	→	#102 = S I N[#100] ;
IF[#100NE0]GOTO5 ;	→	I F[#100NE0] G O T O 5 ;

- **M99**

When control is returned from a subprogram or macro program to the calling program during DNC operation, it becomes impossible to use a return command (M99P****) for which a sequence number is specified.

Alarm

Number	Message	Contents
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
210	CAN NOT COMAND M198/M199	Or M198 is executed in the DNC operation. Modify the program.

4.4 SIMULTANEOUS INPUT/OUTPUT

While an automation operation is being performed, a program input from an I/O device connected to the reader/punch interface can be executed and the program can be registered in memory at the same time.
In addition, a program can be output through the reader/punch interface while the program registered in memory being executed.

Simultaneous Input/Output

Procedure

- **Basic simultaneous input procedure**

1. Search for the program (file) to be output and executed.
2. Press the REMOTE switch on the machine operator’s panel to set RMT mode. For details of the use of the REMOTE switch, refer to the relevant manual supplied by the machine tool builder.
3. Set the simultaneous output operation mode select signal to 1.
4. Press the cycle start switch.
5. Program output and execution is performed on a block–by–block basis.

Program check screen

```
PROGRAM CHECK                                        F0001 N00100

N100 X100. Y100. ;
N200 G01   X50.  Y50. ;
N300 G00   X0  Y0 ;

 (RELATIVE)    (DIST TO GO)    G00   G17   G90
 X   100.000   X      0.000    G22   G94   G21
 Y   100.000   Y      0.000    G40   G49   G80
 Z     0.000   Z      0.000    G98   G50   G67
 A     0.000   A      0.000          B
 C     0.000   C      0.000    H     M
HD.T          NX.T             D     M
    F                 S              M
 ACT.F           SACT             REPEAT
 RMT STRT      MTN  ***        21:20:05
[  ABS  ]  [  REL  ]  [        ]  [        ]  [ (OPRT) ]
```

When a program is displayed, three blocks are displayed: the block currently being executed and the next two to be executed. When the single block function is selected, only the block currently being executed is displayed.
Any comment enclosed between a control–out mark (() and control–in mark ()) within a block is not displayed.

- **Basic simultaneous output procedure**

1. Search for the program (file) to be output and executed.
2. Press the REMOTE switch on the machine operator’s panel to set RMT mode. For details of the use of the REMOTE switch, refer to the relevant manual supplied by the machine tool builder.
3. Set the simultaneous output operation mode select signal to 1.
4. Press the cycle start switch.
5. Program output and execution is performed on a block–by–block basis.

Limitations

- **M198 (command for calling a program from within an external input/output unit)**

M198 cannot be executed in the input, output and run simultaneous mode. An attempt to do so results in alarm No. 210.

- **Macro control command**

A macro control command cannot be executed in the input, output and run simultaneous mode. An attempt to do so results in P/S alarm No. 123.

- **Alarm**

If an alarm condition occurs during the input, output and run simultaneous mode, a block being processed when the alarm condition occurs and all blocks before that are input or output.

- **File name**

In the output and run simultaneous mode, if a device used is a floppy disk drive or FA card, the file name is the execution program number.

- **Sub program call**

When a program is being executed in the output and run simultaneous mode, if a subprogram is called, only the main program is output.

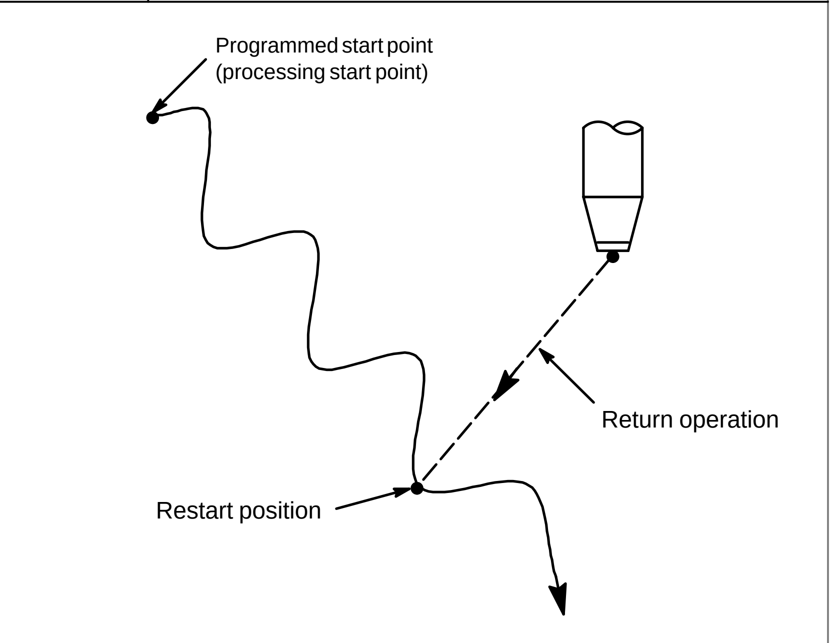
Alarm

Number	Message	Contents
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
210	CAN NOT COMMAND M198/M199	M198 or M199 is executed in the DNC operation. M198 is executed in the DNC operation. Modify the program.
222	DNC OP. NOT AL-LOWED IN BG.–EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.

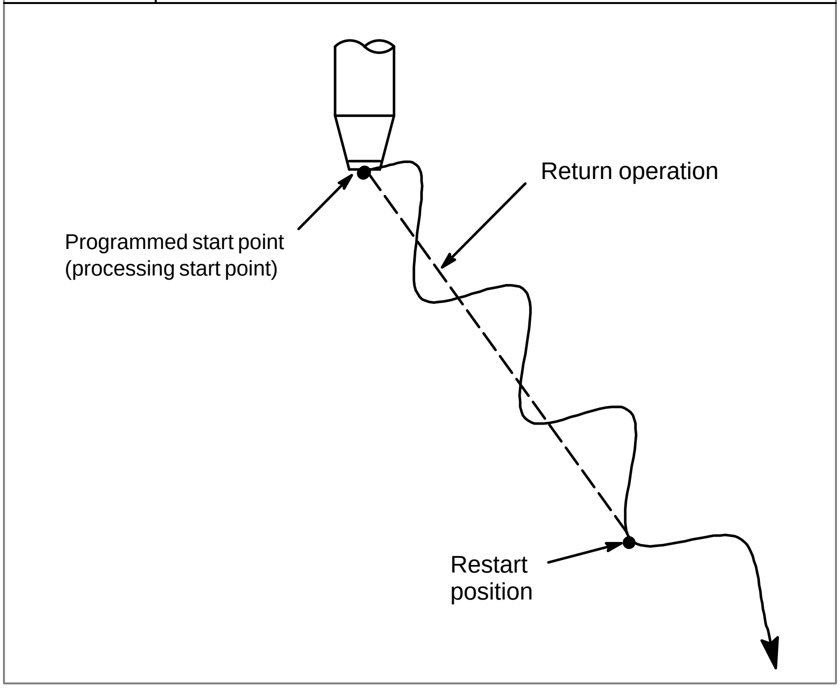
4.5 PROGRAM RESTART

When you want to restart the machining operation after a day off, you can use this function. Machining can be restarted from the target block by specifying the sequence or block number of that block.
This function can also be used as a high–speed program check function.
There are three restart methods: the P–type, Q–type, and laser–specification methods.

P type FS0–L type	Machining can be restarted from any point.



Q type	Before machining is restarted, the machine must be moved to the programmed start point (processing start point).



Procedure for program restart by specifying a sequence number (P and Q types)

Procedure 1

[P TYPE]

1 Retract the nozzle and perform the required operations (such as replacement of the nozzle). Change the offset value if required. (Go to Procedure 2.)[Q type]

[Q TYPE]

1 When the power is turned on or an emergency stop is released, perform the required operations at that time, including reference position return.

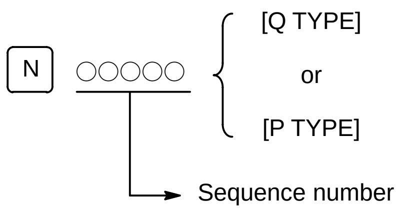
2 Move the machine manually to the programmed start point (processing start point) and set the modal data and coordinate system in the same conditions as those at the processing start.

3 If required, change the offset value. (Go to Procedure 2.)

Procedure 2

[Common to P and Q types]

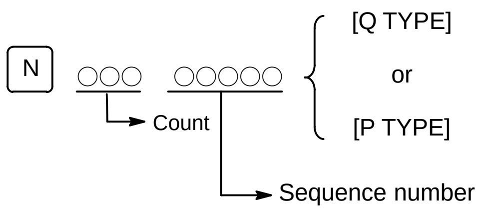
1 Set the program restart switch on the machine operator's panel to the ON position.



2 Press the PROG function key to display the program to be restarted.

3 Find the start of the program.

4 Enter the sequence number of the block from which the program is to be restarted, then press the **[P TYPE]** or **[Q TYPE]** soft key.



When the same sequence number appears more than once, the location of the target sequence number must be specified. Specify the count before the sequence number.

5 A search is made for the sequence number, and the program restart screen appears on the screen.

```
PROGRAM RESTART                              O0002  0N0100

  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           ****************
  1 X 1. 459
  2 Y 10. 309            T ****************
  3 Z 7. 320             S *****

                                         S     0   T0000
 MEM  ****   ***  ***            10 : 10 : 40
( RSTR )(        )( FL.SDL )(        )( (OPRT) )
```

DESTINATION indicates the position at which processing is to restart. DISTANCE TO GO indicates the distance from the current nozzle position to the position where processing is to restart. The number to the left of each axis name indicates the order of the axis (determined by parameter setting) along which the nozzle moves to the restart position. The coordinates and amount of travel for restarting the program can be displayed for up to five axes. If your system supports six or more axes, pressing the **[RSTR]** soft key again displays the data for the sixth and subsequent axes. (The program restart screen displays only the data for CNC–controlled axes.)
M: Fourteen most recently specified M codes
T: Two most recently specified T codes
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 Set the program restart switch to the OFF position. At this time, the number to the left of each axis name under DISTANCE TO GO blinks.

7 Check the screen for the M, T, and B codes to be executed. When M, T, and B codes are to be executed, place the CNC in **MDI** mode, then execute the M, T, and B functions in MDI mode. After execution, restore the CNC to 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 the nozzle will hit the workpiece or other object when it moves to the processing restart position. If the nozzle will hit an object, move the nozzle manually to a position from which the nozzle can move to the processing restart position without encountering any obstacles.

9 Press the cycle start button. The nozzle moves to the processing restart position at the dry run feedrate sequentially along axes in the order specified by parameter settings (No. 7310). Processing is then restarted.

Procedure for program restart by specifying a block number (P and Q types)

Procedure 1

[P TYPE]

1 Retract the nozzle and perform the required operations (such as replacement of the nozzle). Change the offset value if required. (Go to Procedure 2.)

[Q TYPE]

1 When the power is turned on or an emergency stop is released, perform the required operations at that time, including reference position return.

2 Move the machine manually to the programmed start point (processing start point) and set the modal data and coordinate system under the same conditions as those at processing start.

3 If required, change the offset value. (Go to Procedure 2.)

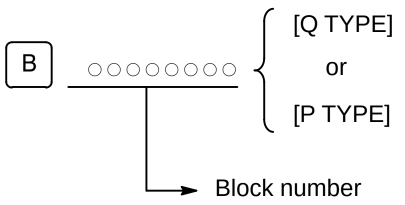
Procedure 2

[Common to P and Q types]

1 Set the program restart switch on the machine operator's panel to the ON position.

2 Press the PROG function key to display the program to be restarted.

3 Find the start of the program. Press the RESET key.



4 Enter the number of the block from which the program is to be restarted, then press the **[P TYPE]** or **[Q TYPE]** soft key. The block number can consist of up to eight digits.

5 A search in made for the block number, and the program restart screen appears on the screen.

```
PROGRAM RESTART                              O0002  N01000
                                 BC : 00000002
   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 indicates the position at which processing is to restart.
DISTANCE TO GO indicates the distance from the current nozzle position to the position where processing is to restart. The number to the left of each axis name indicates the order of the axis (determined by parameter setting) along which the nozzle moves to the restart position.
The coordinates and amount of travel for restarting the program can be displayed for up to five axes. If your system supports six or more axes, pressing the **[RSTR]** soft key again displays the data for the sixth and subsequent axes. (The program restart screen displays only the data for CNC–controlled axes.)
M: Fourteen most recently specified M codes
T: Two most recently specified T codes
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 Set the program restart switch to the OFF position. At this time, the number to the left of each axis name under DISTANCE TO GO blinks.

7 Check the screen for the M and T codes to be executed. When M and T codes are to be executed, place the CNC in **MDI** mode, then execute the M and T functions in MDI mode. After execution, restore the CNC to 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 the nozzle will hit the workpiece or other object when it moves to the processing restart position. If the nozzle will hit an object, move the nozzle manually to a position from which the nozzle can move to the processing restart position without encountering any obstacles.

9 Press the cycle start button. The nozzle moves to the processing restart position at the dry run feedrate sequentially along the axes in the order specified by parameter settings (No. 7310). Processing is then restarted.

Explanations

• Block number

When the NC 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 noting the number displayed on the screen. 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 value of the displayed number + 1.

The number of blocks is counted from the start of processing, assuming one NC statement 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 and clearing the number of blocks

The number of blocks is also stored after the power is turned off. The stored number of blocks is cleared by cycle start in reset state.

• Block number displayed when a program is halted or stopped

The program screen usually displays the number of the block being executed. Once the execution of a block is complete, the CNC is reset, or the program is stopped by single–block stop, the program screen displays the number of the block 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 blocks.

• Block number consisting of more than eight digits

When the length of the block number displayed on the program screen exceeds eight digits, the block number is reset to 0 and counting continues.

Procedure for program restart by specifying a block (laser specification)

Procedure 1

1 Retract the nozzle and perform required operations (such as replacement of the nozzle).

Procedure 2

1 Select EDIT mode, then move the cursor to the block from which the program is to be restarted.

2 Select MEM mode, then press the laser specification program restart switch (for the actual switch name, refer to the manual provided by the machine tool builder).

3 Press the cycle start button. A search is made for the block from the top of the program.
When the block before the block from which the program is to be restarted is read, the nozzle is positioned to that point and enters the feed hold state.

4 Press the laser specification program restart switch again, then press the cycle start button again. Automatic operation starts from the target block.

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 to an external workpiece origin offset value)

• Restart block

The block from which the program is to be restarted need not be that block executed when operation was interrupted. The program can be restarted from any block. When P–type restart is performed, the restart block must use the same coordinate system as the block executed when operation was interrupted.

• Single block

When single–block mode is set during movement to the restart position, operation stops every time the nozzle completes movement along an axis. When operation is stopped in 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 for which it has not yet been performed. A return operation cannot be performed again on axes for which a return has already been completed.

• Reset

Never perform reset during the period between the start of a search for restart and the restart of processing. Otherwise, restart must be performed again from the first step.

- **Manual absolute**

Regardless of whether processing has started, manual absolute mode must be set when manual operation is performed.

- **Reference position return**

If no absolute–position detector (absolute pulse coder) is provided, before restart, always perform reference position return after turning on the power.

- **Dwell command and laser output command**

During search operation, neither a dwell command nor laser output command is executed.

- **Laser specification**

(1) The program executed just before can be restarted.

(2) The program executed before the power was turned off cannot be restarted when the power is turned on again.

(3) For programs that are to be restarted, it is impossible to change a work coordinate system.

(4) Move commands (G27, G28, G29, G30, and G53) that use the machine coordinate system are ignored during program restart.

(5) No program can be restarted from a midpoint of a spatial arc interpolation (G12) command.

(6) A program cannot be restarted from a block in a subprogram.

(7) All manual operations must be performed with “manual absolute” turned on no matter whether machining has not been started or has been finished.

(8) Neither the dwell command nor the laser output command is executed during a search.

Alarm

Alarm number	Explanation
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, an emergency stop was released, or alarm 094 to 097 was reset, P–type restart was specified.
098	After the power was turned on, a restart operation was performed without reference position return, but a G28 command was found in the program.
099	A move command was issued from the MDI panel during a restart operation.
5020	An erroneous parameter was specifed for restarting a program.

CAUTION

As a rule, the nozzle cannot be returned to the correct position in any of the following cases.
Special care must be taken in the following cases because none of them causes an alarm:

- Manual operation is performed when the manual absolute mode is off.
- Manual operation is performed in machine lock mode.
- The mirror image function is used.
- Manual intervention is performed while the nozzle is moving along an axis to return to the target position.
- Restart is specified for a block between the block for skip cutting and absolute command block which appears after that block.

4.6 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.
This function is effective, when the floppy cassette directory display option is avairable and the floppy cassette is selected as the valid I/O device.

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 soft key **[PRGRM]** 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 M099 were executed during scheduled operation, or M198 was executed during DNC operation.

4.7 SUBPROGRAM CALL FUNCTION (M198)

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:

To use this function, the Floppy Cassette directory display option must be installed.

Format

1. FS15 tape format

M198 P○○○○ L∆∆∆∆ ;

- M198: I/O devices call instruction
- P○○○○: File number for a file in the I/O device
- L∆∆∆∆: Number of repetitions

2. Other than FS15 tape format

M198 P○○○○ ∆∆∆∆ ;

- M198: I/O devices call instruction
- P○○○○: Number of repetitions
- ∆∆∆∆: File number for a file in the I/O device

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.

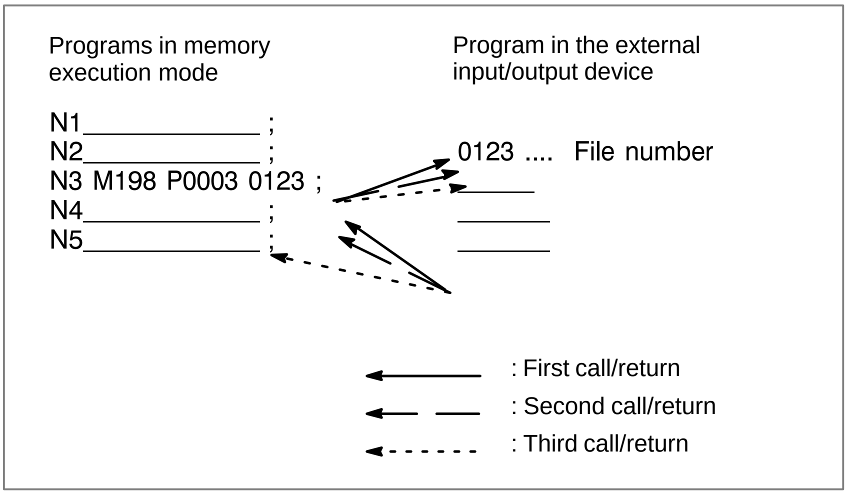


Fig.4.7 Program Flow When M198 is Specified

Restrictions

NOTE

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.

4.8 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.

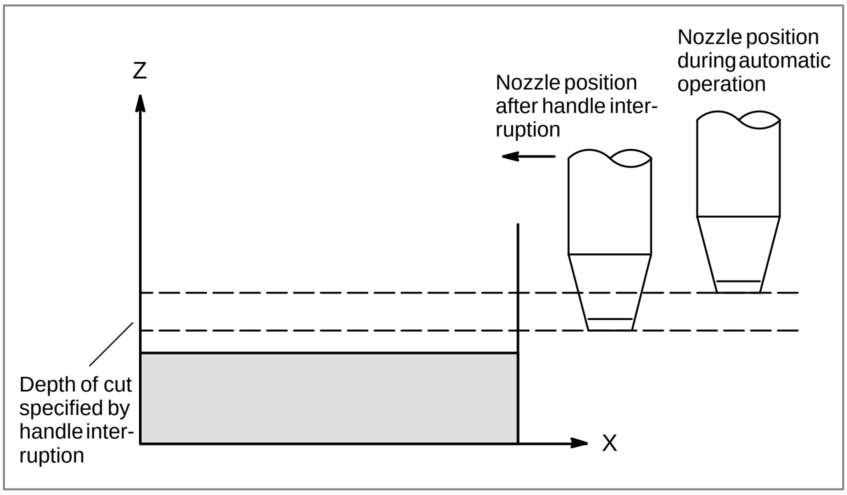


Fig.4.8 Manual Handle Interruption

- 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.

WARNING

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).

NOTE

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.

Display	Relation
Machine lock	Machine lock is effective. The nozzle does not move even when this signal turns on.
Interlock	Interlock is effective. The nozzle 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 function key POS, then press the chapter selection soft key **[HNDL]**.
The move amount by 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 UNI : 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 is cleared when the manual reference position return ends every axis.

- **Display of five axes or more**

Actual position display of five axes (or more) system is the same “ALL” position display. (See III–11.1.3)

4.9 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 MDI panel.

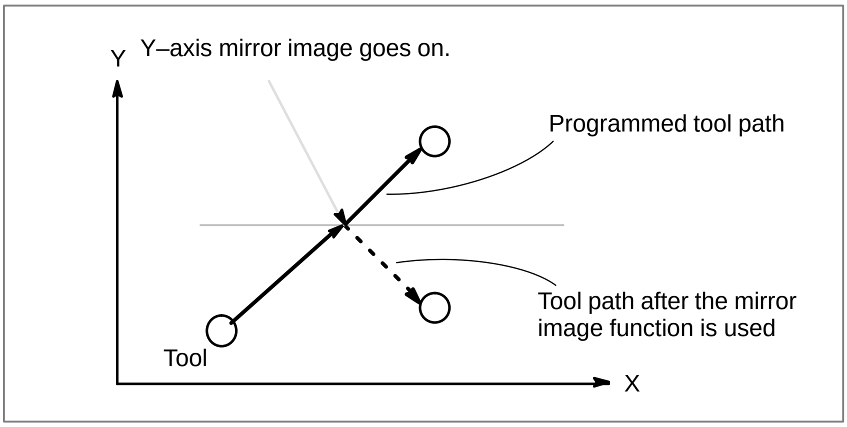


Fig.4.9 Mirror Image

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.

```
SETTING (MIRROR IMAGE)                          O0020 N00001

     MIRROR IMAGE   X = 1  (0 : OFF  1 : ON)
     MIRROR IMAGE   Y = 0  (0 : OFF  1 : ON)
     MIRROR IMAGE   Z = 0  (0 : OFF  1 : OM)






  >_
  MEM  **** *** ***                 14 : 47 : 57
[OFFSET] [SETING] [ WORK ] [      ] [(OPRT)]
```

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 bit 0 of parameter 0012 (MIRx) to 1 or 0.
- 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) cannot be reserved.

4.10 RETRACE FUNCTION

With the retrace function, the nozzle can be moved in the reverse direction (reverse movement) by using the REVERSE switch during automatic operation to trace the programmed path. The retrace function also enables the user to move the nozzle in the forward direction again (forward return movement) along the retraced path until the retrace start position is reached. When the nozzle reaches the retrace start position, the nozzle resumes movement according to the program.
This function is specific to a two–dimensional machine tool, and therefore cannot be used with a three–dimensional function.

Procedure for Retrace Operation

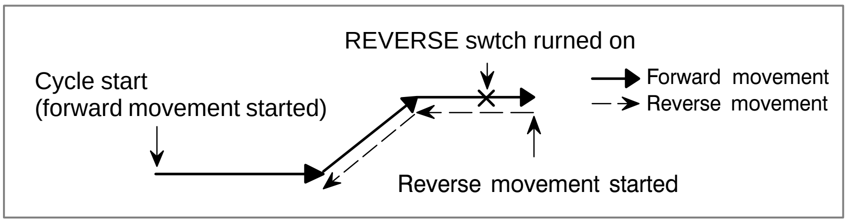
Procedure

- **Forward movement → Reverse movement**

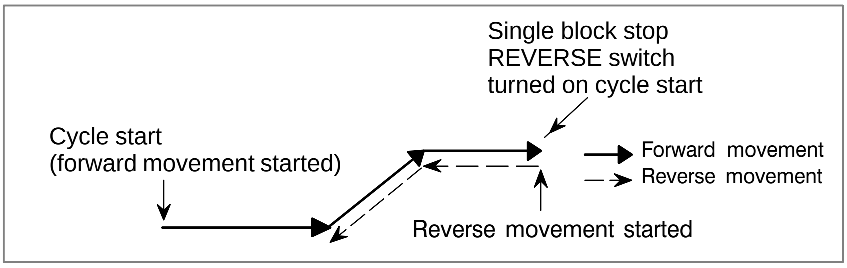
To move the nozzle in the forward direction, turn off the REVERSE switch on the operator's panel, then press the cycle start switch. If the REVERSE switch on the operator's panel is on, the nozzle moves in the reverse direction or completes reverse movement.
Three methods are available for moving the nozzle in the reverse direction along the programmed path.

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

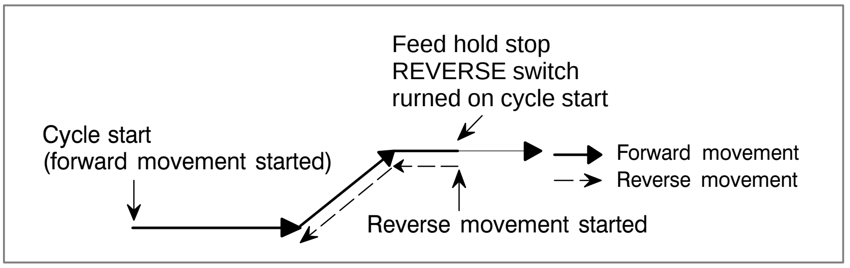
In the case of 1) above, the nozzle starts reverse movement after completion of the block currently being executed (after execution up to the position of a single block stop). Turning on the REVERSE switch on the operator's panel does not immediately start reverse movement.



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



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

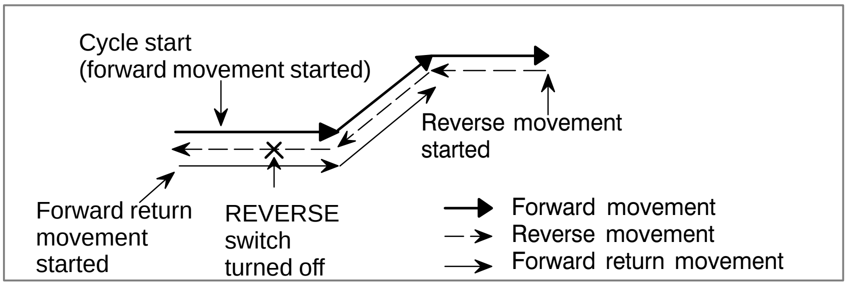


- **Reverse movement → Forward return movement**

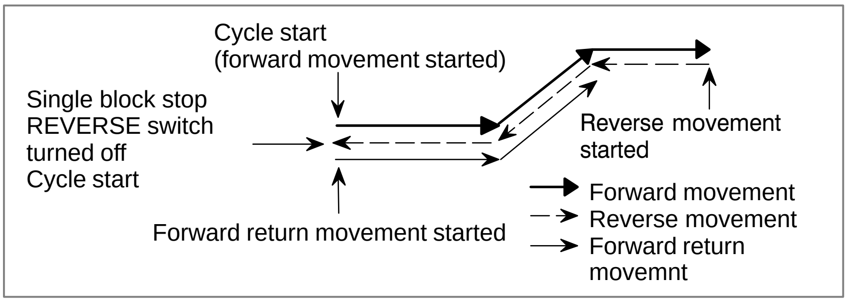
Three methods are available for moving the nozzle in the forward direction again along the retraced path.

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

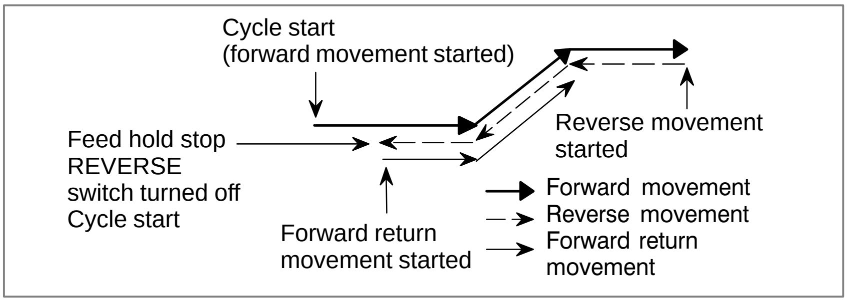
In the case of 1) above, the nozzle starts forward return movement after completion of the block currently being executed (after execution up to the position of a single block stop). Turning off the REVERSE switch on the operator's panel does not immediately start forward return movement.



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

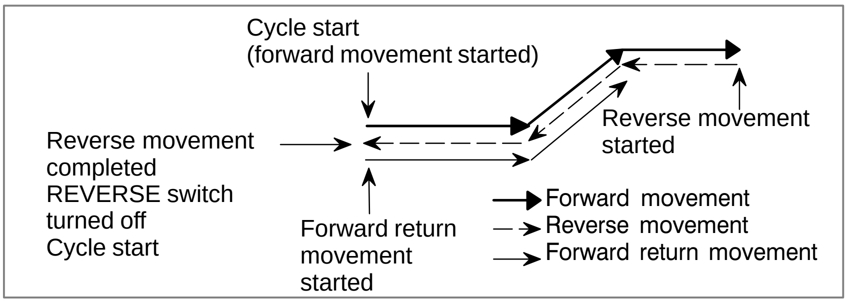


In the case of 3) above, the nozzle starts forward return movement at the position of a feed hold stop when the cycle start switch is pressed.



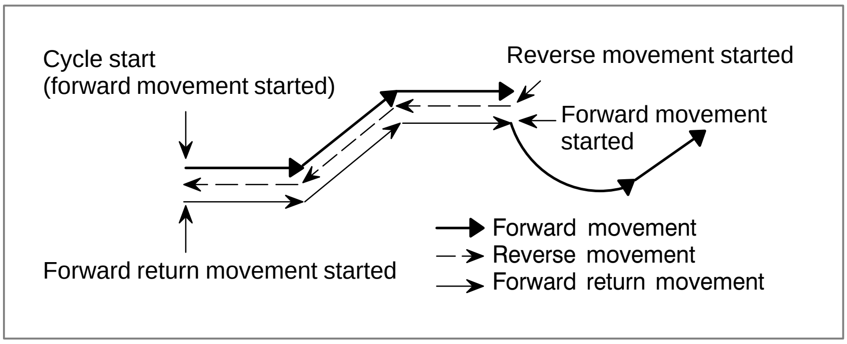
- **Reverse movement → Reverse movement completion → Forward return movement**

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

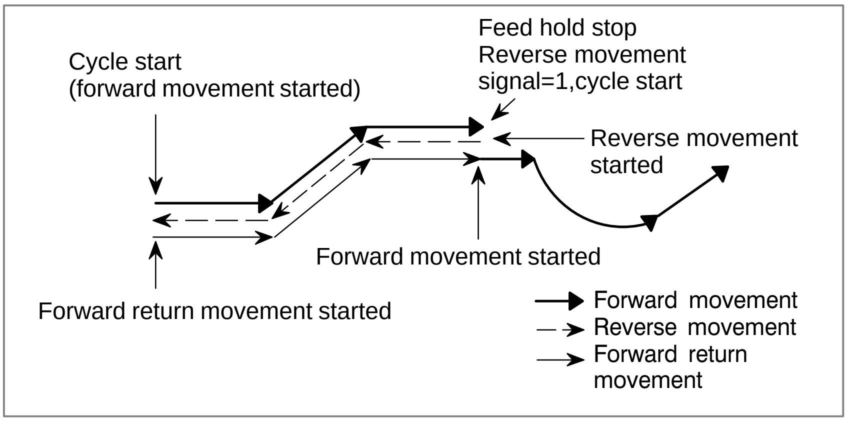


- **Forward return movement → Forward movement**

When the nozzle completes a forward return movement up to the block where reverse movement was started, the nozzle automatically resumes forward movement. Programmed commands are read and program execution is continued. No particular operation is required to resume forward movement.
When beam movement switches from forward return movement to forward movement, the display of RTRY (Re–TRY) in the lower–right corner of the screen disappears.



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



Explanations

- **Forward movement and reverse movement**

In automatic operation, a program is usually executed in the order that commands are specified. This mode of execution is referred to as forward movement. The retrace function can execute in reverse, program blocks that have already been executed. This mode of execution is referred to as reverse movement. In reverse movement, the nozzle can retrace the nozzle path followed by forward movement.
A program can be executed in the reverse direction only for those blocks that have already been executed in the forward direction.
Approximately 40 to 80 blocks can be executed in the reverse direction, depending on the program.
During reverse movement, the REVERSE MOVEMENT LED is on and RVRS blinks in the lower–right corner of the screen to indicate that the nozzle is undergoing reverse movement.
The nozzle can perform reverse movement one block at a time when the single block mode is set.

- **Forward return movement**

The nozzle can be moved again along the retraced path of the blocks in the forward direction up to the block where reverse movement was started. This movement is referred to as forward return movement. In forward return movement, the beam moves along the same path as forward movement up to the position where reverse movement started.
When the beam returns to the block where reverse movement was started, the nozzle resumes forward movement according to the program.
In forward return movement, the REVERSE MOVEMENT LED is off and RTRY (Re–TRY) blinks in the lower–right corner of the screen to indicate that the nozzle is undergoing forward return movement. When the beam switches from forward return movement to forward movement, RTRY (Re–TRY) disappears from the lower–right corner of the screen.
The nozzle can perform forward return movement one block at a time when the single block mode is set.

- **Reverse movement completion**

When there are no more blocks for which to perform reverse movement (when the nozzle has moved back along the path of all memorized blocks or the nozzle has not yet started forward movement), operation stops. This is referred to as reverse movement completion.
Upon reverse movement completion, the REVERSE MOVEMENT LED goes off, and RVED (ReVerse EnD) blinks in the lower–right corner of the screen to indicate that reverse movement is completed.

- **Reset**

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

- **Feedrate**

A feedrate for reverse movement can be specified using parameter (No. 1414). When this parameter is set to 0, the feedrate used for forward movement is used.
For forward return movement, the feedrate for forward movement is always used.
In reverse movement and forward return movement, the feedrate override function, rapid traverse override function, and dry run function are enabled.

Limitations

- **Block that disables reverse movement**

Reverse movement stops when any of the commands or modes listed below appears. If an attempt is made during forward movement to stop forward movement with feed hold stop and then move the beam in the reverse direction when any of the commands and modes below is specified, the reverse movement completion state occurs.

· Cylindrical interpolation (G07.1, G107)
· Polar coordinate interpolation mode (G12.1)
· Inch/metric conversion (G20/ G21)
· Reference position return check (G27)
· Return to reference position (G28)
· Return from reference position (G29)
· 2nd, 3rd, and 4th reference position return (G30)
· Floating reference position return (G30.1)
· Machine coordinate system selection (G53)
· High speed remote buffer A (G05)
· High precision contour control (RISC) (G05P10000)
· AI contour control (G05. IQ1)
· Advanced preview control (G08)
· Spatial circular interpolation (G12)
· Starts the spatial corner rounding mode (G33)
· Stops the spatial corner rounding mode (G34)
· Three–dimensional coordinate conversion (G68)
· Three–dimensional coordinate conversion cancel (G69)
· AI–axis length compensation (G71)
· Three–dimensional conversion (G98)
· Three–dimensional conversion cancel (G99)

- **Circular interpolation(G02,G03)**

Be sure to specify the radius of an arc with R.

> **WARNING**
> If an end point is not correctly placed on an arc (if a leading line is produced) when an arc center is specified using I, J, and K, the nozzle does not perform correct reverse movement.

- **Interrupt–type custom macro**

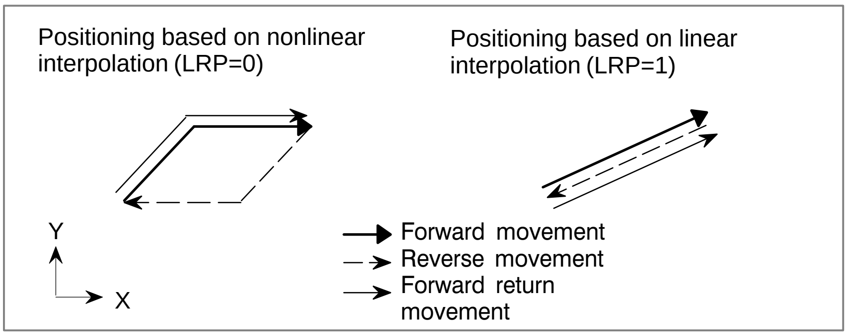
1. Never initiate an interrupt during reverse movement.
2. Never perform reverse movement for an interrupte block and the program that has issure the interrupt.

- **Switching automatic operation mode**

If the operation mode is switched after a single block stop from memory operation to MDI operation or vice versa during reverse movement or forward return movement, reverse movement, forward return movement, and forward movement can no longer be performed. To restart operation, return the mode to the original mode, then press the cycle start switch.

- **Positioning (G00)**

When the nozzle is positioned based on nonlinear interpolation by setting bit 1 (LRP) of parameter No. 1401 to 0, the path of the beam for reverse movement does not match the path for forward movement. The path for forward return movement is the same as the path for forward movement. When the nozzle is positioned based on linear interpolation by setting bit 1 (LRP) of parameter No. 1401 to 1, the path of the beam for reverse movement matches the path for forward movement.



- **Dwell (G04)**

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

- **Programmable data setting (G10)**

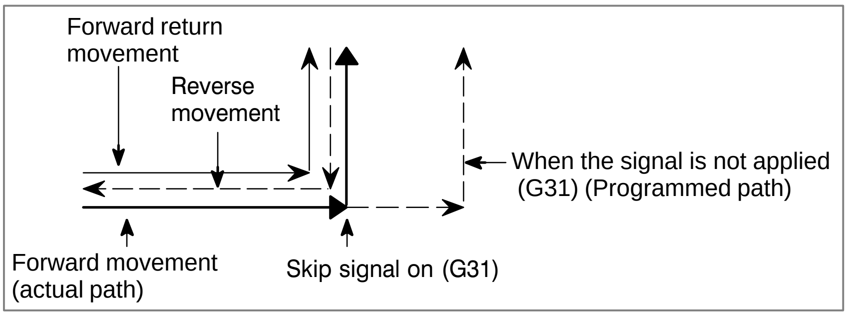
A cutter compensation value, parameter, pitch error data, workpiece zero point offset value, setting specified or modified using the programmable data setting code (G10) are ignored in reverse movement and forward return movement.

- **Stored stroke check function on/off (G22,G23)**

The on/off state of the stored stroke check function present at the end of forward movement remains unchanged during reverse movement and forward return movement. This means that the actual on/off state may differ from the modal G22/G23 indication. When reverse movement or forward return movement is cancelled upon reset, the modal G22/G23 indication at that time becomes valid.
The setting of an area with G22 X_Y_Z_I_J_K at the end of forward movement remains unchanged.

- **Skip funtion (G31)**

In reverse movement and forward return movement, the skip signal is ignored. In reverse movement and forward return movement, the beam moves along the path actually followed in forward movement.



- **Auxiliary function**

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

- **Cutter compensation value**

Even if a cutter compensation value is modified in reverse movement or forward return movement, the beam moves according to the compensation value used when the block was executed in the forward movement.

- **Custom macro operation**

All custom macro operations are ignored in reverse movement and forward return movement.
The values of macro variables present at the end of forward movement remain unchanged.

- **Manual intervention**

When the nozzle has been moved by manual intervention, return the nozzle to the original position before moving the nozzle in the reverse direction after a feed hold stop or single block stop.In reverse movement, the nozzle cannot move along the path made during manual intervention. All movements made by manual intervention are ignored in reverse movement and forward return movement.

- **Mirror image**

When a block with the mirror image function specified by a signal or setting is memorized in forward movement, the mirror image function is eliminated; the block is memorized as originally programmed.
Accordingly, in reverse movement and forward return movement, the beam moves along the programmed path.In reverse movement or forward return movement, the mirror image function can be specified by a signal or setting. When the nozzle performs reverse movement or forward return movement for a block where the mirror image function is specified by the programmable mirror image code (G51.1), the beam moves along the actual path incorporating the mirror image function.

> **CAUTION**
> During reverse movement, the nozzle may touch a workpiece. To prevent this condition from occurring, raise the nozzle, or take some other precautions, before starting reverse movement.

4.11 MANUAL INTERVENTION AND RETURN

In cases such as when beam movement along an axis is stopped by feed hold during automatic operation so that manual intervention can be used to replace the nozzle: When automatic operation is restarted, this function returns the nozzle to the position where manual intervention was started. To use the conventional program restart function, the switches on the operator's panel must be used in conjunction with the MDI keys. This function does not require such operations.

Before this function can be used, MIN (bit 0 of parameter No. 7001) must be set to 1.

Explanations

- **Manual absolute on/off**

In manual absolute off mode, the nozzle 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 nozzle 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**

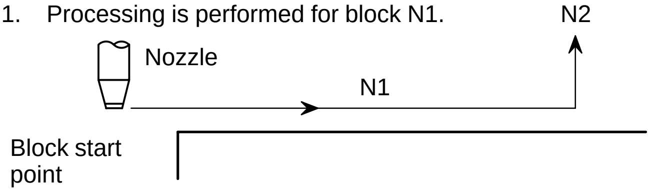
When the nozzle is replaced for a reason, the beam movement cannot be restarted by a changed offset in the middle of the interrupted block.

- **Machine lock, mirror image, and scaling**

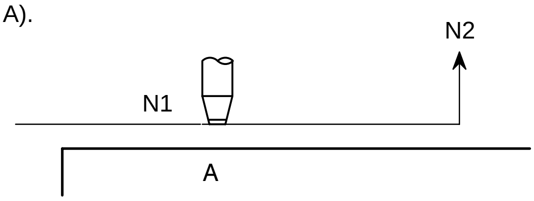
When performing manual intervention, never use the machine lock, mirror image, or scaling functions.

Example

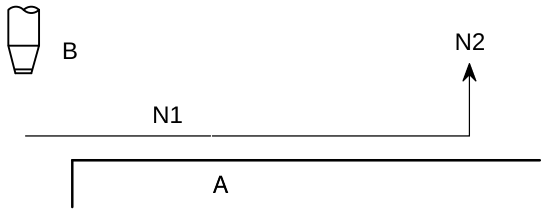
1. Processing is performed for block N1.



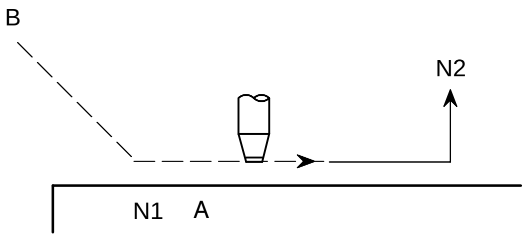
2. The nozzle is stopped by feed hold during the execution of block N1 (point A).



3. The nozzle is manually retracted to point B, then nozzle movement is restarted.



4. The nozzle automatically returns to point A at the dry run feedrate, then the remaining move command in block N1 is executed.



WARNING

When performing manual intervention, pay particular attention of machining and the shape of the workpiece so that the machine and nozzle are not damaged.

4.12 DNC OPERATION WITH MEMORY CARD

4.12.1 Specification

"DNC operation with Memory Card" is a function that it is possible to perform machining with executing the program in the memory card, which is assembled to the memory card interface, where is the left side of the screen.

There are two methods to use this function as follows.

(a) By starting automatic operation (cycle start) during the DNC operation mode (RMT), it is possible to perform machining (DNC operation) while a program is being read from a memory card, as by using the external input/output unit such as a floppy cassette and so on. (Fig.4.12.1 (a))

(b) It is possible to read sub–programs written in the memory card and execute them by the command Subprogram call (M198). (Fig.4.12.1 (b))

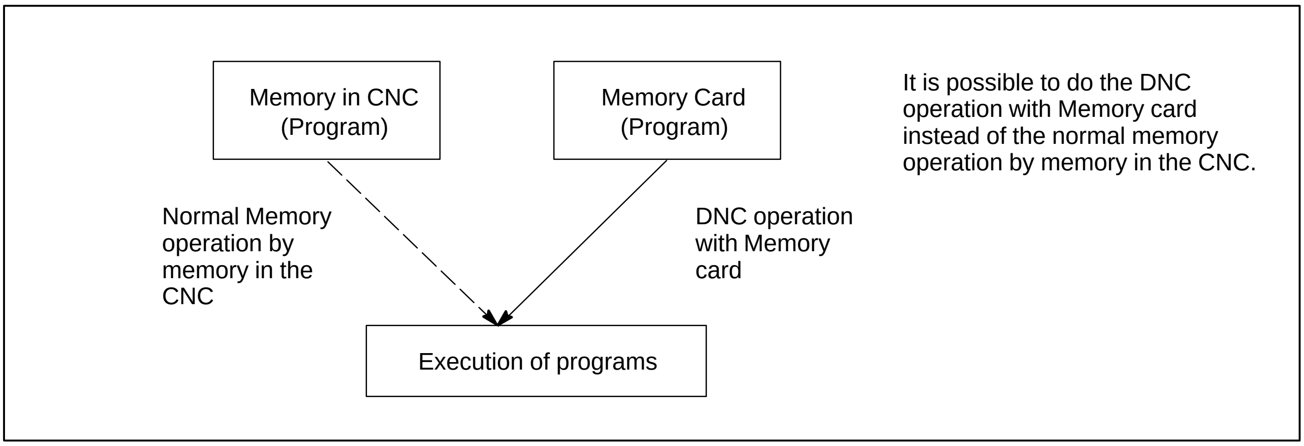


Fig.4.12.1 (a)

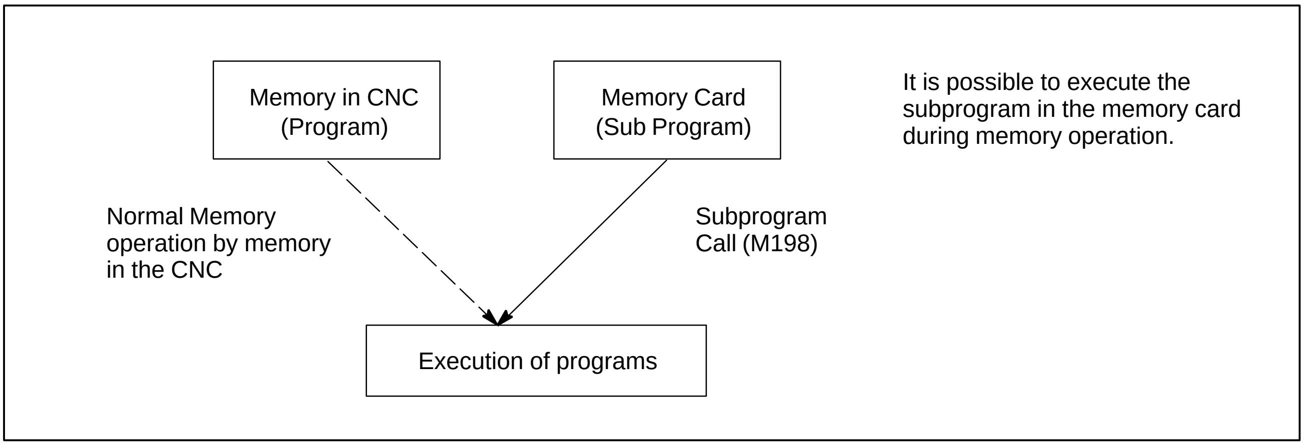


Fig.4.12.1 (b)

NOTE

1 To use this function, it is necessary to set the parameter of No.20 to 4 by setting screen.
No.20 [I/O CHANEL: Setting to select an input/output unit]
Setting value is 4.: It means using the memory card interface.

2 When CNC control unit is a stand–alone type, the memory card interface on the left side of the screen of the display unit is available. But the interface on the control unit is not available.

4.12.2 Operations

4.12.2.1 DNC operation

Handling explanation

Please set the parameter of No.20 to 4 in the setting screen in advance.

(1) Change to RMT mode.

(2) Push [PROGRAM] function key.

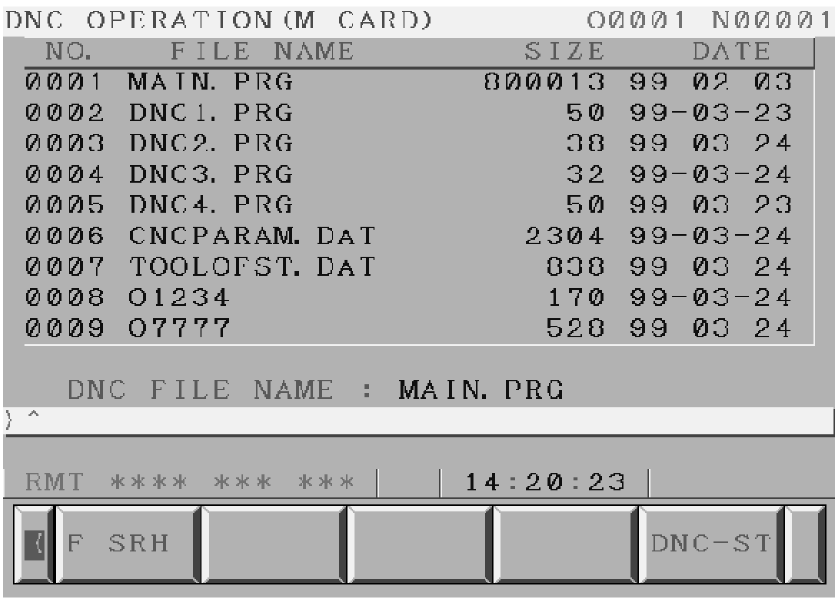
(3) Push [>] soft key (continuance menu).

(4) When [DNC–CD] soft key is pushed, the following screen is displayed.

(5) The screen can be scrolled by page key. An arbitrary file number is input, and [F SRH] soft key is pushed. Then the arbitrary file name is displayed at the top of DNC operation (memory card) screen.

(6) When the file number that is executed is input and the [DNC–ST] soft key is pushed, the file name that is selected is set to DNC FILE.

(7) When the cycle start is done, the program that is selected is executed.



4.12.2.2 Subprogram call (M198)

When the following block in a program in CNC memory is executed, a subprogram file in memory card is called.

Format

1. Normal format

M198 P○○○○ ΔΔΔΔ ;

- M198: Memory card call instruction
- P○○○○: Number of repetition
- ΔΔΔΔ: File number for a file in the memory card

2. FS15 tape format

M198 P○○○○ LΔΔΔΔ ;

- M198: Memory card call instruction
- P○○○○: File number for a file in the memory card
- LΔΔΔΔ: File number of repetition

Explanation

When the custom macro option is provided, both format 1 and 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 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.

NOTE
Please set the parameter of No. 20 to 4 in the setting screen in advance.

4.12.3 Limitation and Notes

(1) The memory card can not be accessed, such as display of memory card list and so on, during the DNC operation with memory card.

(2) It is possible to execute the DNC operation with memory card on multi path system. However, it is not possible to call programs from the plural paths at the same time.

(3) The selection of DNC operation file that is set at DNC OPERATION screen is cleared by the power supply turn off and on. After the power supply is turned on again, it is necessary to select the DNC operation file again.

(4) Please do not pull out and insert memory card during the DNC operation with memory card.

(5) It is not possible to call a program in the memory card from the DNC operation program.

(6) In case of using this function, the PMCIA card attachment written at section 6 must be used to prevent a poor connection of the memory card from occurring by vibration of the machine.

(7) In case of the stand–alone type *i* series that the display unit is a Display link unit, this function can not be used.

(8) The memory card interface on the stand–alone type controller is not available. Please use the memory card interface on the display unit.

4.12.4 Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
0138	DNM							

[Data type] Bit

#7 (DNM) The DNC operation with memory card function is
0 : disable.
1 : enable.

4.12.5 Connecting PCMCIA Card Attachment

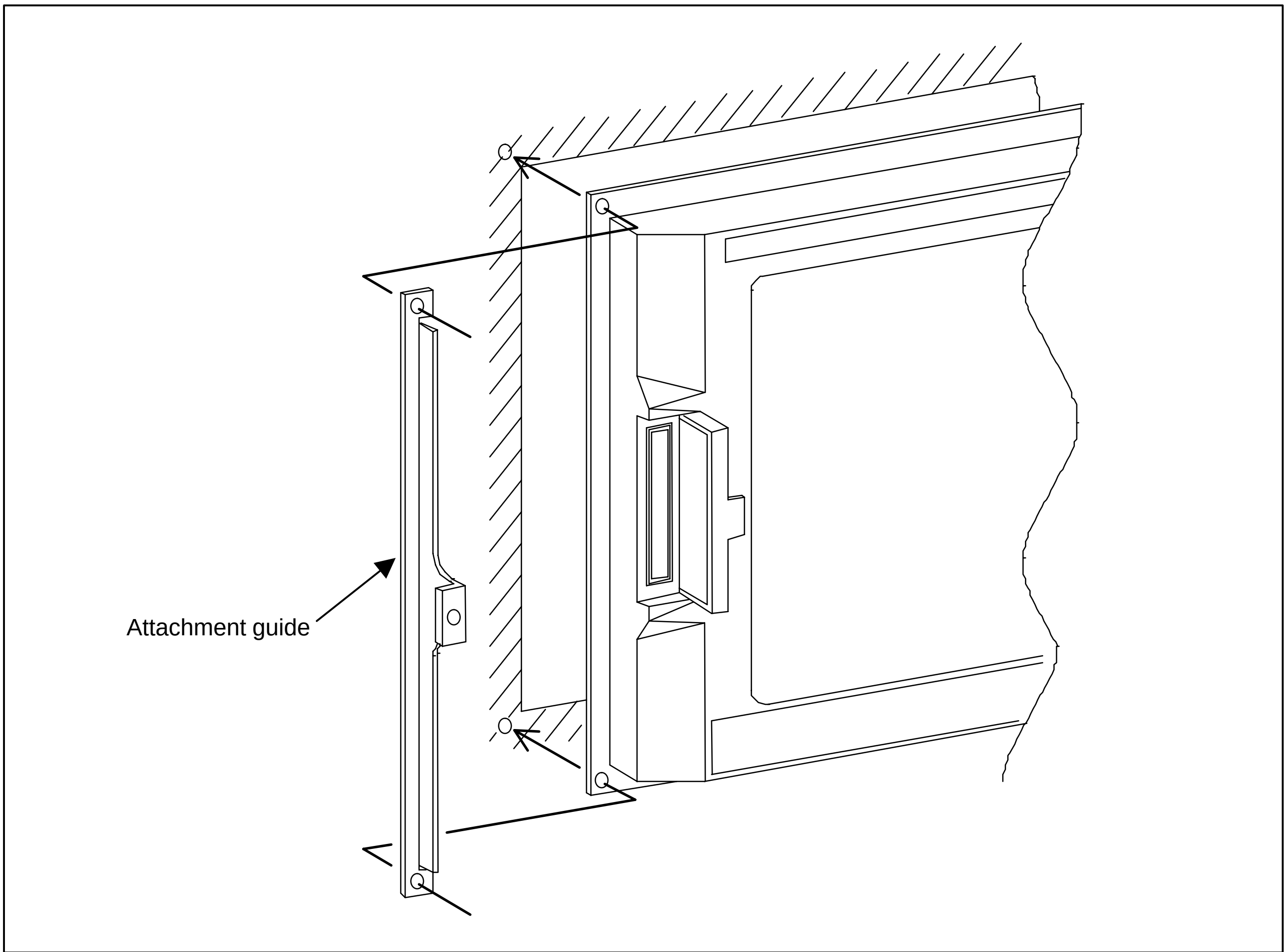
4.12.5.1 Specification number

Specification	Remarks
A02B–0236–K160	For 7.2″ LCD or 8.4″ LCD
A02B–0236–K161	For 9.5″ LCD or 10.4″ LCD

4.12.5.2 Assembling

1) How to assemble to the unit
Assemble an attachment guide and a control unit to the cabinet by screwing together as follow figure.

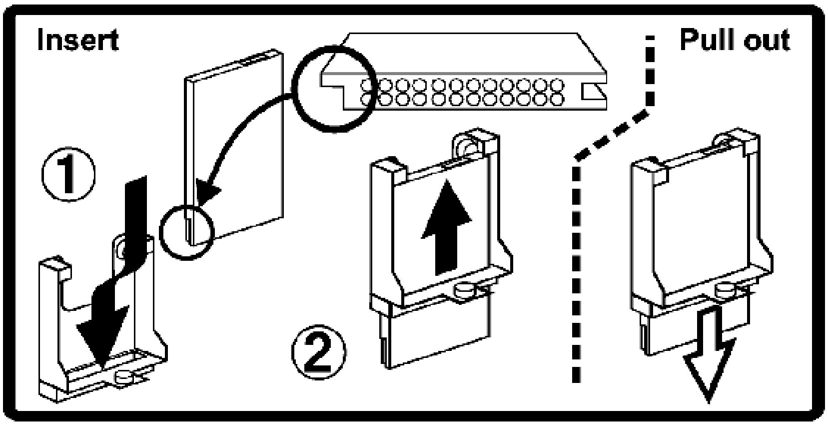
The attachment guide is 1.6mm thick. Pay attention for the length of the screws when you assemble them.



2) How to mount the card

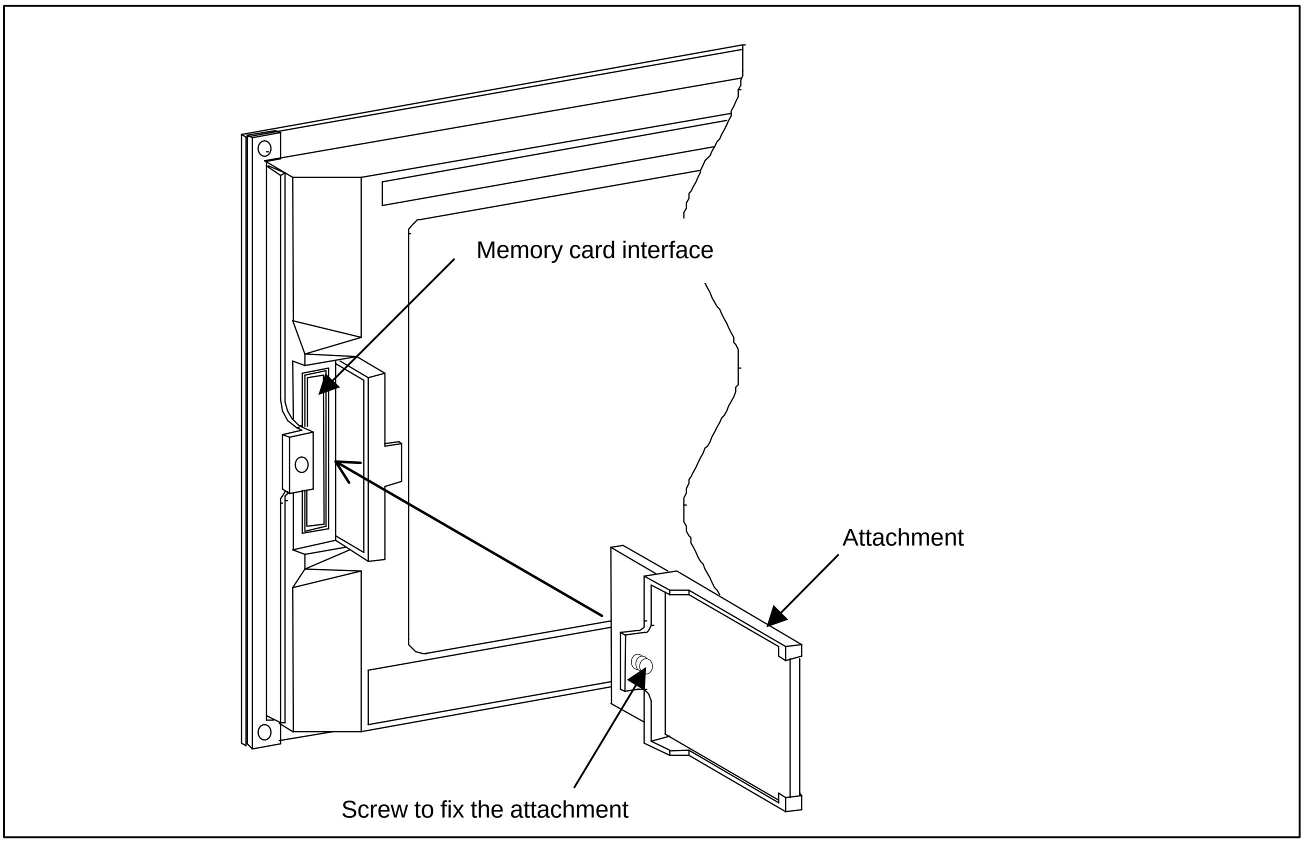
(a) Insert the card to slit of the attachment. Please pay attention to the direction of the card. (Please mach the direction of ditch on the card.)

(b) Push up the card to the upper end of the attachment.



3) Assembling of the attachment

Insert the memory card with the attachment into the memory card interface as following figure. And, fix the attachment guide by screwing the screw of the attachment guide by manual.



4) Appearance after connection

NOTE

1 In both case of stand–alone type *i* series and LCD mounted type *i* series, the memory card interface where is the left side of the screen of the display unit. (The memory card interface on the stand–alone type controller is not available.)

2 It is impossible to assemble the display unit and the attachment guide from inside of the cabinet.

3 The memory card must be used in the condition, as the coolant cannot be poured directly on it.

4.12.6 Recommended Memory Card

Maker	Type	Capacity
Hitachi LTD	HB289016A4	16MB
	HB289032A4	32MB
	HB289160A4	160MB
Matushita electric	BN–012AB	12MB
	BN–020AB	20MB
	BN–040AB	40MB
SanDisk	SDP3B–4	4MB
	SDP3B–20	20MB
	SDP3B–40	40MB

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the nozzle, 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, T, and B (2nd auxiliary function) commands, is available for checking a program together with machine lock.

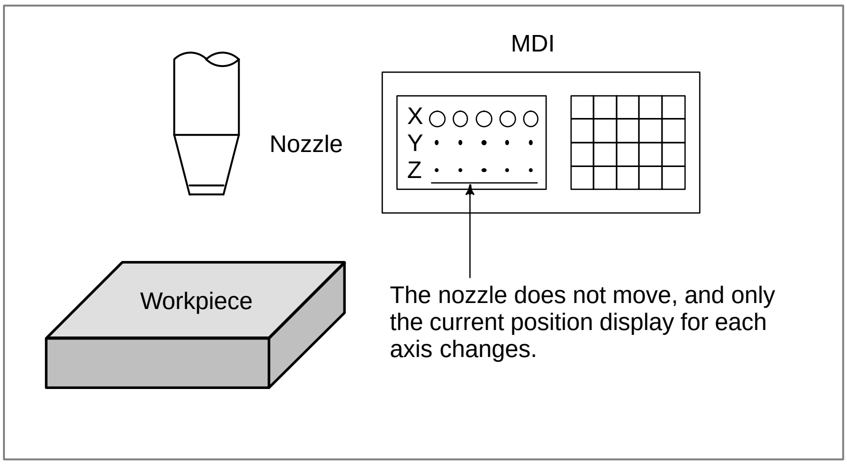


Fig.5.1 Machine lock

Procedure for Machine Lock and Auxiliary Function Lock

- **Machine Lock**

Press the machine lock switch on the operator's panel. The nozzle does not move but the position along each axis changes on the display as if the nozzle 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 nozzle is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.

> **WARNING**
> The positional relationship between the workpiece coordinates and machine coordinates may differ before and after automatic operation using machine lock. In such a case, specify the workpiece coordinate system by using a coordinate setting command or by performing manual reference position return.

- **Auxiliary Function Lock**

Press the auxiliary function lock switch on the operator's panel. M, T, and B codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Restrictions

- **M, T, B command by only machine lock**

M, T, and B 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 nozzle 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, M99, and M198 (subprogram calling function) commands are executed even in the auxiliary function lock state. M codes for calling a subprogram (parameters No. 6071 to 6079) and those for calling a custom macro (parameter No. 6080 to 6089) are also executed.

5.2 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.

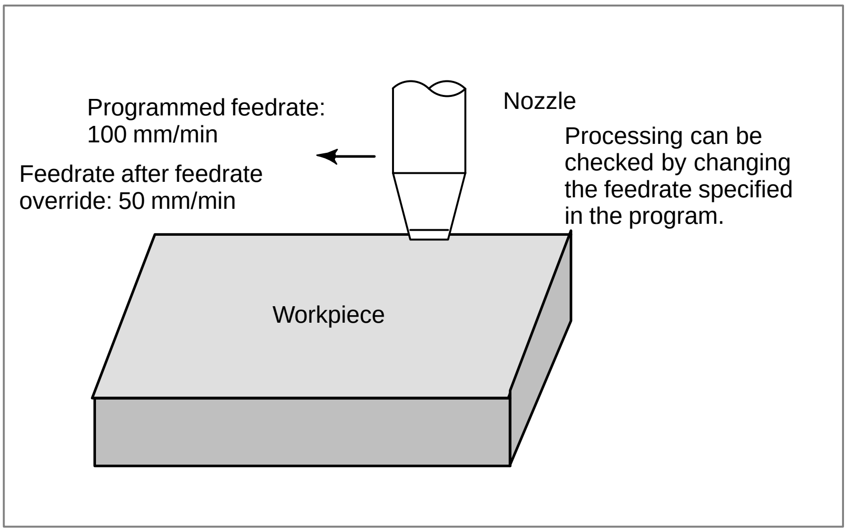
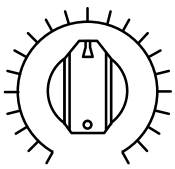


Fig.5.2 Feedrate override

Procedure for Feedrate Override



0 200

JOG FEED RATE 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).

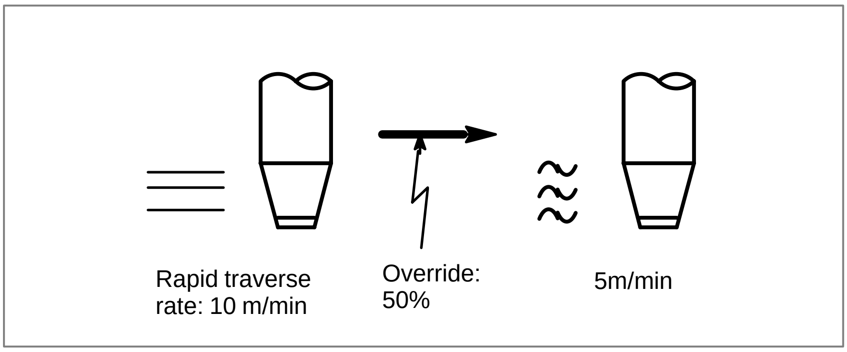
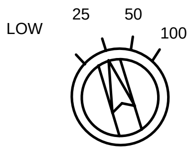


Fig.5.3 Rapid traverse override

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 in G27, G28, G29, G30, G53
3) Manual rapid traverse
4) Rapid traverse of manual reference position return

5.4 DRY RUN

The nozzle 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 beam under the state that the workpiece is removed from the table.

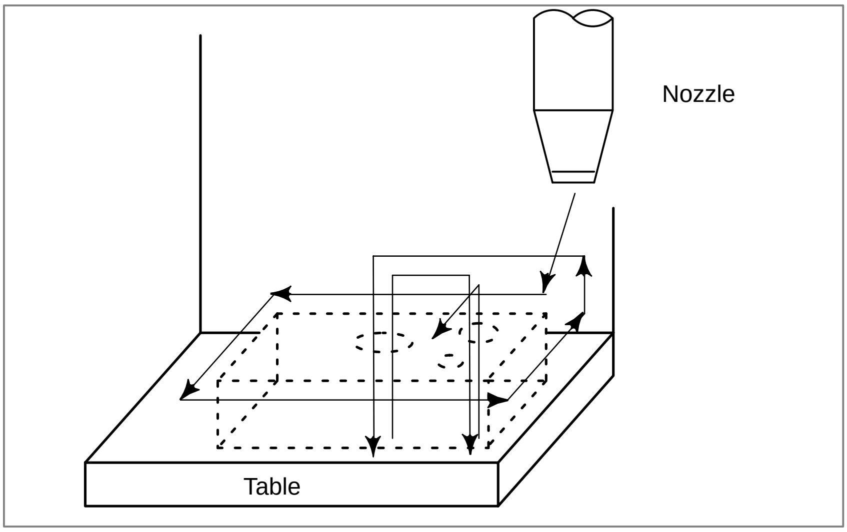


Fig.5.4 Dry run

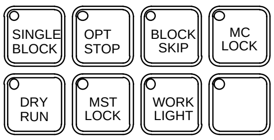
Procedure for Dry Run

Procedure

Press the dry run switch on the machine operator's panel during automatic operation.
The beam 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 × Max.JV ***2)**
OFF	Dry run speed × JV,or rapid traverse rate ***1)**	Dry run feedrate × 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.

JV: Jog feedrate override

*2) Clamped to the maximum cutting feedrate

JVmax: 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 nozzle 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.

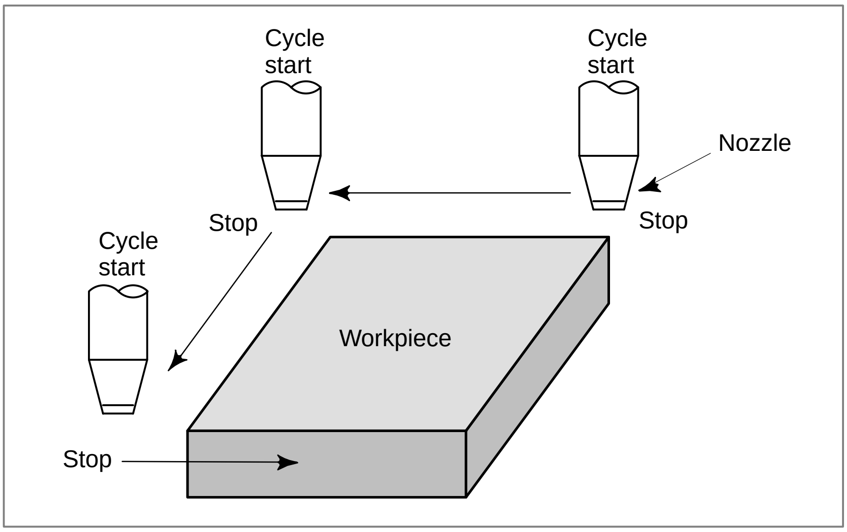


Fig.5.5 Single block

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 nozzle 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.

- **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, P, L.

6 SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the nozzle 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.

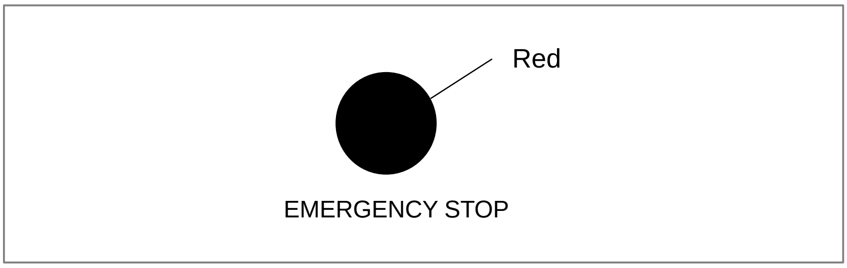


Fig.6.1 Emergency stop

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.
EMERGENCY STOP stops lasing and the machine enters the LRDY state.

6.2 OVERTRAVEL

When the beam tries to move beyond the stroke end set by the machine tool limit switch, the nozzle decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

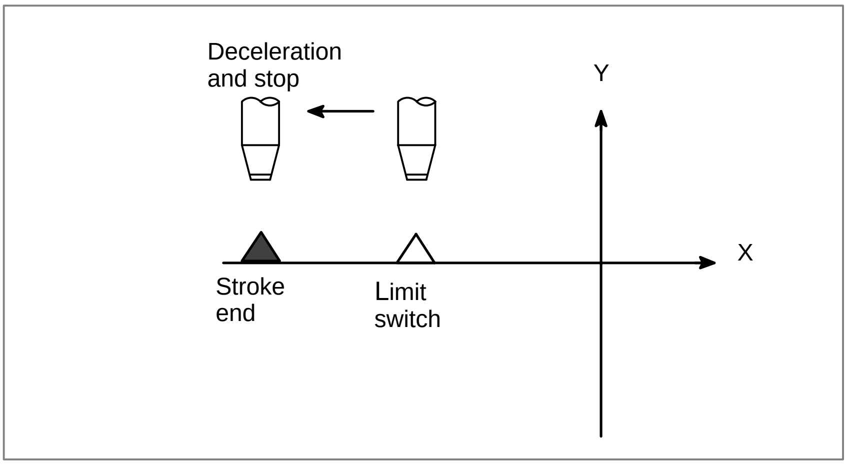


Fig.6.2 Overtravel

Explanation

- **Overtravel during automatic operation**

When the nozzle touches a limit switch along an axis during automatic operation, the nozzle is decelerated and stopped along all axes and an overtravel alarm is displayed.

- **Overtravel during manual operation**

In manual operation, the nozzle is decelerated and stopped only along the axis for which the nozzle has touched a limit switch. The nozzle still moves along the other axes.

- **Releasing overtravel**

Press the reset button to reset the alarm after moving the nozzle to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

Alarm

Alarm No.	Message	Description
506	Overtravel: +n	The nozzle has exceeded the hardware–specified overtravel limit along the positive nth axis (n: 1 to 8).
507	Overtravel: –n	The nozzle has exceeded the hardware–specified overtravel limit along the negative nth axis (n: 1 to 8).

6.3 STROKE CHECK

Two forbidden areas for tool can be specified with stored stroke check 1, stored stroke check 2, and stored stroke check 3.

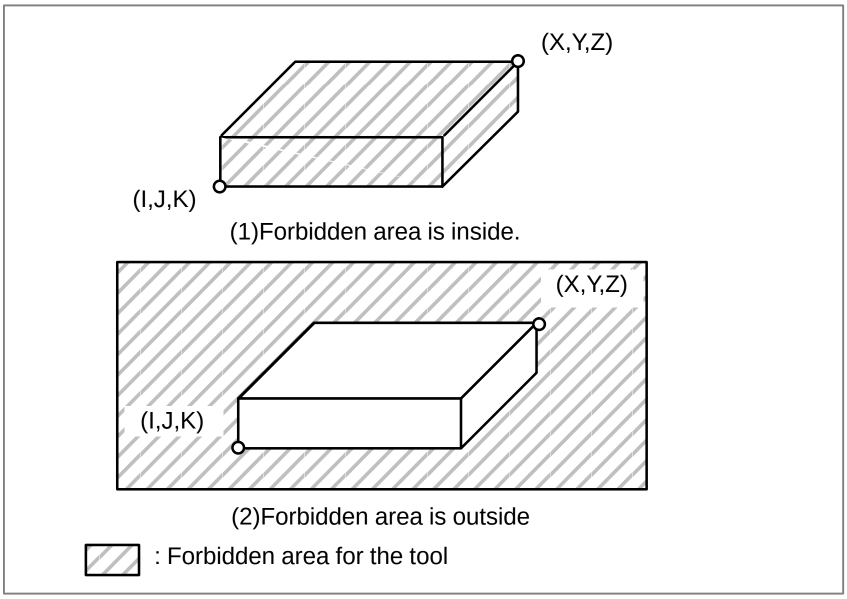


Fig.6.3 (a) Stroke check

When the tool exceeds a stored stroke limit, an alarm is displayed and the nozzle is decelerated and stopped.
When the nozzle enters a forbidden area and an alarm is generated, the nozzle can be moved in the reverse direction from which the nozzle came.

Explanation

- **Stored stroke check 1**

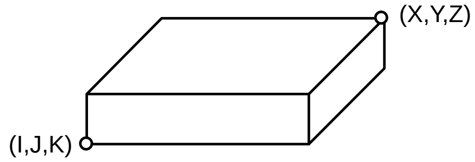
Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.

- **Stored stroke check 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 nozzle to enter the forbidden area, and a G23 command permits the nozzle 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_;



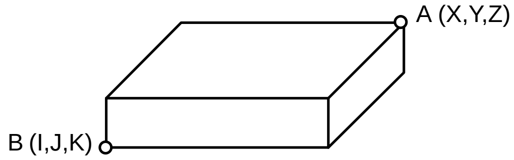
X>I, Y>J, Z>K
X–I >ζ (In least command increment)
Y–J >ζ (In least command increment)
Z–K >ζ ((In least command increment)

$$\zeta\ (mm) = \frac{F}{7500}$$

F=Rapid traverse speed (mm/min)

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 >ζ (In least command increment)
Y–J >ζ (In least command increment)
Z–K >ζ ((In least command increment)

$$\zeta\ (mm) = \frac{F}{7500}$$

F=Rapid traverse speed (mm/min)

Fig.6.3 (c) Creating or changing the forbidden area using a parameters

In stored stroke check 2, even if you mistake the order of the coordinate value of the two points, a rectangular, with the two points being the apexes, will be set as the area.
The X, Y, Z, I, J, and K values set in parameter Nos. 1322 and 1323 must be coordinate values in the machine coordinate system, and they must be set in the least command increment (output unit). The X, Y, Z, I, J, and K values programmed using G22 must be coordinate values in the machine coordinate system, and they must be set in the least input increment (input unit).
The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

- **Stored stroke check 3**

Set the boundary with parameters No. 1324 and 1325. The area inside the boundary becomes the forbidden area.

- **Checkpoint for the forbidden area**

Depending on the result of checking whether the nozzle enters the forbidden area or which part of the nozzle enters the forbidden area, the way of measuring X, Y, Z, I, J, and K varies.
If point A (The top of the nozzle) 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 nozzle chuck) is checked, the distance “b” must be set. When checking the nozzle tip (like point A), and if the nozzle length varies for each nozzle, setting the forbidden area for the longest nozzle requires no re–setting and results in safe operation.

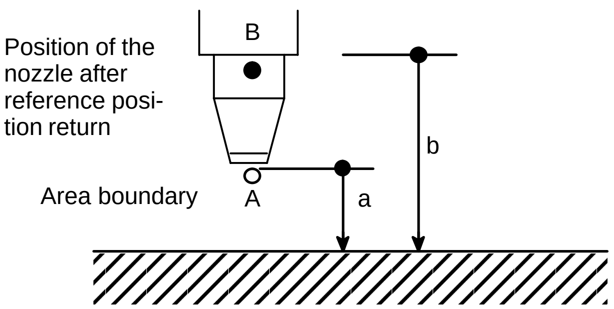


Fig.6.3 (d) Setting the forbidden area

- **Forbidden area over lapping**

Area can be set in piles.

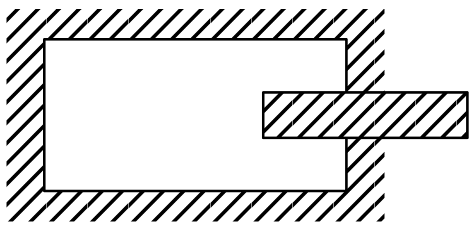


Fig.6.3 (e) Setting the forbidden area over lapping

Unnecessary limits should be set beyond the machine stroke.

- **Overrun amount of stored stroke limit**

If the maximum rapid traverse rate is F (mm/min), the maximum overrun amount, L (mm), of the stored stroke limit is obtained from the following expression:

L (mm) = F/7500

The nozzle enters the specified inhibited area by up to L (mm). Bit 7 (BFA) of parameter No. 1300 can be used to stop the nozzle when it reaches a point L mm short of the specified area. In this case, the nozzle will not enter the inhibited area.

- **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 enters a forbidden area and an alarm is generated, the nozzle can be moved only in the backward direction. To cancel the alarm, move the nozzle backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the nozzle 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.

- **Timing for displaying an alarm**

Parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the nozzle enters the forbidden area or immediately after the nozzle has entered the forbidden area.

NOTE

In setting a forbidden area, if the two points to be set are the same, the area is as follows:

(1) When the forbidden area is stored stroke check 1, all areas are forbidden areas.

(2) When the forbidden area is stored stroke check 2 or stored stroke check 3, all areas are movable areas.

Alarms

Alarm Number	Message	Contents
500	OVER TRAVEL: +n	Exceeded the n–th axis (1–8) + side stored stroke limit I.
501	OVER TRAVEL: –n	Exceeded the n–th axis (1–8) – side stored stroke limit I.
502	OVER TRAVEL: +n	Exceeded the n–th axis (1–8) + side stored stroke limit II.
503	OVER TRAVEL: –n	Exceeded the n–th axis (1–8) – side stored stroke limit II.
504	OVER TRAVEL: +n	Exceeded the n–th axis (1–8) + side stored stroke limit III.
505	OVER TRAVEL: –n	Exceeded the n–th axis (1–8) – side stored stroke limit III.

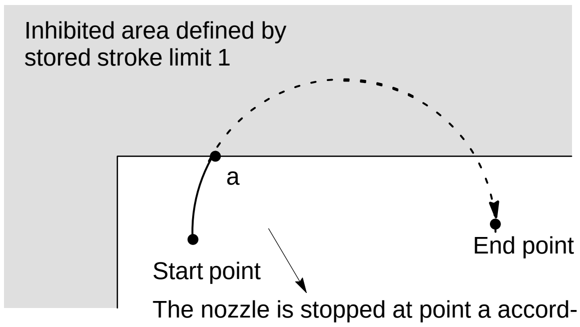
6.4 STROKE LIMIT CHECK PRIOR TO PERFORMING MOVEMENT

During automatic operation, before the movement specified by a given block is started, whether the nozzle enters the inhibited area defined by stored stroke limit 1 is checked by determining the position of the end point from the current position of the X, Y, and Z axes and a specified amount of travel. If the nozzle is found to enter the inhibited area defined by a stored stroke limit, the nozzle is stopped immediately upon the start of movement for that block, and an alarm is displayed.

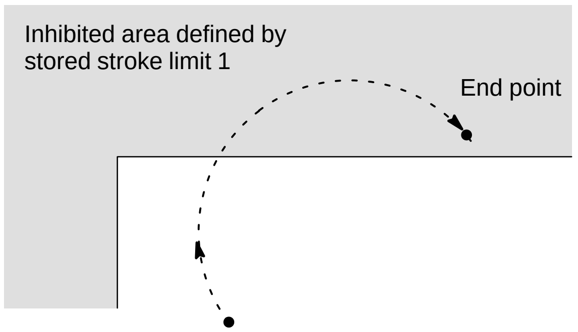
WARNING

Whether the coordinates of the end point, reached as a result of traversing the distance specified in each block, are in a inhibited area is checked. In this case, the path followed by a move command is not checked. However, if the nozzle enters the inhibited area defined by stored stroke limit 1, an alarm is issued. (See the examples below.)

Example 1)



The nozzle is stopped at point a according to stored stroke limit 1.



Immediately upon movement commencing from the start point, the nozzle is stopped to enable a stroke limit check to be performed before movement.

Explanations

When a stroke limit check prior to movement is performed, whether to check the movement performed by a G31 (skip) block can be determined using NPC (bit 2 of parameter No. 15600).

Limitations

- **Machine lock**

If machine lock is applied at the start of movement, no stroke limit check made before movement is performed.

- **Program restart**

When a program is restarted, an alarm is issued if the restart position is within a inhibited area.

- **Manual intervention following a feed hold stop**

When the execution of a block is restarted after manual intervention following a feed hold stop, no alarm is issued even if the end point following a manual intervention is within a inhibited area.

- **A block consisting of multiple operations**

If a block consisting of multiple operations is executed, an alarm is issued at the start point of any operation whose end point falls within a inhibited area.

- **Simple synchronous control**

In simple synchronous control, only the master axis is checked; no slave axes are checked.

- **Three–dimensional coordinate conversion**

In three–dimensional coordinate conversion mode, no check is made.

- **Drawing**

No check is performed while drawing is being performed as part of dynamic graphic display (only drawing (no machining) is being performed).

- **PMC axis control**

No check is made for a movement based on PMC axis control.

- **High–speed high–precision contour control (HPCC)**

No check is made for a movement based on high–speed, high–precision contour control (HPCC).

Alarm

Number	Message	Contents
4700	PROGRAM ERROR (OT +)	The value specified in the X-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4701	PROGRAM ERROR (OT –)	The value specified in the X-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4702	PROGRAM ERROR (OT +)	The value specified in the Y-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4703	PROGRAM ERROR (OT –)	The value specified in the Y-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4704	PROGRAM ERROR (OT +)	The value specified in the Z-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4705	PROGRAM ERROR (OT –)	The value specified in the Z-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)

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                                  O000  O0000

     100     PARAMETER WRITE ENABLE
     510     OVER TR1AVEL  :+X
     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        XIK                   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 : P/S alarm (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 749 : Overheat alarms
No. 900 to 999 : System alarms
No. 4000 to 4999 : Laser alarms
No. 5000 to : P/S alarm (Program errors)

* 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)
(4)Page No.

3 Change the page by the 1–page change key.

4 To delete the recorded information, press the softkey **[(OPRT)]** then the **[DELETE]** key.

```
 ALARM HISTORY                                  O0100  N00001
(1)01.02.14 16:43:48                                 PAGE=1
  (2)010  (3)MPROPER G–CODE                            (4)
 01.02.13 8:22:21
  506   OVER TRAVEL : +1
 01.02.12 20:15:43
  417 SERVO ALARM : X AXIS DGTL PARAM




   MEM  ****  ***  ***              19 : 47 : 45
 [ ALARM ][ MSG ][ HISTRY ][      ][ (OPRT) ]
```

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

Procedure

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, 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
010	PUNCHING	Data being output via reader puncher interface
011	READING	Data being input via reader puncher interface
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.

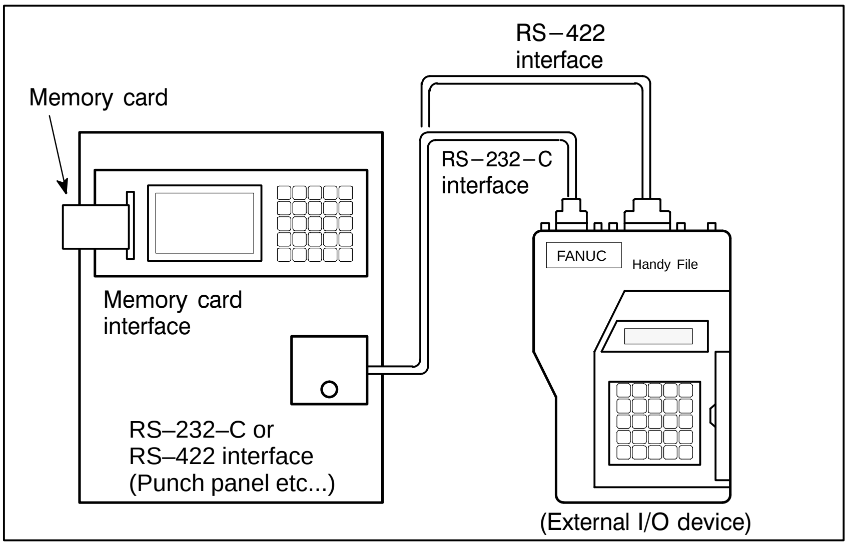
No.	Display	Meaning of data
030	CHARACTER NUMBER TH DATA	The position of the character which caused TH alarm is displayed by the number of characters from the beginning of the block at TH alarm
031	TH DATA	Read code of character which caused TH alarm

8 DATA INPUT/OUTPUT

NC data is transferred between the NC and external input/output devices such as the Handy File.
The memory card interface located to the left of the display can be used to read information on a memory card in the CNC or write it to the card.
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 III–2 “**OPERATIONAL DEVICES**”.



8.1 FILES

Of the external input/output devices, the FANUC Handy File use floppy disks as their input/output medium.
In this manual, these input/output medium is generally referred to as a floppy.
Unlike an NC tape, a floppy allows the user to freely choose from several types of data stored on one medium on a file–by–file basis.
Input/output is possible with data extending over more than one floppy disk.

Explanations

• What is a File

The unit of data, which is input/output between the floppy and the CNC by one input/output operation (pressing the VREADW or VPUNCHW key), is called a HfileI. When inputting CNC programs from, or outputting them to the floppy, for example, one or all programs within the CNC memory are handled as one file.
Files are assigned automatically file numbers 1,2,3,4 and so on, with the lead file as 1.

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.

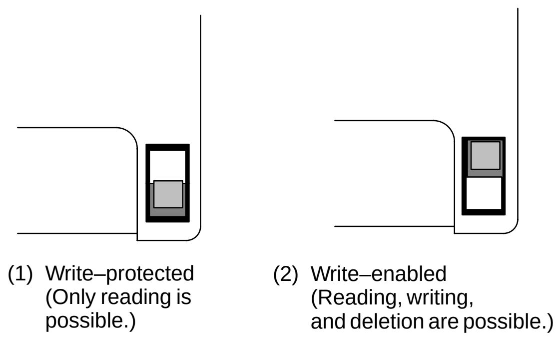


Fig.8.1 Protect swtich

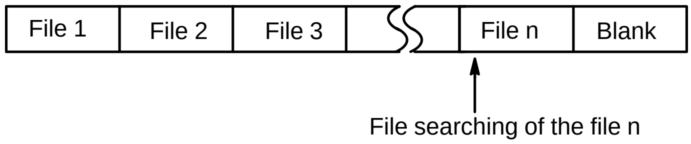
- **Writing memo**

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 III–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 EDIT or MEMORY switch on the machine operator's panel.

2 Press function key PROG , then the program contents display screen or program check screen appears.

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

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 No.)

8.3 FILE DELETION

Files stored on a floppy can be deleted file by file as required.

File deletion

Procedure

1. Insert the floppy into the input/output device so that it is ready for writing.
2. Press the EDIT switch on the machine operator's panel.
3. Press function key PROG , then the program contents display screen appears.
4. Press soft key **[(OPRT)]**
5. Press the rightmost soft key ▷ (next–menu key).
6. Enter address N.
7. Enter the number (from 1 to 9999) of the file to delete.
8. Press soft key **[DELETE]** and then press soft key **[DELETE]**.
   The file specified in step **7** is deleted.

Explanations

- **File number after the file is deleted**

When a file is deleted, the file numbers after the deleted file are each decremented by one. Suppose that a file numbered k was deleted. In this case, files are renumbered as follows:

Before deletion	after deletion
1 to (k>1)	1 to (k>1)
k	Deleted
(k+1) to n	k to (n>1)

- **Protect switch**

Set the write protect switch to the write enable state to delete the files.

8.4 PROGRAM INPUT/OUTPUT

8.4.1 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 **III–8.2**.

4 Press function key PROG , then the program contents display screen or program directory screen appears.

5 Press soft key **[(OPRT)]**.

6 Press the rightmost soft key ▷ (next–menu key).

7 After entering address O, specify a program number to be assigned to the program. When no program number is specified here, the program number used on the floppy or NC tape is assigned.

8 Press soft keys **[READ]** and **[EXEC]**
The program is input and the program number specified in step **7** is assigned to the program.

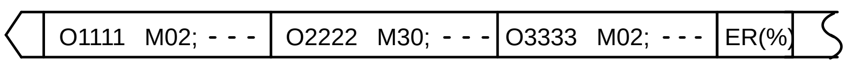
Explanations

- **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 (P/S 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.

- When a program is entered with a program number
The O–number on the NC tape is ignored and the specified number is assigned to the program. When the program is followed by additional programs, the first additional program is given the program number. Additional program numbers are calculated by adding one to the last 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.

[(OPRT)] [BG–END]

- **Additional program input**

You can input a program to be appended to the end of a registered program.

Registered program	Input program	Program after input
○1234 ;	○5678 ;	○1234 ;
□□□□□□□ ;	○○○○○○○ ;	□□□□□□□ ;
□□□□□ ;	○○○○○ ;	□□□□□ ;
□□□□ ;	○○○○ ;	□□□□ ;
□□□ ;	○○○ ;	□□□ ;
%	%	%
		○5678 ;
		○○○○○○○ ;
		○○○○○ ;
		○○○○ ;
		○○○ ;
		%

In the above example, all lines of program O5678 are appended to the end of program O1234. In this case, program number O5678 is not registered. When inputting a program to be appended to a registered program, press the **[READ]** soft key without specifying a program number in step 8. Then, press the **[CHAIN]** and **[EXEC]** soft keys.

- In entire program input, all lines of a program are appended, except for its O number.
- When canceling additional input mode, press the reset key or the [CAN] or [STOP] soft key.
- Pressing the [CHAIN] soft key positions the cursor to the end of the registered program. Once a program has been input, the cursor is positioned to the start of the new program.
- Additional input is possible only when a program has already been registered.

- **Defining the same program number as that of an existing program**

If an attempt has been made to register a program having the same number as that of a previously registered program, P/S alarm 073 is issued and the program cannot be registered.

Alarm

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.

8.4.2 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 , then the program contents display screen or program directory screen appears.
5. Press soft key **[(OPRT)]**.
6. Press the rightmost soft key ▷ (next–menu key).
7. Enter address O.
8. Enter a program number. If –9999 is entered, all programs stored in memory are output.
   To output multiple programs at one time, enter a range as follows :
   OΔΔΔΔ,O□□□□
   Programs No.ΔΔΔΔ to No.□□□□ are output.
   The program library screen displays program numbers in ascending order when bit 4 (SOR) of parameter No. 3107 is set to 1.
9. Press soft keys **[PUNCH]** and **[EXEC]**
   The specified program or programs 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 P/S alarm (No. 86) 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.

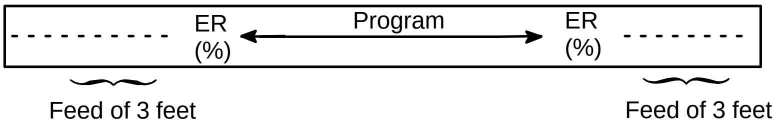
- **Punching programs in the background**

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.
<O> (Program No.) **[PUNCH] [EXEC]**: Punches out a specified program.
<O> H–9999I **[PUNCH] [EXEC]**: 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

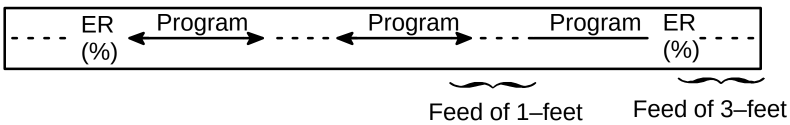
By setting NCR (bit 3 of parameter No. 0100), CRs can be omitted so that each LF appears without a CR.

- **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.

8.5 OFFSET DATA INPUT AND OUTPUT

8.5.1 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 III– 8.5.2. 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 III–8.2.

4 Press function key OFFSET SETTING, then the cutter compensation screen appears.

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.

8.5.2 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, then the cutter compensation screen appears.

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

(1) For cutter compensation memory A
G10 L11 P_R_;
where P_: Offset No.
R_: Cutter compensation amount

(2) For cutter compensation memory B
Setting/changing the geometric compensation amount
G10 L10 P_R_;
Setting/changing the wear compensation amount
G10 L11 P_R_;

(3) For cutter compensation memory C
Setting/changing the geometric compensation amount for H code
G10 L10 P_R_;
Setting/changing the geometric compensation amount for D code
G10 L12 P_R_;
Setting/changing the wear compensation amount for H code
G10 L11 P_R_;
Setting/changing the wear compensation amount for D code
G10 L13 P_R_;

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.

8.6 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.

8.6.1 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 **III–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 **III–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, then the setting screen appears.

6 Enter 1 in response to the prompt for "PARAMETER WRITE (PWE)" in setting data. Alarm P/S100 (indicating that parameters can be written) appears.

7 Press soft key SYSTEM .

8 Press chapter selection soft key **[PARAM]**, then the parameter screen appears.

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 in response to the prompt for "PARAMETER WRITE (PWE)" in setting data.

15 Turn the power to the CNC back on.

16 Release the EMERGENCY STOP button on the machine operator's panel.

8.6.2 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 , then the parameter screen appears.

5 Press chapter selection soft key **[PARAM]**.

6 Press soft key **[(OPRT)]**.

7 Press rightmost soft key ▷ (next–menu key).

8 Press soft keys **[PUNCH]**.

9 To output all parameters, press the **[ALL]** soft key. To output only parameters which are set to other than 0, press the **[NON–0]** soft key.

10 Press soft key **[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 .

- **Suppressing output of parameters set to 0**

To suppress the output of the following parameters, press the **[PUNCH]** soft key then **[NON–0]** soft key.

	Other than axis type	Axis type
Bit type	Parameter for which all bits are set to 0	Parameter for an axis for which all bits are set to 0.
Value type	Paramter whose value is 0.	Parameter for an axis for which the value is 0.

- **Output file name**

When the floppy disk directory display function is used, the name of the output file is PARAMETER.
Once all parameters have been output, the output file is named ALL PARAMETER. Once only parameters which are set to other than 0 have been output, the output file is named NON–0. PARAMETER.

8.6.3 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 **III–8.6.4.** 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 **III–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 in response to the prompt for writing parameters (PWE). Alarm P/S100 (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 in response to the prompt for "PARAMETER WRITE (PWE)" in 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 Nos. 3620 to 3624 and pitch error compensation data must be set correctly to apply pitch error compensation correctly (See **III–11.5.2**).

8.6.4 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 **"PITCH ERROR"**.

8.7 INPUTTING/ OUTPUTTING CUSTOM MACRO COMMON VARIABLES

8.7.1 Inputting Custom Macro Common Variables

The value of a custom macro common variable (#500 to #999) 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 **III–8.7.2.** 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 which has been output, as described in Section III–8.7.2, in memory according to the program input procedure described in Section III–8.4.1.

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 check 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. (When the option is added as the common variables, #500 to #999 can be input and output.)
When the option for adding a common variable is specified, values from #500 to #999 can be input and output. #100 to #149 (When the option is added as the common variables, #100 to #199 can be input and output.) can be input and output when bit 3 (PV5) of parameter No. 6001 is set to 1.

8.7.2 Outputting Custom Macro Common Variable

Custom macro common variables (#500 to #999) 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. (When the option is added as the common variables, #500 to #999.)
When the option for adding a common variable is specified, values from #500 to #999 can be input and output.
#100 to #149 (When the option is added as the common variables, #100 to #149.) can be input and output when bit 3 (PU5) of parameter No. 6001 is set to 1.

8.8 DISPLAYING DIRECTORY OF FLOPPY CASSETTE

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
   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 : 51 : 12
[ PRGRM ][        ][  DIR  ][        ][ (OPRT) ]
```

8.8.1 Displaying the Directory

Displaying the directory of floppy cassette 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 the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FLOPPY]**.

5 Press page key ↑ PAGE or PAGE ↓ .

6 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.8.8.1 (a)

7 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 the rightmost soft key ▷ (next–menu key).
4. Press soft key **[FLOPPY]**.
5. Press soft key **[(OPRT)]**.
6. Press soft key **[F SRH]**.
7. Enter a file number.
8. Press soft keys **[F SET]** and **[EXEC]**.
9. Press a page key to display another page of the directory.
10. Press soft key **[CAN]** to return to the soft key display shown in the screen of **Fig.8.8.1 (a)**.

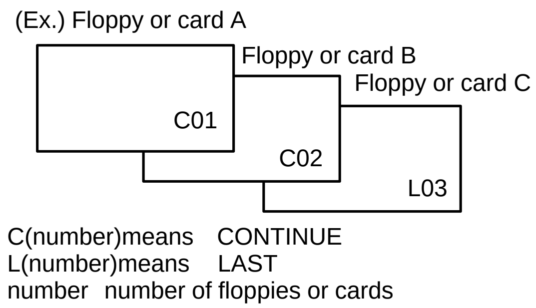
```
DIRECTORY (FLOPPY)                    O0001  N00000
  NO. FILE NAME                        (METER) VOL
   0005  O0040                               1.3
   0006  O0050                               1.9
   0007  O0100                               1.9
   0008  O1000                               1.9
   0009  O9500                               1.6


SEARCH
 FILE NO. =
>_
 EDIT  ****  ***  ***            11 : 54 : 19
( F SET )(       )(       )( CAN )( EXEC )
```

Fig.8.8.1 (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.



8.8.2 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 the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FLOPPY]**.

5 Press soft key **[(OPRT)]**.

6 Press soft key **[READ]**.

```
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
READ
 FILE NO. =                           PROGRAM NO. =
>_
EDIT   * * * *   * * *   * * *          11 : 55 : 04
[ F SET ][ O SET ][ STOP ][ CAN ][ EXEC ]
```

7 Enter a file number.

8 Press soft key **[F SET]**.

9 To modify the program number, enter the program number, then press soft key **[O SET]**.

10 Press soft key **[EXEC]**. The file number indicated in the lower–left corner of the screen is automatically incremented by one.

11 Press soft key **[CAN]** to return to the soft key display shown in the screen of **Fig.8.8.1 (a)**.

8.8.3 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 the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FLOPPY]**.

5 Press soft key **[(OPRT)]**.

6 Press soft key **[PUNCH]**.

```
DIRECTORY (FLOPPY)                      O0002  N01000
  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
PUNCH
 FILE NO. =                        PROGRAM NO. =
>_
EDIT  ****  ***  ***               11 : 55 : 26
( F SET )( O SET )( STOP )( CAN )( EXEC )
```

7 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.

8 Press soft key **[O SET]**.

9 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]**.

10 Press soft key **[CAN]** to return to the soft key display shown in the screen of Fig.8.8.1 (a).

8.8.4 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 the rightmost soft key ▷ (next–menu key).

4 Press soft key **[FLOPPY]**.

5 Press soft key **[(OPRT)]**.

6 Press soft key **[DELETE]**.

```
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
DELETE
 FILE NO. =          NAME=
>_
EDIT  ****  ***  ***                  11 : 55 : 51
( F SET )( F NAME )(          )(  CAN  )(  EXEC  )
```

7 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]**.

8 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.

9 Press soft key **[CAN]** to return to the soft key display shown in the screen of **Fig.8.8.1 (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 (No. 102).
Set the I/O device number to parameter (No. 112) 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

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.

8.9 OUTPUTTING A PROGRAM LIST FOR A SPECIFIED GROUP

CNC programs stored in memory can be grouped according to their names, thus enabling the output of CNC programs in group units. Section III–11.3.2 explains the display of a program listing for a specified group.

Procedure for Outputting a Program List for a Specified Group

Procedure

1 Display the program list screen for a group of programs, as described in Section III–11.3.2.

```
PROGRAM DIRECTORY (GROUP)              O0001  N00010

        PROGRAM (NUM.)          MEMORY (CHAR.)
        USED:          60                 3321
        FREE:           2                  429
   O0020  (GEAR–1000  MAIN     )
   O0040  (GEAR–1000  SUB–1    )
   O0200  (GEAR–1000  SUB–2    )
   O2000  (GEAR–1000  SUB–3    )



 >_
 EDIT  ****  ***  ***  ***        16 : 52 : 13
[ PRGRM ][  DIR  ][       ][       ][ (OPRT) ]
```

2 Press the **[(OPRT)]** operation soft key.

```
[BG–ED][O–SRH][     ][     ][GROUP]
```

3 Press the right–most soft key ▷ (continuous menu key).

```
[     ][READ][PUNCH][     ][     ]
```

4 Press the **[PUNCH]** operation soft key.

```
[AL–GRP][     ][STOP][CAN][EXEC]
```

5 Press the **[AL–GRP]** operation soft key.

The CNC programs in the group for which a search is made are output. When these programs are output to a floppy disk, they are output to a file named GROUP.PROGRAM.

8.10 DATA INPUT/OUTPUT ON THE ALL IO SCREEN

To input/output a particular type of data, the corresponding screen is usually selected. For example, the parameter screen is used for parameter input from or output to an external input/output unit, while the program screen is used for program input or output. However, programs, parameters, offset data, and macro variables can all be input and output using a single common screen, that is, the ALL IO screen.

```
READ/PUNCH (PROGRAM)                          O1234 N12345

I/O CHANNEL                3      TV CHECK            OFF
DEVICE NUM.                0      PUNCH CODE          ISO
BAUDRATE                4800      INPUT CODE        ASCII
STOP BIT                   2      FEED OUTPUT        FEED
NULL INPUT (EIA)          NO      EOB OUTPUT (ISO)     CR
TV CHECK (NOTES)          ON      BAUDRATE CLK.     INNER
CD CHECK (232C)          OFF      RESET/ALARM          ON
PARITY BIT               OFF      SAT COMMAND        HOST
INTERFACE              RS422      COM PROTCOL           A
END CODE                 EXT      COM CODE          ASCII
(0:EIA   1:ISO)>1_
 MDI   ****   ***   ***   ***                12:34:56
[ PRGRM ][ PARAM ][OFFSET][ MACRO ][ (OPRT) ]
```

Fig.8.10 ALL IO screen (when channel 3 is being used for input/output)

8.10.1 Setting Input/Output–Related Parameters

Input/output–related parameters can be set on the ALL IO screen. Parameters can be set, regardless of the mode.

Setting input/output–related parameters

Procedure

1 Press function key SYSTEM .

2 Press the rightmost soft key ▷ (next–menu key) several times.

3 Press soft key **[ALL IO]** to display the ALL IO screen.

> **NOTE**
> 1 If program or floppy is selected in EDIT mode, the program directory or floppy screen is displayed.
> 2 When the power is first turned on, program is selected by default.

```
READ/PUNCH (PROGRAM)                         O1234 N12345

I/O CHANNEL              3     TV CHECK            OFF
DEVICE NUM.              0     PUNCH CODE          ISO
BAUDRATE              4800     INPUT CODE        ASCII
STOP BIT                 2     FEED OUTPUT        FEED
NULL INPUT (EIA)        NO     EOB OUTPUT (ISO)     CR
TV CHECK (NOTES)        ON     BAUDRATE CLK.     INNER
CD CHECK (232C)        OFF     RESET/ALARM          ON
PARITY BIT             OFF     SAT COMMAND        HOST
INTERFACE            RS422     COM PROTCOL           A
END CODE               EXT     COM CODE          ASCII
(0:EIA  1:ISO)>1_
 MDI   ****  ***  ***  ***               12:34:56
[PRGRM] [PARAM] [OFFSET] [MACRO] [(OPRT)]
```

> **NOTE**
> Baud rate clock, CD check (232C), reset/alarm report, and the parity bit for parameter No. 134, as well as the communication code, end code, communication protocol, interface, and SAT command for parameter No. 135 are displayed only when channel 3 is being used for input/output.

4 Select the soft key corresponding to the desired type of data (program, parameter, and so forth).

5 Set the parameters corresponding to the type of input/output unit to be used. (Parameter setting is possible regardless of the mode.)

8.10.2 Inputting and Outputting Programs

A program can be input and output using the ALL IO screen.
When entering a program using a cassette or card, the user must specify the input file containing the program (file search).

File search

Procedure

1 Press soft key **[PRGRM]** on the ALL IO screen, described in Section 8.10.1.

2 Select **EDIT** mode. A program directory is displayed.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

- A program directory is displayed only in EDIT mode. In all other modes, the ALL IO screen is displayed.

```
                                              O0001 N00010

             PROGRAM (NUM.)          MEMORY (CHAR.)
       USED   :      60                  3321
       FREE   :       2                   429

    O0010 O0001 O0003 O0002 O0555 O0999
    O0062 O0004 O0005 O1111 O0969 O6666
    O0021 O1234 O0588 O0020 O0040



  >_
  EDIT  ****  ***  ***  ***              14:46:09
 ( F SRH )( READ )( PUNCH )( DELETE )( (OPRT) )
```

4 Enter address N.

5 Enter the number of the file to be found.

- N0

  The first floppy file is found.
- One of N1 to N9999

  Among the files numbered from 1 to 9999, a specified file is found.
- N–9999

  The file immediately after that used most recently is found.
- N–9998

  When –9998 is specified, the next file is found. Then, each time a file input/output operation is performed, N–9999 is automatically inserted. This means that subsequent files can be sequentially found automatically.

  This state is canceled by specifying N0, N1 to N9999, or N–9999, or upon a reset.

( ) ( ) ( ) (CAN) (EXEC)

6 Press soft keys **[F SRH]** and **[EXEC]**.
The specified file is found.

Explanations

- **Difference between N0 and N1**

When a file already exists in a cassette or card, specifying N0 or N1 has the same effect. If N1 is specified when there is no file on the cassette or card, an alarm is issued because the first file cannot be found. Specifying N0 places the head at the start of the cassette or card, regardless of whether the cassette/card already contains files. So, no alarm is issued in this case. N0 can be used, for example, when a program is written into a new cassette or card, or when a previously used cassette or card is used once all the files it contains have been erased.

- **Alarm issue during file search**

If an alarm (file search failure, for example) is generated during file search, the CNC does not issue an alarm immediately. However, a P/S alarm (No. 086) is issued if input/output is subsequently performed on that file.

- **File search using N–9999**

Instead of sequentially searching for files by specifying actual file numbers every time, the user can specify the first file number, then find the subsequent files by specifying N–9999. When N–9999 is specified, the time required for file search can be reduced.

Inputting a program

Procedure

1 Press soft key **[PRGRM]** on the ALL IO screen, described in Section 8.10.1.

2 Select EDIT mode. A program directory is displayed.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

· A program directory is displayed only in EDIT mode. In all other modes, the ALL IO screen is displayed.

```
                                          O0001 N00010

            PROGRAM (NUM.)          MEMORY (CHAR.)
      USED   :      60                    3321
      FREE   :       2                     429

   O0010 O0001 O0003 O0002 O0555 O0999
   O0062 O0004 O0005 O1111 O0969 O6666
   O0021 O1234 O0588 O0020 O0040



 >_
 EDIT  ****  ***  ***  ***               14:46:09
 [ F SRH ][ READ ][ PUNCH ][ DELETE ][ (OPRT) ]
```

4 To specify a program number to be assigned to an input program, enter address O, followed by the desired program number.
If no program number is specified, the program number in the file or on the NC tape is assigned as is.

```
 [      ][      ][STOP][ CAN ][EXEC]
```

5 Press soft key **[READ]**, then **[EXEC]**.
The program is input with the program number specified in step **4** assigned.
To cancel input, press soft key **[CAN]**.
To stop input prior to its completion, press soft key **[STOP]**.

Outputting programs

Procedure

1 Press soft key **[PRGRM]** on the ALL IO screen, described in Section 8.10.1.

2 Select EDIT mode. A program directory is displayed.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

· A program directory is displayed only in EDIT mode. In all other modes, the ALL IO screen is displayed.

```
                                          O0001 N00010

           PROGRAM (NUM.)          MEMORY (CHAR.)
     USED   :     60                    3321
     FREE   :      2                     429

  O0010 O0001 O0003 O0002 O0555 O0999
  O0062 O0004 O0005 O1111 O0969 O6666
  O0021 O1234 O0588 O0020 O0040



 >_
 EDIT  ****  ***  ***  ***                14:46:09
 [ F SRH ][ READ ][PUNCH ][DELETE][ (OPRT) ]
```

4 Enter address O.

5 Enter a desired program number.
If –9999 is entered, all programs in memory are output.
To output a range of programs, enter OΔΔΔΔ, O□□□□.. The programs numbered from ΔΔΔΔ to □□□□ are output.
When bit 4 (SOR) of parameter No. 3107 for sorted display is set to 1 on the program library screen, programs are output in order, starting from those having the smallest program numbers.

```
[    ][    ][STOP][ CAN ][EXEC]
```

6 Press soft key **[PUNCH]**, then **[EXEC]**.
The specified program or programs are output. If steps **4** and **5** are omitted, the currently selected program is output.
To cancel output, press soft key **[CAN]**.
To stop output prior to its completion, press soft key **[STOP]**.

Deleting files

Procedure

1 Press soft key **[PRGRM]** on the ALL IO screen, described in Section 8.10.1.

2 Select EDIT mode. A program directory is displayed.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

· A program directory is displayed only in EDIT mode. In all other modes, the ALL IO screen is displayed.

```
                                          O0001 N00010

              PROGRAM (NUM.)          MEMORY (CHAR.)
        USED   :      60                  3321
        FREE   :       2                   429

   O0010 O0001 O0003 O0002 O0555 O0999
   O0062 O0004 O0005 O1111 O0969 O6666
   O0021 O1234 O0588 O0020 O0040

  >_
  EDIT  ****  ***  ***  ***               14:46:09
 [ F SRH ][ READ ][PUNCH ][DELETE][ (OPRT) ]
```

4 Press soft key **[DELETE]**.

5 Enter a file number, from 1 to 9999, to indicate the file to be deleted.

```
[     ][     ][     ][CAN][EXEC]
```

6 Press soft key **[EXEC]**.
The k–th file, specified in step 5, is deleted.

Explanations

- **File numbers after deletion**

After deletion of the k–th file, the previous file numbers (k+1) to n are decremented by 1 to k to (n–1).

Before deletion	After deletion
1 to (k–1)	1 to (k–1)
K	Delete
(k+1) to n	k to (n–1)

- **Write protect**

Before a file can be deleted, the write protect switch of the cassette must be set to make the cassette writable.

8.10.3 Inputting and Outputting Parameters

Parameters can be input and output using the ALL IO screen.

Inputting parameters

Procedure

1 Press soft key **[PARAM]** on the ALL IO screen, described in Section 8.10.1.

2 Select EDIT mode.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

```
READ/PUNCH (PARAMETER)                     O1234 N12345

I/O CHANNEL              3      TV CHECK           OFF
DEVICE NUM.              0      PUNCH CODE         ISO
BAUDRATE              4800      INPUT CODE       ASCII
STOP BIT                 2      FEED OUTPUT       FEED
NULL INPUT (EIA)        NO      EOB OUTPUT (ISO)    CR
TV CHECK (NOTES)        ON      BAUDRATE CLK.    INNER
CD CHECK (232C)        OFF      RESET/ALARM         ON
PARITY BIT             OFF      COM CODE         ASCII
END CODE               EXT      COM PROTCOL          A
INTERFACE            RS422      SAT COMMAND       HOST
(0:EIA 1:ISO)>1_
 MDI  ****  ***  ***  ***               12:34:56
[        ][ READ ][ PUNCH ][        ][        ]
```

```
[    ][    ][    ][ CAN ][ EXEC ]
```

4 Press soft key **[READ]**, then **[EXEC]**.
The parameters are read, and the ”INPUT” indicator blinks at the lower–right corner of the screen. Upon the completion of input, the ”INPUT” indicator is cleared from the screen.
To cancel input, press soft key **[CAN]**.

Outputting parameters

Procedure

1 Press soft key **[PARAM]** on the ALL IO screen, described in Section 8.10.1.

2 Select EDIT mode.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

```
READ/PUNCH (PARAMETER)                         O1234 N12345

I/O CHANNEL                3     TV CHECK             OFF
DEVICE NUM.                0     PUNCH CODE           ISO
BAUDRATE                4800     INPUT CODE         ASCII
STOP BIT                   2     FEED OUTPUT         FEED
NULL INPUT (EIA)          NO     EOB OUTPUT (ISO)      CR
TV CHECK (NOTES)          ON     BAUDRATE CLK.      INNER
CD CHECK (232C)          OFF     RESET/ALARM           ON
PARITY BIT               OFF     COM CODE           ASCII
END CODE                 EXT     COM PROTCOL            A
INTERFACE              RS422     SAT COMMAND         HOST
(0:EIA 1:ISO)>1_
 MDI  ****  ***  ***  ***                 12:34:56
[      ][ READ ][ PUNCH ][      ][      ]
```

```
[      ][      ][      ][ CAN ][EXEC]
```

4 Press soft key **[PUNCH]**, then **[EXEC]**.
The parameters are output, and the ”OUTPUT” indicator blinks at the lower–right corner of the screen. Upon the completion of output, the ”OUTPUT” indicator is cleared from the screen.
To cancel output, press soft key **[CAN]**.

8.10.4 Inputting and Outputting Offset Data

Offset data can be input and output using the ALL IO screen.

Inputting offset data

Procedure

1 Press soft key **[OFFSET]** on the ALL IO screen, described in Section 8.10.1.

2 Select EDIT mode.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

```
READ/PUNCH (OFFSET)                         O1234 N12345

I/O CHANNEL              3      TV CHECK          OFF
DEVICE NUM.              0      PUNCH CODE        ISO
BAUDRATE              4800      INPUT CODE      ASCII
STOP BIT                 2      FEED OUTPUT      FEED
NULL INPUT (EIA)        NO      EOB OUTPUT (ISO)   CR
TV CHECK (NOTES)        ON      BAUDRATE CLK.   INNER
CD CHECK (232C)        OFF      RESET/ALARM        ON
PARITY BIT             OFF      COM CODE        ASCII
END CODE               EXT      COM PROTCOL         A
INTERFACE            RS422      SAT COMMAND      HOST
(0:EIA 1:ISO)>1_
 MDI  ****  ***  ***  ***                 12:34:56
[        ][  READ  ][ PUNCH  ][        ][        ]
```

```
[        ][        ][        ][  CAN  ][ EXEC  ]
```

4 Press soft key **[READ]**, then **[EXEC]**.
The offset data is read, and the "INPUT" indicator blinks at the lower–right corner of the screen.
Upon the completion of input, the "INPUT" indicator is cleared from the screen.
To cancel input, press soft key **[CAN]**.

Outputting offset data

Procedure

1 Press soft key **[OFFSET]** on the ALL IO screen, described in Section 8.10.1.

2 Select EDIT mode.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

```
READ/PUNCH (OFFSET)                          O1234 N12345

I/O CHANNEL              3      TV CHECK            OFF
DEVICE NUM.              0      PUNCH CODE          ISO
BAUDRATE              4800      INPUT CODE        ASCII
STOP BIT                 2      FEED OUTPUT        FEED
NULL INPUT (EIA)        NO      EOB OUTPUT (ISO)     CR
TV CHECK (NOTES)        ON      BAUDRATE CLK.     INNER
CD CHECK (232C)        OFF      RESET/ALARM          ON
PARITY BIT             OFF      COM CODE          ASCII
END CODE               EXT      COM PROTCOL           A
INTERFACE            RS422      SAT COMMAND        HOST
(0:EIA 1:ISO)>1_
 MDI   ****  ***  ***  ***                 12:34:56
[      ][ READ ][PUNCH][      ][      ]
```

```
[      ][      ][      ][ CAN ][EXEC]
```

4 Press soft key **[PUNCH]**, then **[EXEC]**.
The offset data is output, and the ”OUTPUT” indicator blinks at the lower–right corner of the screen. Upon the completion of output, the ”OUTPUT” indicator is cleared from the screen.
To cancel output, press soft key **[CAN]**.

8.10.5 Outputting Custom Macro Common Variables

Custom macro common variables can be output using the ALL IO screen.

Outputting custom macro common variables

Procedure

1 Press soft key **[MACRO]** on the ALL IO screen, described in Section 8.10.1.

2 Select EDIT mode.

3 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

```
READ/PUNCH (MACRO)                          O1234 N12345

I/O CHANNEL                  3     TV CHECK            OFF
DEVICE NUM.                  0     PUNCH CODE          ISO
BAUDRATE                  4800     INPUT CODE        ASCII
STOP BIT                     2     FEED OUTPUT        FEED
NULL INPUT (EIA)            NO     EOB OUTPUT (ISO)     CR
TV CHECK (NOTES)            ON     BAUDRATE CLK.     INNER
CD CHECK (232C)            OFF     RESET/ALARM          ON
PARITY BIT                 OFF     COM CODE          ASCII
END CODE                   EXT     COM PROTCOL           A
INTERFACE                RS422     SAT COMMAND        HOST
(0:EIA 1:ISO)>1_
 MDI   ****  ***  ***  ***                 12:34:56
[        ][ READ ][ PUNCH ][        ][        ]
```

```
[    ][    ][    ][ CAN ][ EXEC ]
```

4 Press soft key **[PUNCH]**, then **[EXEC]**.
The custom macro common variables are output, and the ”OUTPUT” indicator blinks at the lower–right corner of the screen. Upon the completion of output, the ”OUTPUT” indicator is cleared from the screen.
To cancel output, press soft key **[CAN]**.

NOTE
To input a macro variable, read the desired custom macro statement as a program, then execute the program.

8.10.6 Inputting and Outputting Floppy Files

The ALL IO screen supports the display of a directory of floppy files, as well as the input and output of floppy files.

Displaying a file directory

Procedure

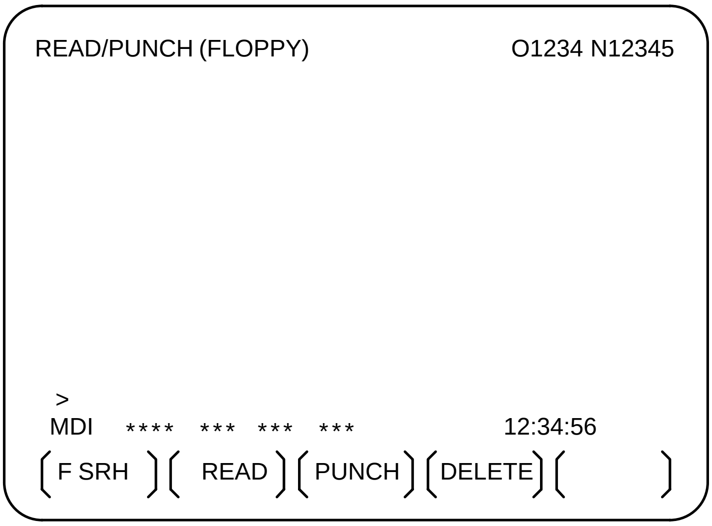
1 Press the rightmost soft key (next–menu key) on the ALL IO screen, described in Section 8.10.1.

2 Press soft key **[FLOPPY**].

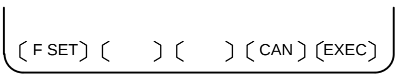
3 Select EDIT mode. The floppy screen is displayed.

4 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.

- The floppy screen is displayed only in EDIT mode. In all other modes, the ALL IO screen is displayed.



5 Press soft key **[F SRH]**.



6 Enter the number of the desired file, then press soft key **[F SET]**.

7 Press soft key **[EXEC]**. A directory is displayed, with the specified file uppermost. Subsequent files in the directory can be displayed by pressing the page key.

```
READ/PUNCH (FLOPPY)                          O1234 N12345
  No.    FILE NAME                           (Meter)  VOL
  0001   PARAMETER                            46.1
  0002   ALL.PROGRAM                          12.3
  0003   O0001                                 1.9
  0004   O0002                                 1.9
  0005   O0003                                 1.9
  0006   O0004                                 1.9
  0007   O0005                                 1.9
  0008   O0010                                 1.9
  0009   O0020                                 1.9
                                               1.9
  F SRH
    File No.=2
  >2_
 EDIT  ****  ***  ***  ***                 12:34:56
[ F SRH ][        ][        ][  CAN   ][  EXEC  ]
```

A directory in which the first file is uppermost can be displayed simply by pressing the page key. (Soft key **[F SRH]** need not be pressed.)

Inputting a file

Procedure

1 Press the rightmost soft key [▷] (next–menu key) on the ALL IO screen, described in Section 8.10.1.

2 Press soft key **[FLOPPY]**.

3 Select EDIT mode. The floppy screen is displayed.

4 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.
The floppy screen is displayed only in EDIT mode. In all other modes, the ALL IO screen is displayed.

```
READ/PUNCH (FLOPPY)                    O1234 N12345





 >
 MDI   ****  ***  ***  ***          12:34:56
[ F SRH ][ READ ][ PUNCH ][DELETE][        ]
```

5 Press soft key **[READ]**.

```
[ F SET ][O SET ][STOP ][ CAN ][EXEC ]
```

6 Enter the number of a file or program to be input.

· Setting a file number: Enter the number of the desired file, then press soft key **[F SET]**.

· Setting a program number: Enter the number of the desired program, then press soft key **[O SET]**.

7 Press soft key **[EXEC]**.
The specified file or program is read, and the ”INPUT” indicator blinks at the lower–right corner of the screen. Upon the completion of input, the ”INPUT” indicator is cleared from the screen.

Outputting a file

Procedure

1 Press the rightmost soft key ▷ (next–menu key) on the ALL IO screen, described in Section 8.10.1.

2 Press soft key **[FLOPPY]**.

3 Select EDIT mode. The floppy screen is displayed.

4 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.
The floppy screen is displayed only in EDIT mode. In all other modes, the ALL IO screen is displayed.

```
READ/PUNCH (FLOPPY)                     O1234 N12345




 >
 MDI   ****  ***  ***  ***              12:34:56
(F SRH  )(  READ )(PUNCH)(DELETE)(        )
```

5 Press soft key **[PUNCH]**.

```
(F SET) (O SET) (STOP) (CAN) (EXEC)
```

6 Enter the number of the program to be output, together with a desired output file number.

- Setting a file number: Enter the number of the desired file, then press soft key **[F SET]**.
- Setting a program number: Enter the number of the desired program, then press soft key **[O SET]**.

7 Press soft key **[EXEC]**.
The specified program is output, and the ”OUTPUT” indicator blinks at the lower–right corner of the screen. Upon the completion of output, the ”OUTPUT” indicator is cleared from the screen.
If no file number is specified, the program is written at the end of the currently registered files.

Deleting a file

Procedure

1 Press the rightmost soft key [▷] (next–menu key) on the ALL IO screen, described in Section 8.10.1.

2 Press soft key **[FLOPPY]**.

3 Select EDIT mode. The floppy screen is displayed.

4 Press soft key **[(OPRT)]**. The screen and soft keys change as shown below.
The floppy screen is displayed only in EDIT mode. In all other modes, the ALL IO screen is displayed.

```
READ/PUNCH (FLOPPY)                    O1234 N12345




 >
 MDI   ****  ***  ***  ***            12:34:56
[ F SRH ][ READ ][ PUNCH ][ DELETE ][       ]
```

5 Press soft key **[DELETE]**.

```
[ F SET ][     ][     ][ CAN ][ EXEC ]
```

6 Enter the number of the desired file, then press soft key **[F SET]**.

7 Press soft key **[EXEC]**. The specified file is deleted. After the file has been deleted, the subsequent files are shifted up.

8.11 DATA INPUT/OUTPUT USING A MEMORY CARD

By setting the I/O channel (parameter No. 20) to 4, files on a memory card can be referenced, and different types of data such as part programs, parameters, and offset data on a memory card can be input and output in text file format.
The major functions are listed below.

- Displaying a directory of stored files

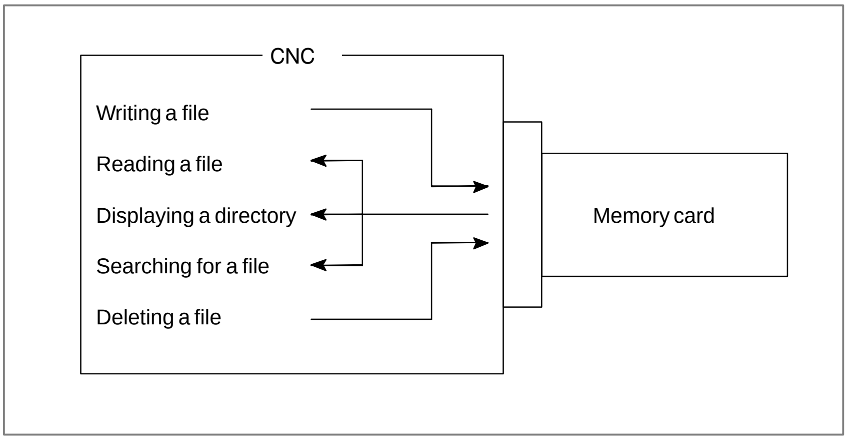
  The files stored on a memory card can be displayed on the directory screen.
- Searching for a file

  A search is made for a file on a memory card and, if found, it is displayed on the directory screen.
- Reading a file

  Text–format files can be read from a memory card.
- Writing a file

  Data such as part programs can be stored to a memory card in text file format.
- Deleting a file

  A file can be selected and deleted from a memory card.



Displaying a directory of stored files

Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[CARD]**. The screen shown below is displayed. Using page keys ⇧ and ⇩ , the screen can be scrolled.

```
DIRECTORY (M–CARD)                           O0034 N00045
   No.     FILE NAME              SIZE           DATE
   0001    O1000                123456       96/07/10
   0002    O1001                  8458       96/07/30
   0003    O0002                  3250       96/07/30
   0004    O2000                 73456       96/07/31
   0005    O2001                  3444       96/07/31
   0006    O3001                  8483       96/08/02
   0007    O3300                   406       96/08/05
   0008    O3400                  2420       96/07/31
   0009    O3500                  7460       96/07/31

 ( PROG )(        )( DIR + )(        )( (OPRT) )
```

5 Comments relating to each file can be displayed by pressing soft key **[DIR+]**.

```
DIRECTORY (M–CARD)                           O0034 N00045
   No.     FILE NAME                        COMMENT
   0001    O1000                      (COMMENT          )
   0002    O1001                      (SUB PROGRAM  )
   0003    O0002                      (12345678          )
   0004    O2000                      (                       )
   0005    O2001                      (                       )
   0006    O3001                      (SKIP–K              )
   0007    O3300                      (HI–SPEED          )
   0008    O3400                      (                       )
   0009    O3500                      (TEST PROGRAM)

 ( PROG )(        )( DIR + )(        )( (OPRT) )
```

6 Repeatedly pressing soft key **[DIR+]** toggles the screen between the display of comments and the display of sizes and dates.
Any comment described after the O number in the file is displayed.
Up to 18 characters can be displayed on the screen.

Searching for a file

Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[CARD]**. The screen shown below is displayed.

DIRECTORY (M–CARD) O0034 N00045

No.	FILE NAME	SIZE	DATE
0001	O1000	123456	96/07/10
0002	O1001	8458	96/07/30
0003	O0002	3250	96/07/30
0004	O2000	73456	96/07/31
0005	O2001	3444	96/07/31
0006	O3001	8483	96/08/02
0007	O3300	406	96/08/05
0008	O3400	2420	96/07/31
0009	O3500	7460	96/07/31

(PROG)( )(DIR +)( )((OPRT))

5 Press soft key **[(OPRT)]**.

(F SRH)(F READ)(N READ)(PUNCH)(DELETE)

6 Set the number of the desired file number with soft key **[F SRH]**. Then, start the search by pressing soft key **[EXEC]**. If found, the file is displayed at the top of the directory screen.

When a search is made for file number 19

DIRECTORY (M–CARD) O0034 N00045

No.	FILE NAME	COMMENT
0019	O1000	(MAIN PROGRAM)
0020	O1010	(SUBPROGRAM–1)
0021	O1020	(COMMENT)
0022	O1030	(COMMENT)

Reading a file

Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG.

3 Press the rightmost soft key (next–menu key).

4 Press soft key **[CARD]**. Then, the screen shown below is displayed.

```
DIRECTORY (M–CARD)                          O0034 N00045
  No.     FILE NAME          SIZE             DATE
 0001     O1000            123456          96/07/10
 0002     O1001              8458          96/07/30
 0003     O0002              3250          96/07/30
 0004     O2000             73456          96/07/31
 0005     O2001              3444          96/07/31
 0006     O3001              8483          96/08/02
 0007     O3300               406          96/08/05
 0008     O3400              2420          96/07/31
 0009     O3500              7460          96/07/31

[ PROG ][      ][ DIR + ][      ][ (OPRT) ]
```

```
[F SRH][F READ][N READ][PUNCH][DELETE]
```

5 Press soft key **[(OPRT)]**.

6 To specify a file number, press soft key **[F READ]**. The screen shown below is displayed.

```
DIRECTORY (M–CARD)                          O0001 N00010
  No.     FILE NAME                      COMMENT
 0019     O1000                    (MAIN  PROGRAM)
 0020     O1010                    (SUBPROGRAM–1)
 0021     O1030                    (COMMENT        )

 READ
          FILE NAME=20             PROGRAM No.=120
 >
 EDIT  ****  ****  ***                  15:40:21
[F NAME][ O SET ][ STOP ][ CAN ][ EXEC ]
```

7 Enter file number 20 from the MDI panel, then set the file number by pressing soft key **[F SET]**. Next, enter program number 120, then set the program number by pressing soft key **[O SET]**. Then, press soft key **[EXEC]**.

- File number 20 is registered as O0120 in the CNC.
- Set a program number to register a read file with a separate O number. If no program number is set, the O number in the file name column is registered.

8 To specify a file with its file name, press soft key **[N READ]** in step 6 above. The screen shown below is displayed.

```
DIRECTORY (M–CARD)                          O0001 N00010
   No.    FILE NAME                        COMMENT
  0012    O0050                       (MAIN  PROGRAM)
  0013    TESTPRO                     (SUB PROGRAM–1)
  0014    O0060                       (MACRO PROGRAM)
~                                                          ~
  READ           FILE NAME     =TESTPRO
               PROGRAM No.     =1230
  >
  EDIT  ****  ****  ***                    15:40:21
 [F NAME] [ O SET ] [  STOP  ] [  CAN  ] [  EXEC  ]
```

9 To register file name TESTPRO as O1230, enter file name TESTPRO from the MDI panel, then set the file name with soft key **[F NAME]**. Next, enter program number 1230, then set the program number with soft key [O SET]. Then, press soft key **[EXEC]**.

Writing a file

Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[CARD]**. The screen shown below is displayed.

```
DIRECTORY (M–CARD)                          O0034 N00045
   No.    FILE NAME          SIZE              DATE
  0001     O1000           123456           96/07/10
  0002     O1001             8458           96/07/30
  0003     O0002             3250           96/07/30
  0004     O2000            73456           96/07/31
  0005     O2001             3444           96/07/31
  0006     O3001             8483           96/08/02
  0007     O3300              406           96/08/05
  0008     O3400             2420           96/07/31
  0009     O3500             7460           96/07/31

( PROG )(      )( DIR + )(      )( (OPRT) )
```

5 Press soft key **[(OPRT)]**.

```
( F SRH )( F READ )( N READ )( PUNCH )( DELETE )
```

6 Press soft key **[PUNCH]**.

7 Enter a desired O number from the MDI panel, then set the program number with soft key **[O SET]**.
When soft key **[EXEC]** is pressed after the setting shown below has been made, for example, the file is written under program number O1230.

```
PUNCH               FILE NAME  =
                 PROGRAM No.  =1230
>
EDIT  ****  ****  ***                        15:40:21
( F NAME )( O SET )( STOP )( CAN )( EXEC )
```

8 In the same way as for O number setting, enter a desired file name from the MDI panel, then set the file name with soft key **[F SET]**.
When soft key **[EXEC]** is pressed after the setting shown below has been made, for example, the file is written under program number O1230 and file name ABCD12.

```
PUNCH               FILE NAME  =ABCD12
                 PROGRAM No.  =1230
>
EDIT  ****  ****  ***                        15:40:21
( F NAME )( O SET )( STOP )( CAN )( EXEC )
```

Explanations

- **Registering the same file name**

When a file having the same name is already registered in the memory card, the existing file will be overwritten.

- **Writing all programs**

To write all programs, set program number = –9999. If no file name is specified in this case, file name PROGRAM.ALL is used for registration.

- **File name restrictions**

The following restrictions are imposed on file name setting:

<File name setting>	× × × × × × × × .	□□□
	↑	↑
	Not longer than 8 characters	Extension not longer than 3 characters

Deleting a file

Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key PROG .

3 Press the rightmost soft key ▷ (next–menu key).

4 Press soft key **[CARD]**. The screen shown below is displayed.

```
DIRECTORY (M–CARD)                    O0034 N00045
  No.   FILE NAME          SIZE         DATE
  0001   O1000           123456       96/07/10
  0002   O1001             8458       96/07/30
  0003   O0002             3250       96/07/30
  0004   O2000            73456       96/07/31
  0005   O2001             3444       96/07/31
  0006   O3001             8483       96/08/02
  0007   O3300              406       96/08/05
  0008   O3400             2420       96/07/31
  0009   O3500             7460       96/07/31

( PROG )(      )( DIR + )(      )( (OPRT) )
```

5 Press soft key **[(OPRT)]**.

```
(F SRH)(F READ)(N READ)(PUNCH)(DELETE)
```

6 Set the number of the desired file with soft key **[DELETE]**, then press soft key **[EXEC]**. The file is deleted, and the directory screen is displayed again.

When file number 21 is deleted

```
DIRECTORY (M–CARD)                    O0034 N00045
  No.   FILE NAME                  COMMENT
  0019   O1000               (MAIN PROGRAM)
  0020   O1010               (SUBPROGRAM–1)
  0021   O1020               (COMMENT       )
  0022   O1030               (COMMENT       )
```

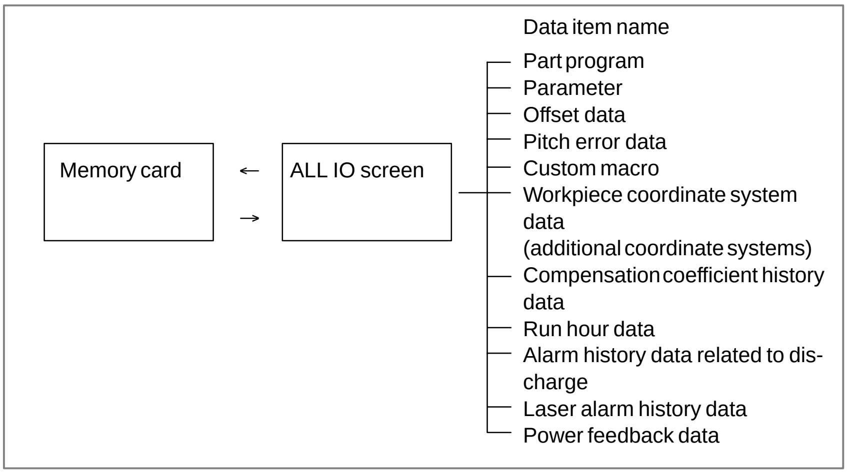
File name O1020 is deleted.

```
DIRECTORY (M–CARD)                    O0034 N00045
  No.   FILE NAME                  COMMENT
  0019   O1000               (MAIN PROGRAM)
  0020   O1010               (SUBPROGRAM–1)
  0021   O1020               (COMMENT       )
  0022   O1030               (COMMENT       )
```

File number 21 is assigned to the next file name.

Batch input/output with a memory card

On the ALL IO screen, different types of data including part programs, parameters, offset data, pitch error data, custom macros, and workpiece coordinate system data can be input and output using a memory card; the screen for each type of data need not be displayed for input/output.



Procedure

1 Press the EDIT switch on the machine operator's panel.

2 Press function key SYSTEM .

3 Press the rightmost soft key ▷ (next–menu key) several times.

4 Press soft key **[ALL IO]**. The screen shown below is displayed.

```
READ/PUNCH (PROGRAM)                      O0001 N00001
   No.    FILE NAME              SIZE         DATE
  *0001   O0222                 332010       96–04–06
  *0002   O1003                 334450       96–05–04
  *0003   MACROVAR.DAT          653400       96–05–12
  *0004   O0002                 341205       96–05–13
     [PROGRAM]
  *O0001  O0002  O0003  O0005  O0100  O0020
  *O0006  O0004  O0110  O0200  O2200  O0441
  *O0330
  >
  EDIT  ****  ****  ***                     10:07:37
 [ PROG ] [ PARAM ] [OFFSET] [      ] [(OPRT)]
```

Upper part : Directory of files on the memory card
Lower part : Directory of registered programs

5 With cursor keys ↑ and ↓ , the user can choose between upper part scrolling and lower part scrolling. (An asterisk (*) displayed at the left edge indicates the part for which scrolling is possible.)

↑ : Used for memory card file directory scrolling.

↓ : Used for program directory scrolling.

6 With page keys ↑ and ↓ , scroll through the file directory or program directory.

Explanations

- **Each data item**

When this screen is displayed, the program data item is selected. The soft keys for other screens are displayed by pressing the rightmost soft key ▷ (next–menu key). Soft key **[M–CARD]** represents a separate memory card function for saving and restoring system RAM data. (See Subsec. NO TAG.)

MACRO	PITCH	WORK		(OPRT)

			M–CARD	(OPRT)

When a data item other than program is selected, the screen displays only a file directory.
A data item is indicated, in parentheses, on the title line.

```
READ/PUNCH (PARAMETER)                    O0001 N00001
  No.    FILE NAME                     SIZE     DATE
  0001   O0222                         32010   96/04/06
  0002   O1003                          4450   96/05/04
  0003   MACROVAR.DAT                 653400   96/05/12
  0004   O0003                          4610   96/05/04
  0005   O0001                          4254   96/06/04
  0006   O0002                           750   96/06/04
  0007   CNCPARAM.DAT                  34453   96/06/04
~                                                     ~
```

- **Program directory display**

Program directory display does not match bit 0 (NAM) of parameter No. 3107, or bit 4 (SOR) of parameter No. 3107.

- **Using each function**

Display the following soft keys with soft key **[(OPRT)]**.

F SRH	F READ	N READ	PUNCH	DELETE

The operation of each function is the same as on the directory (memory card) screen. Soft key **[O SET]**, used for program number setting, and the ”PROGRAM NUMBER =” indication are not displayed for data items other than program.

[F SRH] : Finds a specified file number.
[F READ] : Reads a specified file number.
[PUNCH] : Writes a file.
[N READ] : Reads a file under a specified file name.
[DELETE] : Deletes a specified file number.

> **NOTE**
> With a memory card, RMT mode operation and the subprogram call function (based on the M198 command) cannot be used.

File format and error messages

Format

All files that are read from and written to a memory card are of text format. The format is described below.
A file starts with % or LF, followed by the actual data. A file always ends with %. In a read operation, data between the first % and the next LF is skipped. Each block ends with an LF, not a semicolon (;).

· LF: 0A (hexadecimal) of ASCII code

· When a file containing lowercase letters, kana characters, and several special characters (such as $, \, and !) is read, those letters and characters are ignored.
  Example:

```
%
O0001(MEMORY CARD SAMPLE FILE)
G17 G49 G97
G92 X–11.3 Y2.33
 .
 .
M30
%
```

· ASCII code is used for input/output, regardless of the setting parameter (ISO/EIA).

· Bit 3 of parameter No. 0100 can be used to specify whether the end of block code (EOB) is output as ”LF” only, or as ”LF, CR, CR.”

Error messages

If an error occurs during memory card input/output, a corresponding error message is displayed.

```
 0028    O0003                    7382  96–06–14

M–CARD ERROR                   × × × ×
  FILE No. =         1         PROGRAM No. =13
>_
EDIT  ****  ****  ***                15:40:21
[ F SET ][ O SET ][ STOP ][ CAN ][ EXEC ]
```

× × × × represents a memory card error code.

Memory Card Error Codes

Code	Meaning
99	Part preceding the FAT area on the memory card is destroyed.
102	The memory card does not have sufficient free space.
105	No memory card is mounted.
106	A memory card is already mounted.
110	The specified directory cannot be found.
111	There are too many files under the root directory to allow a directory to be added.
114	The specified file cannot be found.
115	The specified file is protected.
117	The file has not yet been opened.
118	The file is already open.
119	The file is locked.
121	The memory card does not have sufficient free space.
122	The specified file name is invalid.
124	The extension of the specified file is invalid.
129	A non–corresponding function was specified.
130	The specification of a device is invalid.
131	The specification of a pathname is invalid.
133	Multiple files are open at the same time.
135	The device is not formatted.
140	The file has the read/write disabled attribute.

8.12 DATA INPUT/OUTPUT BY EMBEDDED ETHERNET

8.12.1 FTP File Transfer Function

The operation of the FTP file transfer function is described below.

8.12.1.1 Host file list display

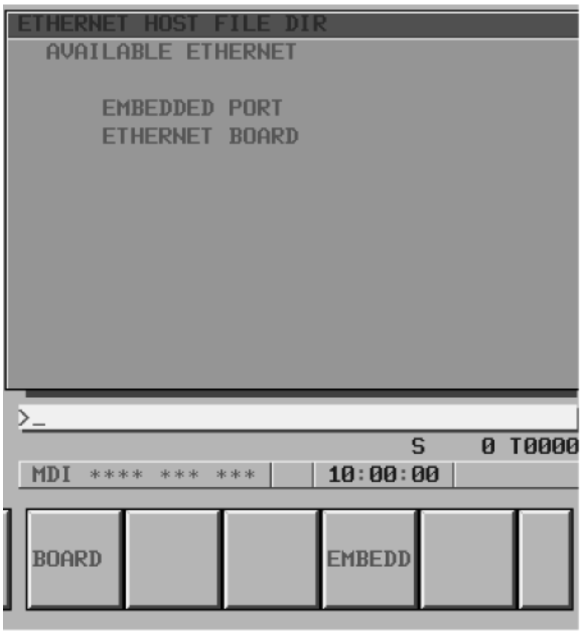
A list of the files held on the hard disk embedded to the host computer is displayed.

Procedure

1 Press the function key PROG .

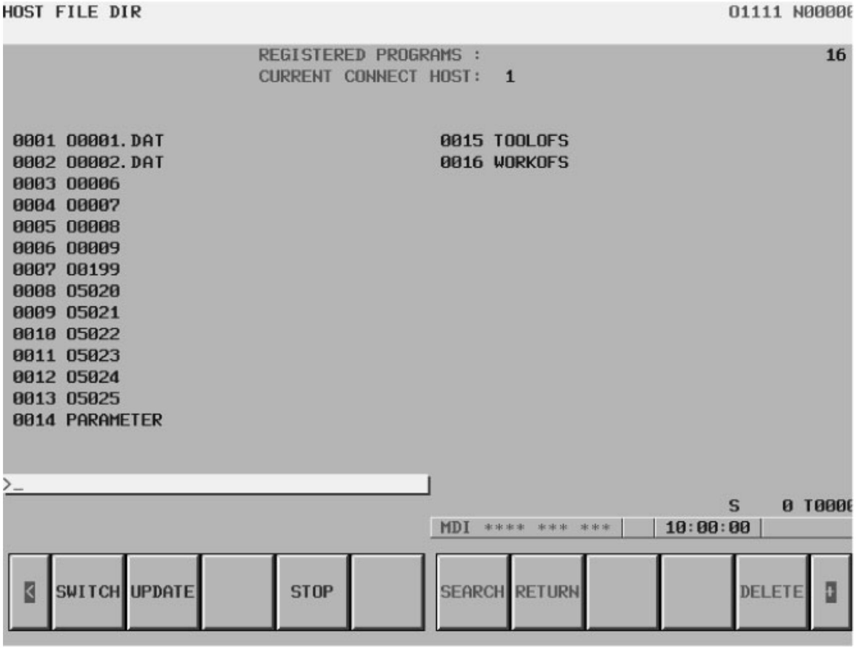
2 Press the continuous menu key at the right end of the soft key display.

3 Press the [HOST] soft key. The host file list screen appears. The Ethernet functions currently available are displayed.



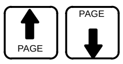
The upper row displays the usable embedded Ethernet function device.
The embedded port or PCMCIA card is displayed.
The lower row displays the usable Ethernet option boards. When no option board is installed, no information is displayed.

4 When you press the [EMBEDD] soft key, a list of the files held on the host computer specified with the embedded Ethernet port is displayed.



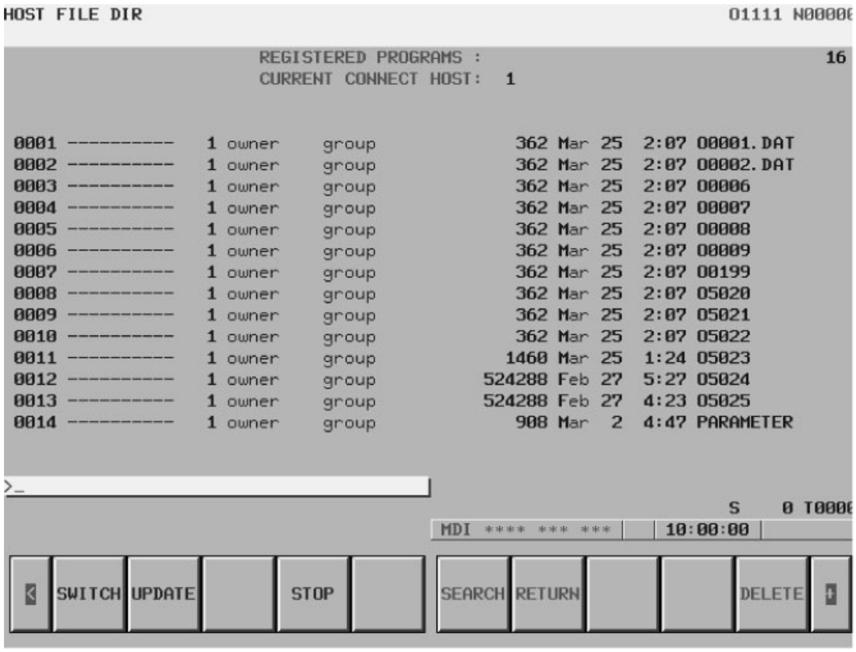
NOTE

Depending on the FTP server software, the number of displayed programs may differ between the host file list screen above and the host file list (detail) screen described below.

5 When a list of files is larger than one page, the screen display can be switched using the page keys  .

6 Press the [UPDATE] soft key to update the screen display.

7 Press the [SWITCH] soft key. The host file list (detail) screen appears.



> **NOTE**
> The host file list (detail) screen shown above is an example of screen display, and information displayed may vary according to the specification of the FTP server used with the host computer.

Display items

- **Number of registered program files** — The number of files registered in the directory (folder) of the host computer currently connected is displayed.
- **Currently connected host** — The number of the host currently connected is displayed.

List of operations

- **SWITCH** — This operation switches between normal display and detail display.
- **UPDATE** — This operation updates information displayed.
- **STOP** — This operation stops [SEARCH] operation.
- **SEARCH** — This operation updates screen information so that a file specified by its file number is placed at the start of the list.
- **DELETE** — This operation deletes a file held on the hard disk embedded to the host computer.
- **READ** — This operation reads a file held on the hard disk embedded to the host computer to the CNC part program storage. This soft key is displayed only when 9 is set as the input/output device number of the CNC, and the CNC is placed in the EDIT mode.

● PUNCH

This operation outputs a file held in the CNC part program storage to the hard disk embedded to the host computer. This soft key is displayed only when 9 is set as the input/output device number of the CNC, and the CNC is placed in the EDIT mode.

8.12.1.2 Host file search

When a list of the files held on the hard disk embedded to the host computer is displayed, a file can be placed at the start of the list by specifying its file number.

Procedure

1 Display the host file list screen.

2 Press the [SEARCH] soft key.

3 Type the file number of a file to be displayed at the start of the list with the MDI keys.
[Input format]
<file–number>

4 Press the [EXEC] soft key.

5 During search, “SEARCH” blinks in the lower–right corner of the screen.

8.12.1.3 Host file deletion

A file held on the hard disk embedded to the host computer can be deleted.

Procedure

1 Display the host file list screen.

2 Press the [DELETE] soft key.

3 Type the file number or file name of a file to be deleted, with the MDI keys.
[Input format]
<file–number>
or
<file–name>

4 Press the [EXEC] soft key.

5 During deletion, “DELETE” blinks in the lower–right corner of the screen.

NOTE
1 When a file number is used for deletion, only a file displayed on the host file list screen can be deleted.
2 The information displayed at the right end of the host file list (detail) screen is recognized as a file name. So, when deleting a host file from the host file list (detail) screen by specifying its file number, check that a file name is displayed at the right end of the screen, before specifying the file number.

8.12.1.4
NC program input

A file (NC program) on the host computer can be read to the CNC memory.

For the host file list screen

Procedure

1 Place the CNC in the EDIT mode.

2 Display the host file list screen.

3 Press the [READ] soft key.

4 Type the file number or file name of an NC program to be input, with the MDI keys.
[Input format]
<file–number>
or
<file–name>

5 Press the [EXEC] soft key.

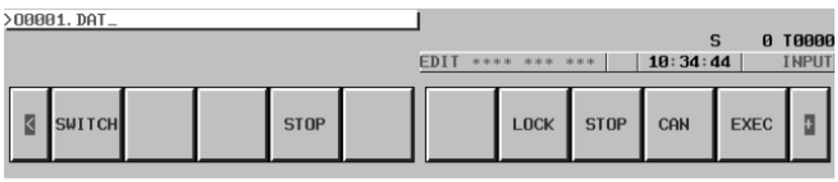
6 During input, "INPUT" blinks in the lower–right corner of the screen.

CAUTION

1 If the CNC memory holds an NC program that has the same O number as that of an NC program to be input, the NC program in the CNC memory is overwritten when bit 2 of parameter No. 3201 is set to 1.

2 If an NC program is input when bit 0 of parameter No. 3201 is set to 1, all NC programs in the CNC memory are automatically deleted before NC program input.

[Example of use]
When a file with the file name O0001.DAT held on the hard disk embedded to the host computer is to be input to the CNC memory, enter O001.DAT. Note, however, that the O number input to the CNC memory depends on the O number described in the file named O0001.DAT.



NOTE
When a file is input from this screen to the CNC memory, the O number described in the file is input.

For the program screen

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key PROG .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [PRGRM] soft key. The program screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [READ] soft key.

8 Type the O number of an NC program to be input, with the MDI keys.
[Input format]
<O–number>

9 Press the [EXEC] soft key.

10 During input, "INPUT" blinks in the lower–right corner of the screen.

CAUTION

1 If the CNC memory holds an NC program that has the same O number as that of an NC program to be input, the NC program in the CNC memory is overwritten when bit 2 of parameter No. 3201 is set to 1.

2 If an NC program is input when bit 0 of parameter No. 3201 is set to 1, all NC programs in the CNC memory are automatically deleted before NC program input.

NOTE

The valid O number of a file to be input to the CNC memory is Oxxxx (with xxxx representing a number) only.

8.12.1.5 NC program output

A file (NC program) in the CNC memory can be output to the host computer.

For the host file list screen

Procedure

1 Place the CNC in the EDIT mode.

2 Display the host file list screen.

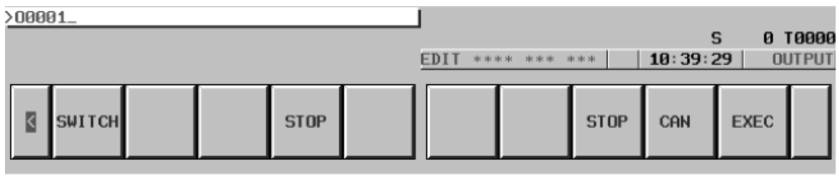
3 Press the [PUNCH] soft key.

4 Type the O number of an NC program to be output, with the MDI keys.
[Input format]
<O–number>

5 Press the [EXEC] soft key.

6 During output, "OUTPUT" blinks in the lower–right corner of the screen.

[Example of use]
When an NC program (O0001) in the CNC memory is to be output to the hard disk embedded to the host computer, enter O0001.



> **NOTE**
> An outputted file name is Oxxxx.

For the program screen

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key PROG .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [PRGRM] soft key. The program screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [PUNCH] soft key.

8 Type the O number of an NC program to be output, with the MDI keys.
[Input format]
<O–number>

9 Press the [EXEC] soft key.

10 During output, "OUTPUT" blinks in the lower–right corner of the screen.

> **NOTE**
> An outputted file name is Oxxxx.

8.12.1.6 Input/output of various types of data

With the FTP file transfer function, the types of data listed below can be input/output. This subsection describes the input/output method.

A) NC parameter
B) Tool offset value
C) Custom macro variable
D) Workpiece orogin offset value
E) Pitch error compensation data
F) M code group
G) Operation history data

Parameter input

The file (NC parameter) on the host computer can be input to the CNC memory.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key [SYSTEM] .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [PARAM] soft key. The parameter screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [READ] soft key.

8 Press the [EXEC] soft key.

9 During input, "INPUT" blinks in the lower–right corner of the screen.

File name

The fixed file name PRAMETER is used.

File format, restrictions

Refer to the operator's manual of each CNC.

Parameter output

The file (NC parameter) in the CNC memory can be output to the host computer.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key SYSTEM .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [PARAM] soft key. The parameter screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [PUNCH] soft key.

8 Press the [EXEC] soft key.

9 During output, “OUTPUT” blinks in the lower–right corner of the screen.

File name

The fixed file name PRAMETER is used.

File format, restrictions

Refer to the operator’s manual of each CNC.

Tool offset value input

The file (tool offset value) on the host computer can be input to the CNC memory.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key OFFSET SETTING .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [OFFSET] soft key. The tool compensation screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [READ] soft key.

8 Press the [EXEC] soft key.

9 During input, “INPUT” blinks in the lower–right corner of the screen.

File name

The fixed file name TOOLOFS is used.

File format, restrictions

Refer to the operator’s manual of each CNC.

Tool offset value output

The file (tool offset value) in the CNC memory can be output to the host computer.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key OFFSET SETTING .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [OFFSET] soft key. The tool compensation screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [PUNCH] soft key.

8 Press the [EXEC] soft key.

9 During output, “OUTPUT” blinks in the lower–right corner of the screen.

File name

The fixed file name TOOLOFS is used.

File format, restrictions

Refer to the operator’s manual of each CNC.

Workpiece origin offset value input

The file (workpiece origin offset value) on the host computer can be input to the CNC memory.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key OFFSET SETTING .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [WORK] soft key. The workpiece coordinate system setting screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [READ] soft key.

8 Press the [EXEC] soft key.

9 During input, “INPUT” blinks in the lower–right corner of the screen.

File name

The fixed file name WORKOFS is used.

File format, restrictions

Refer to the operator’s manual of each CNC.

Workpiece origin offset value output

The file (workpiece origin offset value) in the CNC memory can be output to the host computer.

Procedure

1. Place the CNC in the EDIT mode.
2. Press the function key OFFSET SETTING .
3. Press the continuous menu key at the right end of the soft key display.
4. Press the [WORK] soft key. The workpiece coordinate system setting screen appears.
5. Press the [(OPRT)] soft key.
6. Press the continuous menu key at the right end of the soft key display.
7. Press the [PUNCH] soft key.
8. Press the [EXEC] soft key.
9. During output, "OUTPUT" blinks in the lower–right corner of the screen.

File name

The fixed file name WORKOFS is used.

File format, restrictions

Refer to the operator's manual of each CNC.

Pitch error compensation input

The file (pitch error compensation) on the host computer can be input to the CNC memory.

Procedure

1. Place the CNC in the EDIT mode.
2. Press the function key SYSTEM .
3. Press the continuous menu key at the right end of the soft key display.
4. Press the [PITCH] soft key. The pitch error setting screen appears.
5. Press the [(OPRT)] soft key.
6. Press the continuous menu key at the right end of the soft key display.
7. Press the [READ] soft key.
8. Press the [EXEC] soft key.
9. During input, "INPUT" blinks in the lower–right corner of the screen.

File name

The fixed file name PITCH is used.

File format, restrictions

Refer to the operator's manual of each CNC.

Pitch error compensation output

The file (pitch error compensation) in the CNC memory can be output to the host computer.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key SYSTEM .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [PITCH] soft key. The pitch error setting screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [PUNCH] soft key.

8 Press the [EXEC] soft key.

9 During output, “OUTPUT” blinks in the lower–right corner of the screen.

File name

The fixed file name PITCH is used.

File format, restrictions

Refer to the operator’s manual of each CNC.

M code group input

The file (M code group) on the host computer can be input to the CNC memory.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key SYSTEM .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [M–CODE] soft key. The M code group setting screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [READ] soft key.

8 Press the [EXEC] soft key.

9 During input, “INPUT” blinks in the lower–right corner of the screen.

File name

The fixed file name M–CODE is used.

File format, restrictions

Refer to the operator’s manual of each CNC.

M code group output

The file (M code group) in the CNC memory can be output to the host computer.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key SYSTEM .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [M–CODE] soft key. The M code group setting screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [PUNCH] soft key.

8 Press the [EXEC] soft key.

9 During output, "OUTPUT" blinks in the lower–right corner of the screen.

File name

The fixed file name M–CODE is used.

File format, restrictions

Refer to the operator's manual of each CNC.

Operation history data input

The file (operation history data) on the host computer can be input to the CNC memory.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key SYSTEM .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [OPEHIS] soft key. The operation history screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [READ] soft key.

8 Press the [EXEC] soft key.

9 During input, "INPUT" blinks in the lower–right corner of the screen.

File name

The fixed file name HISTORY is used.

File format, restrictions

Refer to the operator's manual of each CNC.

Operation history data output

The file (operation history data) in the CNC memory can be output to the host computer.

Procedure

1 Place the CNC in the EDIT mode.

2 Press the function key SYSTEM .

3 Press the continuous menu key at the right end of the soft key display.

4 Press the [OPEHIS] soft key. The operation history screen appears.

5 Press the [(OPRT)] soft key.

6 Press the continuous menu key at the right end of the soft key display.

7 Press the [PUNCH] soft key.

8 Press the [EXEC] soft key.

9 During output, “OUTPUT” blinks in the lower–right corner of the screen.

File name

The fixed file name HISTORY is used.

File format, restrictions

Refer to the operator’s manual of each CNC.

8.12.1.7 Checking and changing of the connection host

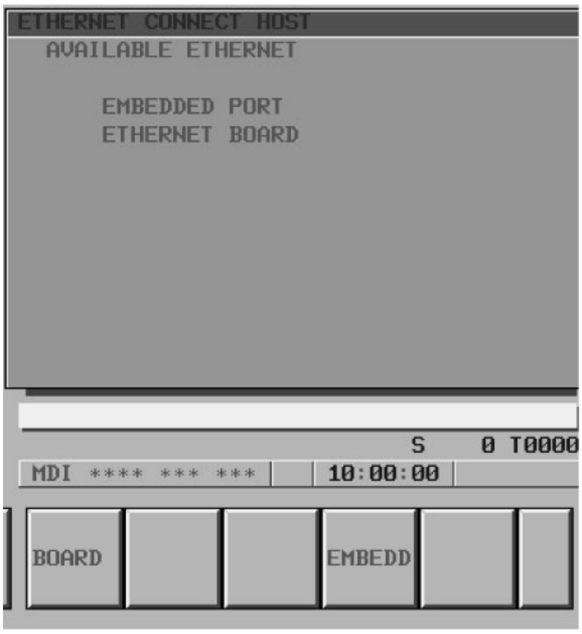
The host computer to which the FTP file transfer function attempts to make a connection as the current communication destination can be checked.

Procedure

1 Press the function key PROG .

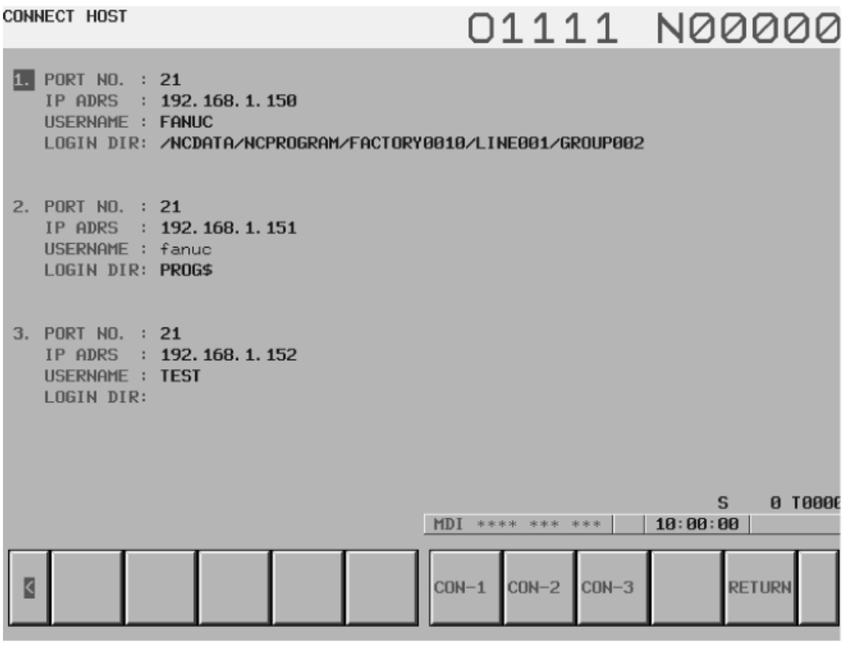
2 Press the continuous menu key at the right end of the soft key display.

3 Press the [CONECT] soft key. The connection host change screen appears. The Ethernet functions currently available are displayed.



The upper row displays the usable embedded Ethernet function device.
The embedded port or PCMCIA card is displayed.
The lower row displays the usable Ethernet option boards. When no option board is installed, no information is displayed.

4 When you press the [EMBEDD] soft key, a list of the connection host computers specified with the embedded Ethernet port is displayed.
If the usable embedded Ethernet function device is the PCMCIA card, the [PCMCIA] soft key is displayed instead of the [EMBEDD] soft key. When you press the [PCMCIA] soft key, a list of the connection host computers specified with the PCMCIA Ethernet card is displayed.



NOTE
The title of the host computer that is the current communication destination of the data server board is displayed in reverse video.

5 The connected host can be changed by pressing the [CON–1], [CON–2], or [CON–3] soft key.

Display items

- **Port number, IP address, user name, login DIR**

Those values that are set on the Ethernet parameter setting screen are displayed.

List of operations

- **CON–1**

This operation changes the connected host to host 1.

- **CON–2**

This operation changes the connected host to host 2.

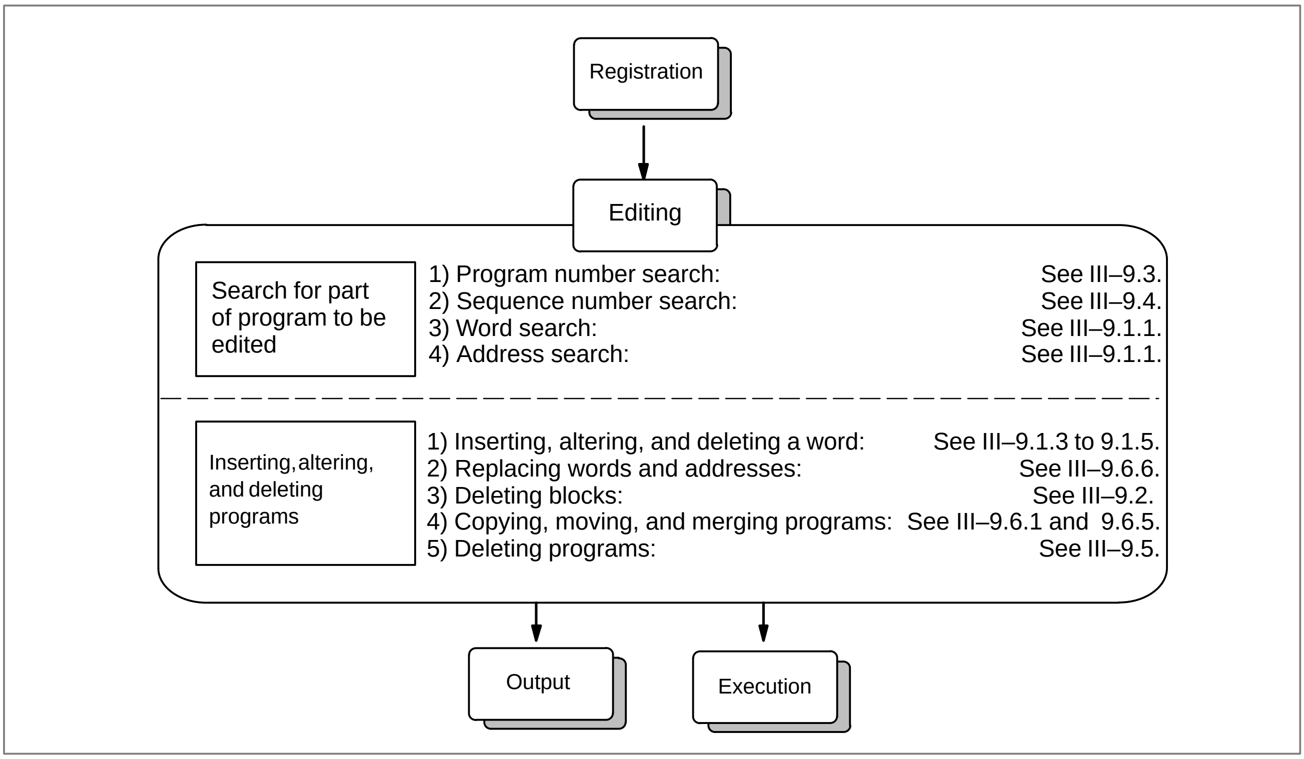
- **CON–3**

This operation changes the connected host to host 3.

9 EDITING PROGRAMS

General

This chapter describes how to edit programs registered in the CNC. Editing includes the insertion, modification, deletion, and replacement of words. Editing also includes deletion of the entire program and automatic insertion of sequence numbers. The extended part program editing function can copy, move, and merge programs. This chapter also describes program number search, sequence number search, word search, and address search, which are performed before editing the program.



9.1 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 Press PROG .

3 Select a program to be edited.
If a program to be edited is selected, perform the operation 4.
If a program to be edited is not selected, search for the program number.

4 Search for a word to be modified.
· Scan method
· Word search method

5 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.

> **WARNING**
> 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.

9.1.1 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.

9.1.2 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.

9.1.3 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  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

Z1250.0 is searched for/ scanned.

2 Key in T 1 5 .

3 Press the INSERT key.

```
Program                                O0050  N01234
 O0050  ;
 N01234  X100.0  Z1250.0  T15  ;
 S12  ;
 N56789  M03  ;
 M02  ;
 %
```

T15 is inserted.

9.1.4 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.

9.1.5 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  ;
 %
```

9.2 DELETING BLOCKS

A block or blocks can be deleted in a program.

9.2.1 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 N01234

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.

9.2.2 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 de- leted.

3 Press the DELETE key.

```
Program                              O0050  N01234
 O0050  ;
 M02  ;
 %
```

Blocks from block containing N01234 to block containing N56789 have been deleted.

> **NOTE**
> If there are too many blocks to be deleted, a P/S alarm (No. 070) may occur. If the alarm occurs, reduce the number of blocks to be deleted.

9.3 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 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 beam 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 beam side to a number from 01 to 15.
   · If the program corresponding to a signal on the machine beam side is not registered, P/S alarm (No. 059) is raised.
4. Press the cycle start button.
   · When the signal on the machine beam 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.

9.4 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 ;***
N02345 X20.0 Z20.0 ;
Target sequence number is found. → ***N02346 X10.0 Y10.0 ;***
:
O0003 ;
:

This section is searched starting at the beginning. (Search operation is 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 searched for, 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 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, 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 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, T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and NC 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 a P/S 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 N88888 in the example above, an alarm is raised.

Alarm

Number	Contents
60	Command sequence number was not found in the sequence number search.

9.5 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.

9.5.1 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.

9.5.2 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.

9.5.3 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.

9.6 EXTENDED PART PROGRAM EDITING FUNCTION

With the extended part program editing function, the operations described below can be performed using soft keys for programs that have been registered in memory.
Following editing operations are available :

· All or part of a program can be copied or moved to another program.
· One program can be merged at free position into other programs.
· A specified word or address in a program can be replaced with another word or address.

9.6.1 Copying an Entire Program

A new program can be created by copying a program.

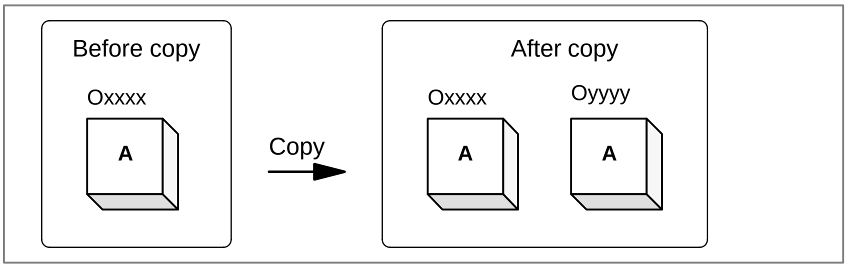
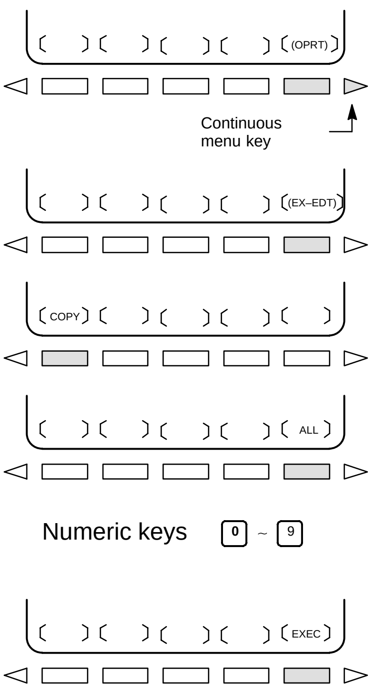


Fig.9.6.1 Copying an Entire Program

In Fig.9.6.1, the program with program number xxxx is copied to a newly created program with program number yyyy. The program created by copy operation is the same as the original program except the program number.

Procedure of copying an entire program



1 Enter the **EDIT** mode.
2 Press function key PROG .
3 Press soft key **[(OPRT)]**.
4 Press the continuous menu key.
5 Press soft key **[EX–EDT]**.
6 Check that the screen for the program to be copied is selected and press soft key **[COPY]**.
7 Press soft key **[ALL]**.
8 Enter the number of the new program (with only numeric keys) and press the INPUT key.
9 Press soft key **[EXEC]**.

9.6.2 Copying Part of a Program

A new program can be created by copying part of a program.

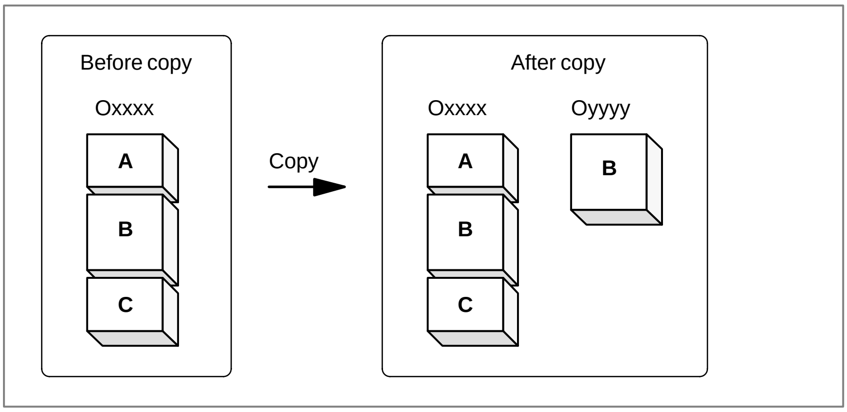
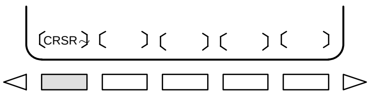


Fig.9.6.2 Copying Part of a Program

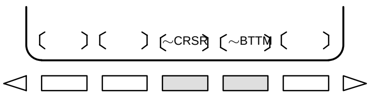
In Fig.9.6.2, part B of the program with program number xxxx is copied to a newly created program with program number yyyy. The program for which an editing range is specified remains unchanged after copy operation.

Procedure for copying part of a program

1 Perform steps **1** to **6** in **III–9.6.1**.



2 Move the cursor to the start of the range to be copied and press soft key **[CRSR∼]**.

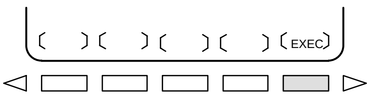


3 Move the cursor to the end of the range to be copied and press soft key **[∼CRSR]** or **[∼BTTM]** (in the latter case, the range to the end of the program is copied regardless of the position of the cursor).

Numeric keys 0 ∼ 9

4 Enter the number of the new program (with only numeric keys) and press the INPUT key.

5 Press soft key **[EXEC]**.



9.6.3 Moving Part of a Program

A new program can be created by moving part of a program.

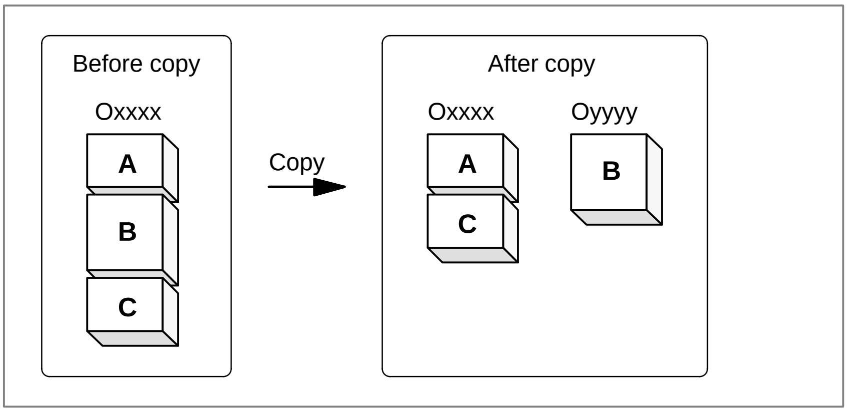
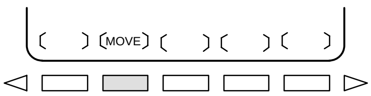


Fig.9.6.3 Moving Part of a Program

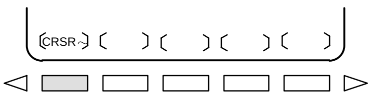
In **Fig.9.6.3**, part B of the program with program number xxxx is moved to a newly created program with program number yyyy; part B is deleted from the program with program number xxxx.

Procedure for moving part of a program

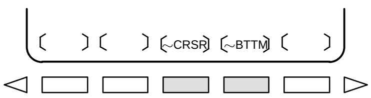
1 Perform steps 1 to 5 in **III–9.6.1**.



2 Check that the screen for the program to be moved is selected and press soft key **[MOVE]**.



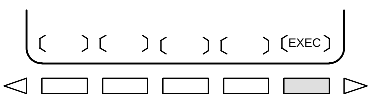
3 Move the cursor to the start of the range to be moved and press soft key **[CRSR∼]**.



4 Move the cursor to the end of the range to be moved and press soft key **[∼CRSR]** or **[∼BTTM]**(in the latter case, the range to the end of the program is copied regardless of the position of the cursor).

Numeric keys 0 ~ 9

5 Enter the number of the new program (with only numeric keys) and press the INPUT key.



6 Press soft key **[EXEC]**.

9.6.4 Merging a Program

Another program can be inserted at an arbitrary position in the current program.

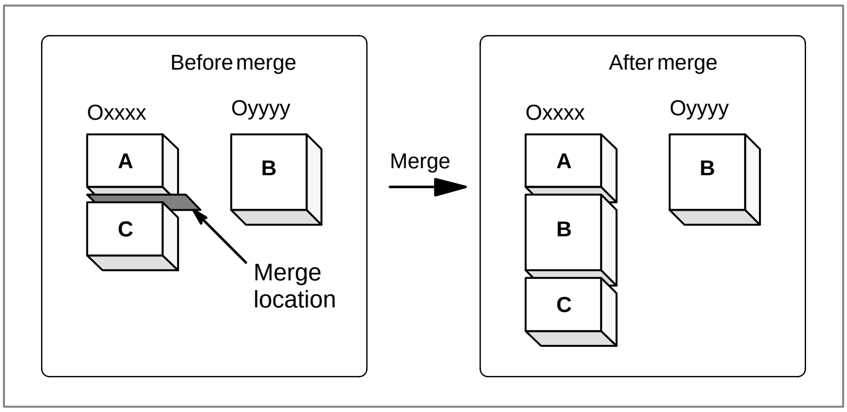
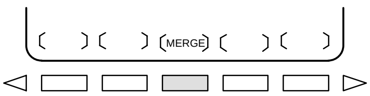
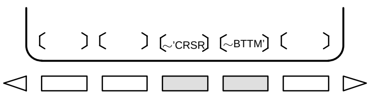


Fig.9.6.4 Merging a program at a specified location

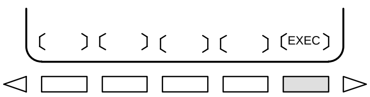
In **Fig.9.6.4,** the program with program number XXXX is merged with the program with program number YYYY. The OYYYY program remains unchanged after merge operation.

Procedure for merging a program





Numeric keys 0 ~ 9



1 Perform steps **1** to **5** in **III–9.6.1**.

2 Check that the screen for the program to be edited is selected and press soft key **[MERGE]**.

3 Move the cursor to the position at which another program is to be inserted and press soft key **[∼'CRSR]** or **[∼BTTM']**(in the latter case, the end of the current program is displayed).

4 Enter the number of the program to be inserted (with only numeric keys) and press the INPUT key.

5 Press soft key **[EXEC]**.
The program with the number specified in step 4 is inserted before the cursor positioned in step 3.

9.6.5 Supplementary Explanation for Copying, Moving and Merging

Explanations

- **Setting an editing range**

The setting of an editing range start point with **[CRSR∼]** can be changed freely until an editing range end point is set with **[∼CRSR]** or **[∼BTTM]**. If an editing range start point is set after an editing range end point, the editing range must be reset starting with a start point.
The setting of an editing range start point and end point remains valid until an operation is performed to invalidate the setting.
One of the following operations invalidates a setting:

· An edit operation other than address search, word search/scan, and search for the start of a program is performed after a start point or end point is set.

· Processing is returned to operation selection after a start point or end point is set.

- **Without specifying a program number**

In copying program and moving program, if **[EXEC]** is pressed without specifying a program number after an editing range end point is set, a program with program number O0000 is registered as a work program.
This O0000 program has the following features:

· The program can be edited in the same way as a general program. (Do not run the program.)

· If a copy or move operation is newly performed, the previous information is deleted at execution time, and newly set information (all or part of the program) is reregistered. (In merge operation, the previous information is not deleted.) However, the program, when selected for foreground operation, cannot be reregistered in the background. (A BP/S alarm No. 140 is raised.) When the program is reregistered, a free area is produced. Delete such a free area with the RESET key.

· When the program becomes unnecessary, delete the program by a normal editing operation.

- **Editing when the system waiting for a program number to be entered**

When the system is waiting for a program number to be entered, no edit operation can be performed.

Limitations

- **Number of digits for program number**

If a program number is specified by 5 or more digits, a format error is generated.

Alarm

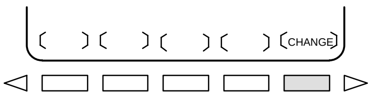
Alarm no.	Contents
70	Memory became insufficient while copying or inserting a program. Copy or insertion is terminated.
101	The power was interrupted during copying, moving, or inserting a program and memory used for editing must be cleared. When this alarm occurs, press the key RESET while pressing function key PROG. Only the program being edited is deleted.

9.6.6 Replacement of Words and Addresses

Replace one or more specified words.
Replacement can be applied to all occurrences or just one occurrence of specified words or addresses in the program.

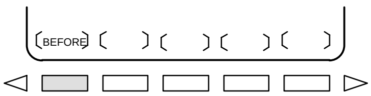
Procedure for replacement of words or addresses

1 Perform steps 1 to 5 in III–9.6.1.



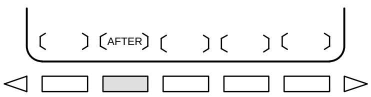
2 Press soft key **[CHANGE]**.

3 Enter the word or address to be replaced.

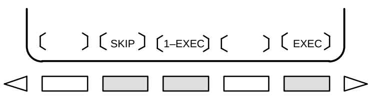


4 Press soft key **[BEFORE]**.

5 Enter the new word or address.



6 Press soft key **[AFTER]**.



7 Press soft key **[EXEC]** to replace all the specified words or addresses after the cursor.
Press soft key **[1–EXEC]** to search for and replace the first occurrence of the specified word or adress after the cursor.
Press soft key **[SKIP]** to only search for the first occurrence of the specified word or address after the cursor.

Examples

- **Replace X100 with Y200**

  [CHANGE] X 1 0 0 **[BEFORE]** Y 2 0 0 **[AFTER][EXEC]**

- **Replace X100Y200 with X30**

  [CHANGE] X 1 0 0 Y 2 0 0 **[BEFORE]** X 3 0 **[AFTER][EXEC]**

- **Replace IF with WHILE**

  [CHANGE] I F **[BEFORE]** W H I L E **[AFTER] [EXEC]**

- **Replace X with ,C10**

  [CHANGE] X **[BEFORE]** , C 1 0 **[AFTER][EXEC]**

Explanation

- **Replacing custom macros**

The following custom macro words are replaceable:
IF, WHILE, GOTO, END, DO, BPRNT, DPRINT, POPEN, PCLOS
The abbreviations of custom macro words can be specified.
When abbreviations are used, however, the screen displays the abbreviations as they are key input, even after soft key **[BEFORE]** and **[AFTER]** are pressed.

Restrictions

- **The number of characters for replacement**

Up to 15 characters can be specified for words before or after replacement. (Sixteen or more characters cannot be specified.)

- **The characters for replacement**

Words before or after replacement must start with a character representing an address.(A format error occurs.)

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 III–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:
An editing unit is a range in which the cursor appears. Programs are deleted, changed, or inserted on a per–editing–unit basis.

(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 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	**ASIN → AS**	**COS → CO**	**ACOS → AC**
TAN → TA	**ATAN → AT**	**SQRT → SQ**	**ABS → AB**
BCD → BC	**BIN → BI**	**FIX → FI**	**FUP → FU**
ROUND → RO	**END → EN**	**EXP → EX**	**THEN → TH**
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 **EDIT** or **MEMORY** mode.
Memory mode is allowed even while the program is being executed.

2 Press function key PROG .

3 Press soft key **[(OPRT)]**, then press soft key **[BG–EDT]**.
The background editing screen is displayed (PROGRAM (BG–EDIT) is displayed at the top left of the screen).

4 Edit a program on the background editing screen in the same way as for ordinary program editing.

5 After editing is completed, press soft key **[(OPRT)]**, then press soft key **[BG–EDT].** The edited program is registered in foreground program memory.

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. (See III–11.4.2)

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. (See III–11.4.2)

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.

CAUTION

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, Teach IN mode. This chapter also describes the automatic insertion of sequence numbers.

10.1 CREATING PROGRAMS USING THE MDI PANEL

Programs can be created in the EDIT mode using the program editing functions described in III–9.

Procedure for Creating Programs Using the MDI Panel

Procedure

1 Enter the **EDIT** mode.

2 Press the PROG key.

3 Press address key O and enter the program number.

4 Press the INSERT key.

5 Create a program using the program editing functions described in III–9.

Explanation

- **Comments in a program**

Comments can be written in a program using the control in/out codes.

```
Example)  O0001 (FANUC SERIES 16) ;
          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 III–9.7).
- Address O and subsequent numbers, or a space can be entered but are omitted when registered.

10.2 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 III–11.4.2).
2. Enter the **EDIT** mode.
3. Press PROG to display the program screen.
4. 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.
5. Press address key N and enter the initial value of N.
6. Press INSERT .
7. Enter each word of a block.
8. Press EOB .
9. 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
( PRGRM )( O SRH )( SRH↑ )( SRH↓ )( REWIND )
```

10.
   - 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.

10.3 CREATING PROGRAMS IN TEACH IN MODE (PLAYBACK)

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

Procedure

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 nozzle 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.

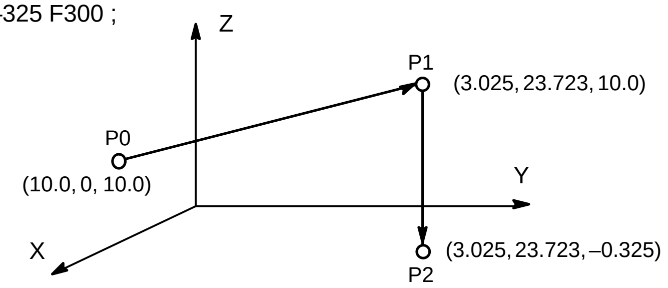
(Example) X10.521 Absolute positon (for mm input)
X10521 Data stored in memory

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 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

HELP POS PROG OFFSET SETTING CUSTOM
SYSTEM MESSAGE GRAPH

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 GRAPH is pressed. See Chapter 13 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, cutter 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

POS

Current position screen

[ABS] [REL] [ALL] [HNDL] [(OPRT)] [▷]

ABS
- Position display of work coordinate system ⇒ See III–11.1.1.
- Display of part count and run time ⇒ See III–11.1.6.
- Display of actual speed ⇒ See III–11.1.5.
- Setting of floating reference position ⇒ See III–11.1.7.

REL
- Position displays relative coordinate system ⇒ See III–11.1.2.
- Display of part count and run time ⇒ See III–11.1.6.
- Display of actual speed ⇒ See III–11.1.5.
- Setting of floating reference position ⇒ See III–11.1.7.
- Setting of relative coordinate values ⇒ See III–11.1.2.

ALL
- Total position display of each coordinate system ⇒ See III–11.1.3.
- Display of part count and run time ⇒ See III–11.1.6.
- Display of actual speed ⇒ See III–11.1.5.
- Setting of floating reference position ⇒ See III–11.1.7.
- Setting of relative coordinate values ⇒ See III–11.1.2.

HNDL
- Manual handle interruption ⇒See III–4.8.

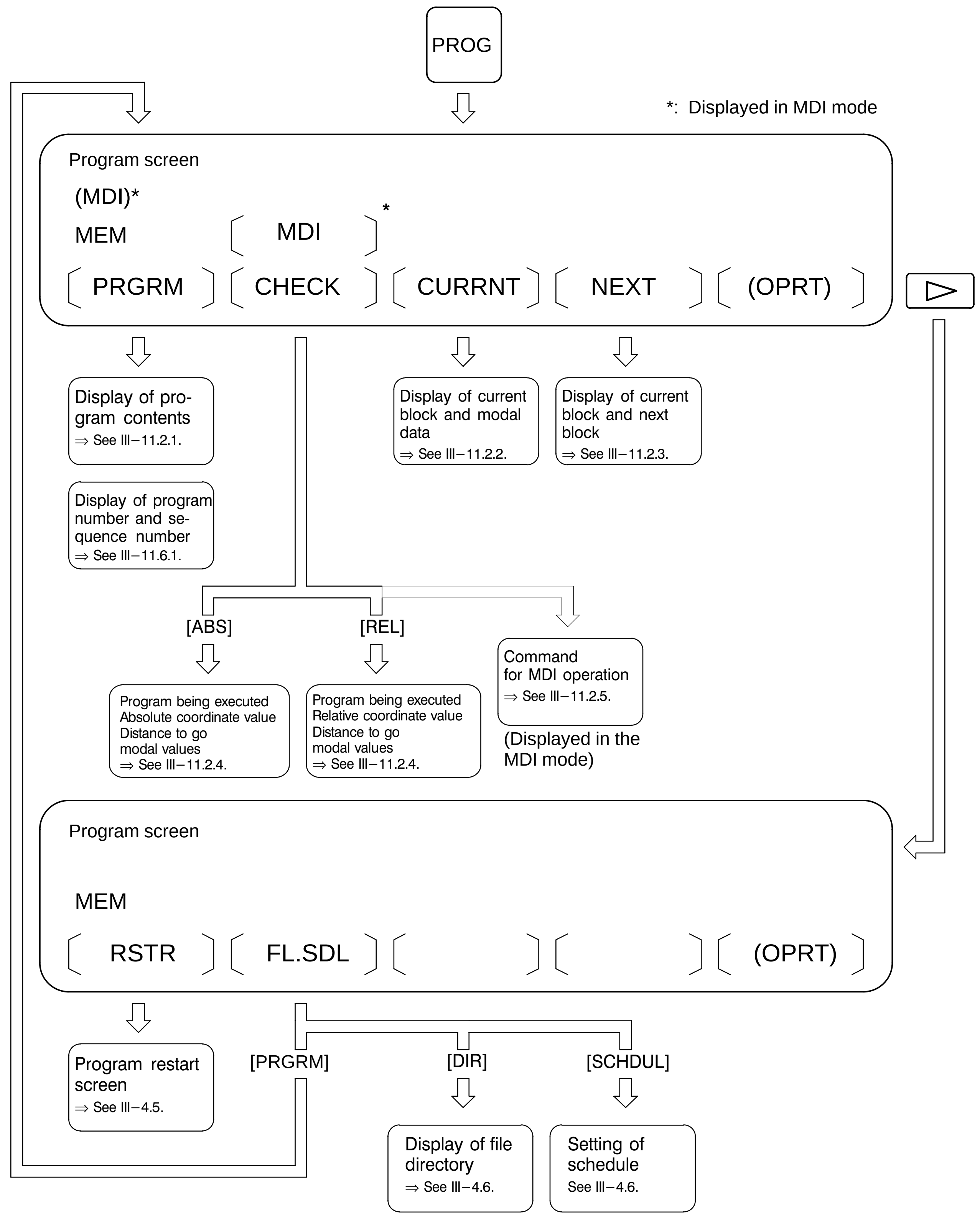
Current position screen

[MONI] [ ] [ ] [ ] [(OPRT)]

MONI
- Display of operating monitor ⇒See III–11.1.8.

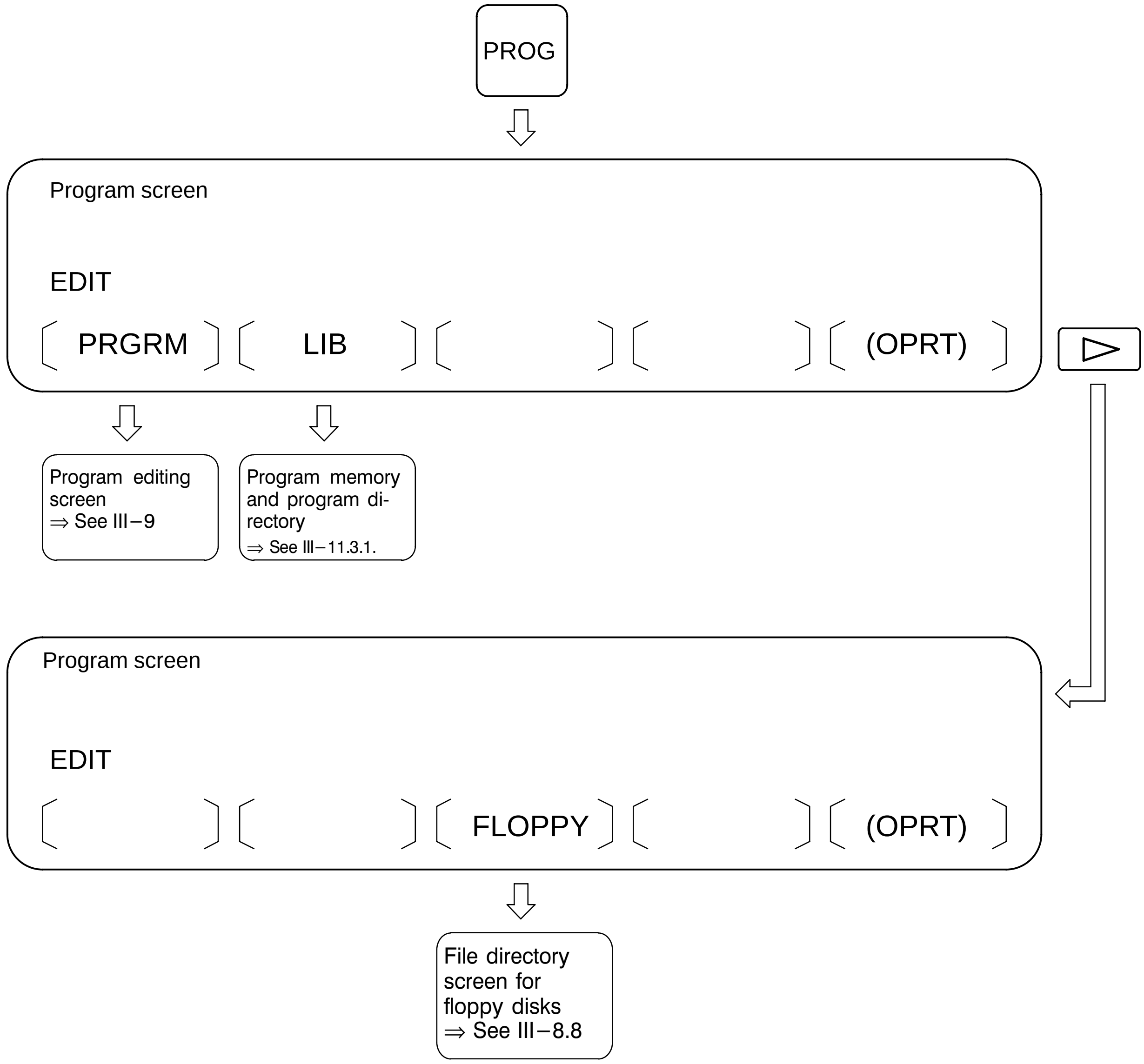
PROGRAM SCREEN

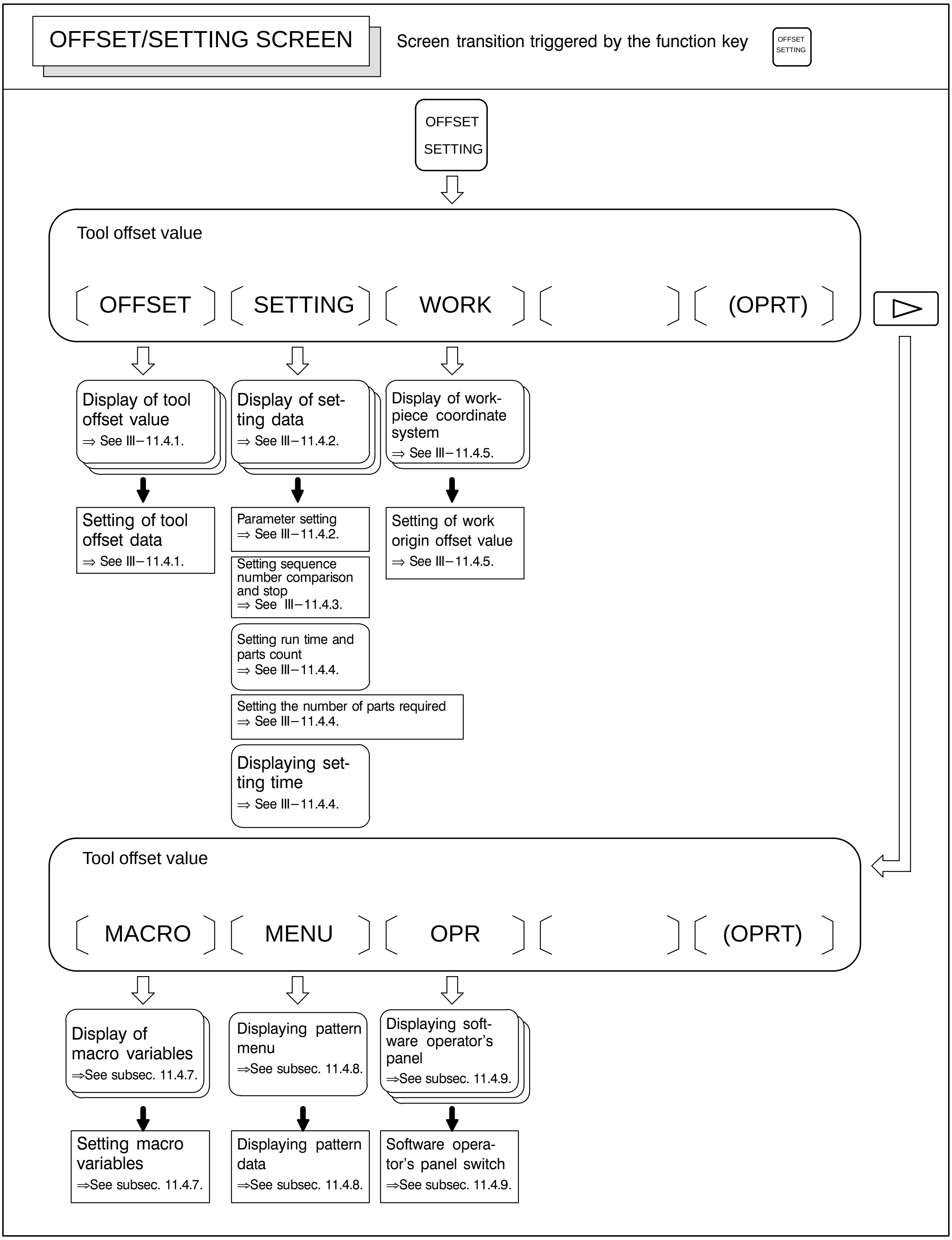
Screen transition triggered by the function key PROG in the MEMORY or MDI mode



PROGRAM SCREEN

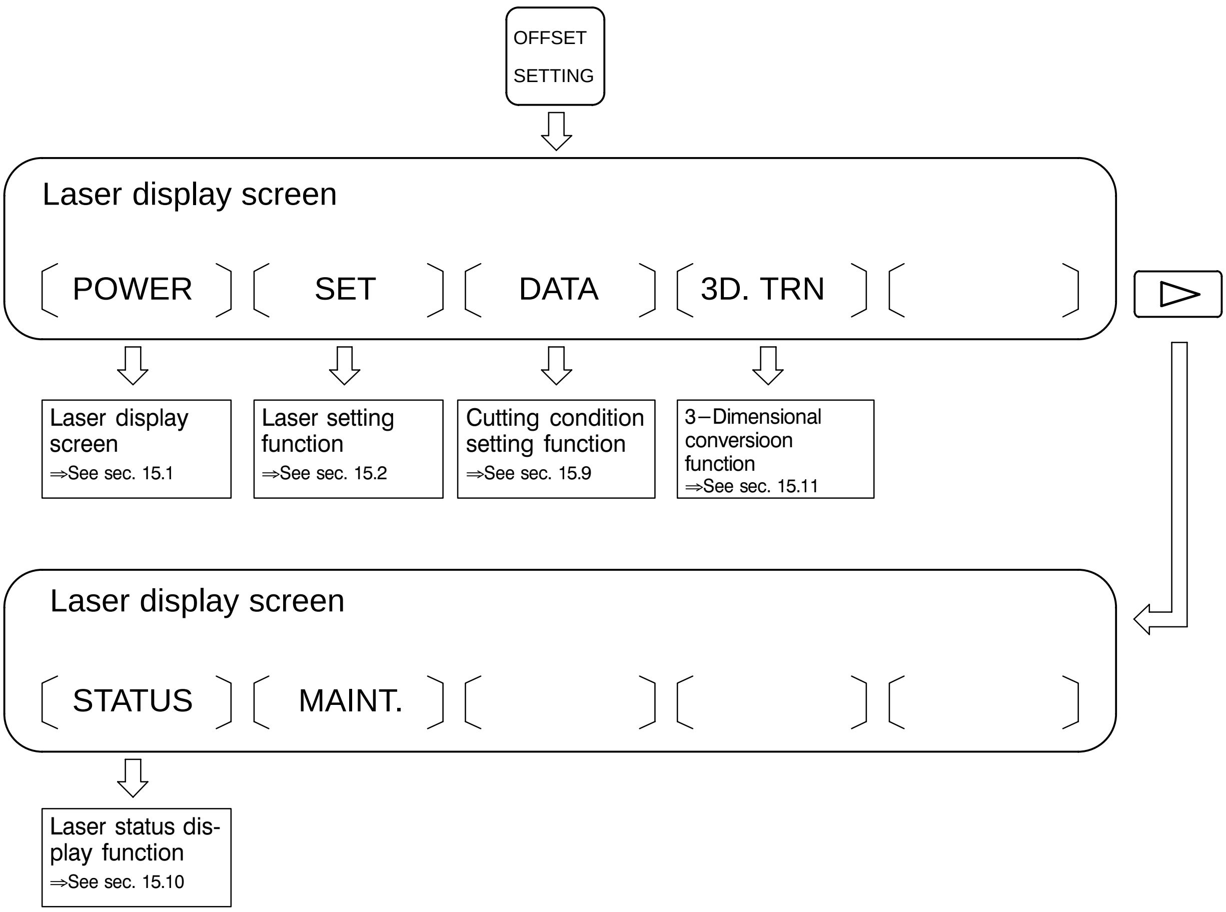
Screen transition triggered by the function key PROG in the EDIT mode




OFFSET/SETTING SCREEN
Screen transition triggered by the function key
OFFSET SETTING
OFFSET SETTING
Tool offset value
OFFSET
SETTING
WORK
(OPRT)
Display of tool offset value
⇒ See III–11.4.1.
Display of setting data
⇒ See III–11.4.2.
Display of workpiece coordinate system
⇒ See III–11.4.5.
Setting of tool offset data
⇒ See III–11.4.1.
Parameter setting
⇒ See III–11.4.2.
Setting sequence number comparison and stop
⇒ See III–11.4.3.
Setting run time and parts count
⇒ See III–11.4.4.
Setting the number of parts required
⇒ See III–11.4.4.
Displaying setting time
⇒ See III–11.4.4.
Setting of work origin offset value
⇒ See III–11.4.5.
Tool offset value
MACRO
MENU
OPR
(OPRT)
Display of macro variables
⇒See subsec. 11.4.7.
Displaying pattern menu
⇒See subsec. 11.4.8.
Displaying software operator's panel
⇒See subsec. 11.4.9.
Setting macro variables
⇒See subsec. 11.4.7.
Displaying pattern data
⇒See subsec. 11.4.8.
Software operator's panel switch
⇒See subsec. 11.4.9.

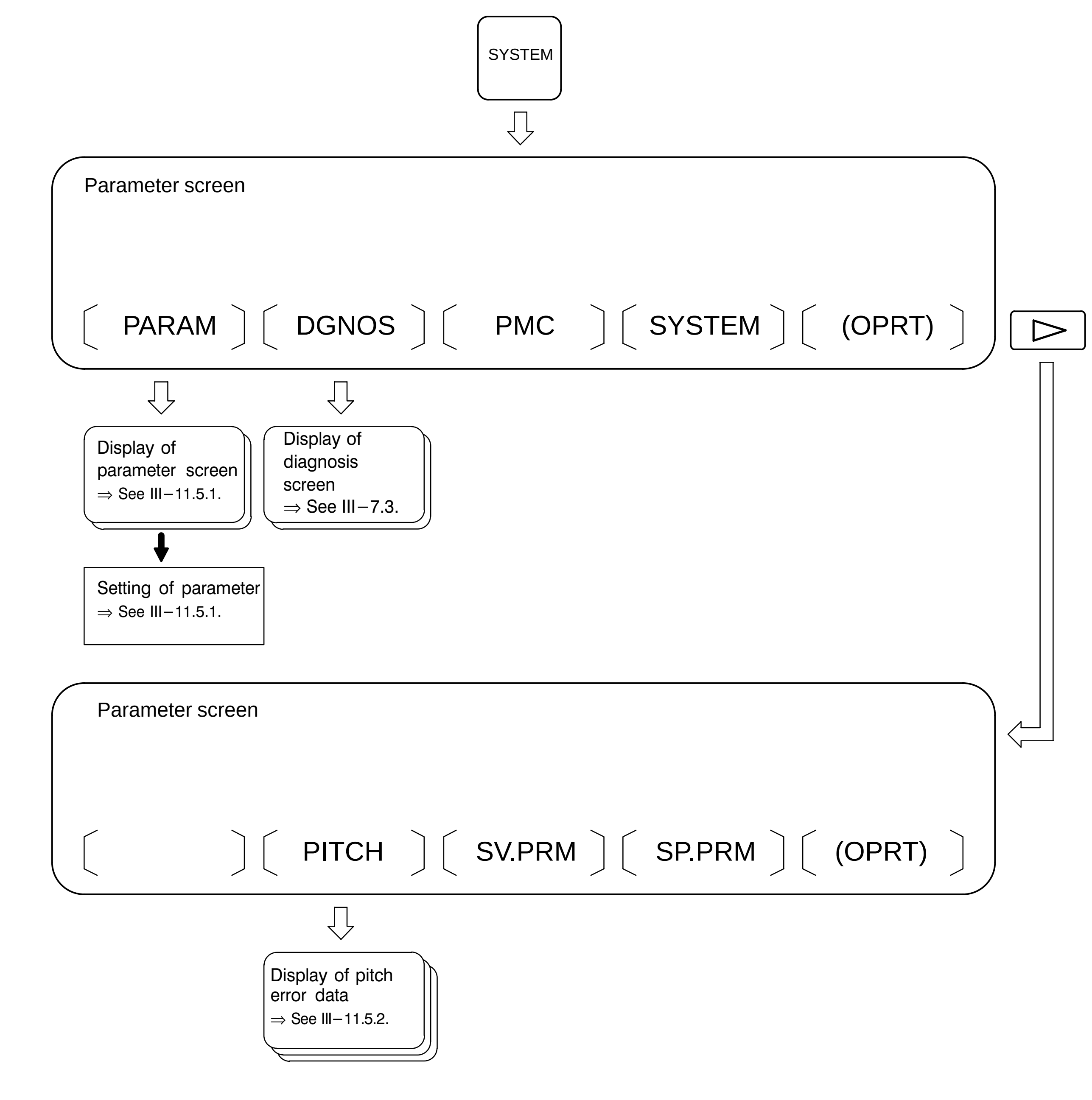
Laser Setting Screen

Screen transition triggered by the function key OFFSET SETTING



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 offset value Tool length offset value Cutter compensation value	III–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. Conversion of tape format (F15)	III–11.4.2
		Sequence number comparison and stop	III–11.4.3
3	Setting data (mirror image)	Mirror image	III–11.4.2
4	Setting data (timer)	Parts required	III–11.4.4
5	Macro variables	Custom macro common variables (#100 to #149) or (#100 to #199) (#500 to #531) or (#500 to #599)	III–11.4.7
6	Parameter	Parameter	III–11.5.1
7	Pitch error	Pitch error compensation data	III–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	III–11.4.9
9	Workpiece coordinate system setting	Workpiece origin offset value	III–11.4.5
10	Current position display screen	Floating reference position	III–11.1.7
11	Laser setting	Contouring data Hole processing data Power supply selection and setting Power control setting Assist gas selection Gas flow selection Gas flow data	III–15.2
12	Gap control setting screen	Reference displacement value Zero point adjustment Start point End point Displacement value	
13	Processing condition setting	Processing data for contouring Processing data for hole processing Processing data for edge processing Servo parameter for high–speed laser processing	III–15.9
14	Three–dimensional transform	Mirror image transform coordinate data Movement transform coordinate data	III–15.11

11.1 SCREENS DISPLAYED BY FUNCTION KEY POS

Press function key POS to display the current position of the beam.

The following three screens are used to display the current position of the beam:
·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. In addition, a floating reference position can be set on these screens.

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 III–4.8 for details on this screen.

11.1.1 Position Display in the Work Coordinate System

Displays the current position of the beam in the workpiece coordinate system. The current position changes as the beam 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]**.

3 On the 7 soft keys type, press the **[ABS]** soft key one more time to display the coordinates along axes other than the six standard axes.

```
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 No. 3104 (DAL, DAC) can be used to select whether the displayed values include beam length offset and cutter compensation.

- **Displaying the sixth and subsequent axes**

On the common screen for 7 soft keys type and 12 soft keys type, only the coordinates for the first to fifth axes are displayed initially when there are six or more controlled axes. Pressing the **[ABS]** soft key displays the coordinates for the sixth and subsequent axes.
(The common screen indicates the current position and modal information on its left side and various functions such as parameters and offsets on its right side.)

11.1.2 Position Display in the Relative Coordinate System

Displays the current position of the beam in a relative coordinate system based on the coordinates set by the operator. The current position changes as the beam 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]**.

3 On the 7 soft keys type, press the **[REL]** soft key one more time to display the coordinates along axes other than the six standard axes.

```
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.

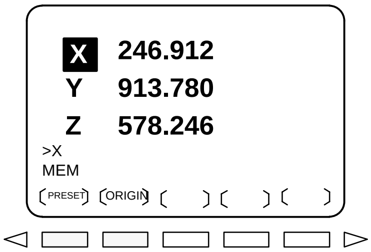
Explanations

• 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

Procedure



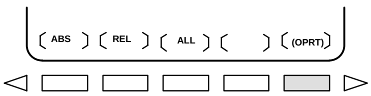
1 Enter an axis address (such as X or Y) on the screen for the relative coordinates. The indication for the specified axis blinks and the soft keys change as shown on the left.

2 · To reset the coordinate to 0, press soft key **[ORGIN]**. The relative coordinate for the blinking axis is reset to 0.

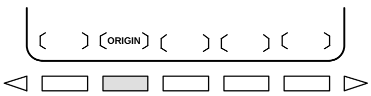
· To preset the coordinate to a specified value, enter the value and press soft key **[PRESET]**. The relative coordinate for the blinking axis is set to the entered value.

Procedure to reset all axes

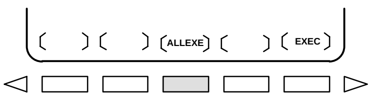
Procedure



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 4 and 5 of parameter No. 3104 (DRL, DRC) can be used to select whether the displayed values include beam length offset and cutter compensation.

• Presetting by setting a coordinate system

Bit 3 of parameter No. 3104 (PPD) 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.

• Displaying the sixth and subsequent axes

On the common screen for 7 soft keys type and 12 soft keys type, only the coordinates for the first to fifth axes are displayed initially when there are six or more controlled axes.
Only the coordinates for the first to fifth axes are displayed initially whenever when there are six or more controlled axes. Pressing the **[REL]** soft key displays the coordinates for the sixth and subsequent axes.

11.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the beam 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 III–11.1.2 for the procedure.

Procedure for displaying overall position display screen

Procedure

1 Press function key POS .

2 Press soft key **[ALL]**.

- **Display with one–path control**

```
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 beam 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 beam 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 No. 3104.

- **Displaing the sixth and subsequent axes**

Only the coordinates for the first to fifth axes are displayed initially whenever there are six or more controlled axes. Pressing the **[ALL]** soft key displays the coordinates for the sixth and subsequent axes.

- **Displaying the fifth and subsequent axes**

Relative coordinates cannot be displayed together with absolute coordinates whenever there are five or more controlled axes. Pressing the **[ALL]** soft key toggles the display between absolute and relative coordinates.

- **Resetting the relative coordinates**

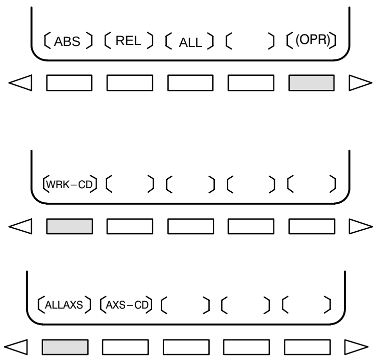
The total position display screen also supports the resetting of the relative coordintes to 0 or presetting of them to specified values. See the procedure for resetting the relative coordintes described in Subsection III–11.1.2

11.1.4 Presetting the Workpiece Coordinate System

A workpiece coordinate system shifted by an operation such as manual intervention can be preset using MDI operations to a pre–shift workpiece coordinate system. The latter coordinate system is displaced from the machine zero point by a workpiece zero point offset value.
A command (G92.1) can be programmed to preset a workpiece coordinate system. (See II–7.2.4 in the section for programming.)

Procedure for Presetting the Workpiece Coordinate System

Procedure



1 Press function key POS .

2 Press soft key **[(OPRT)]**.

3 When **[WRK–CD]** is not displayed, press the continuous menu key ▷.

4 Press soft key **[WRK–CD]**.

5 Press soft key **[ALLAXS]** to preset all axes.

6 To preset a particular axis in step 5, enter the axis name (X , Y , ...) and 0 , then press soft key **[AXS–CD]**.

Explanations

- **Operation mode**

This function can be executed when the reset state or automatic operation stop state is entered, regardless of the operation mode.

- **Presetting relative coordinates**

As with absolute coordinates, bit 3 (PPD) of parameter No. 3104 is used to specify whether to preset relative coordinates (RELATIVE).

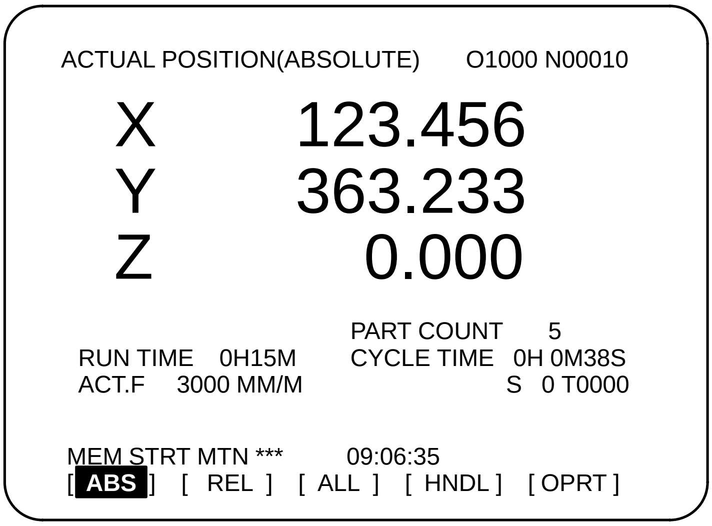
11.1.5 Actual Feedrate Display

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. On the 12 soft keys type, the actual feedrate is always displayed.

Display procedure for the actual feedrate on the current position display screen

Procedure

1 Press function key POS to display a current position display screen.



Actual feedrate is displayed after ACT.F.

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.

Explanations

- **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 No. 3105.

- **Actual feedrate display of rotary axis**

In the case of movement of rotary axis, the speed is displayed in units of deg/min but is displayed on the screen in units of input system at that time. For example, when the rotary axis moves at 50 deg/min, the following is displayed: 0.50 INCH/M

- **Actual feedrate display on the other screen**

The program check screen also displays the actual feedrate.

11.1.6 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

Procedure

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 No. 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 III–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**

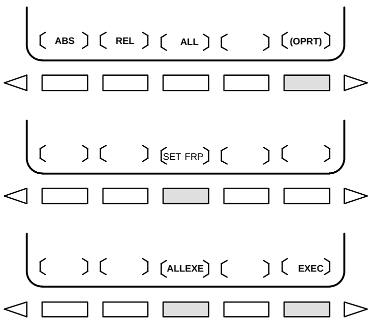
Bit 0 (PCM) of parameter No. 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

11.1.7 Setting the Floating Reference Position

To perform floating reference position return with a G30.1 command, the floating reference position must be set beforehand.

Procedure for setting the floating reference position

Procedure



1 Press function key POS to display a screen used for displaying the current position.

2 Move the beam to the floating reference position by jogging.

3 Press soft key **[(OPRT)]**.

4 Press soft key **[SET FRP]**.

5 To register the floating reference positions for all axes, press soft key **[ALLEXE]**.
To register the floating reference position of a specific axis, enter the name of the axis [X , Y , etc.), then press soft key **[EXEC]**. Two or more names can be entered consecutively (e.g., X Y Z **[EXEC]**).
The above operation stores the floating reference position. It can be checked with parameter (No. 1244).

6 In step **4**, the floating reference position along a specified axis can also be stored by entering the axis name (such as X or Y) and pressing soft key **[SET FRP]**.

Explanations

- **Presetting the relative coordinate system**

By parameter FPC (bit 3 of parameter No. 1201), the relative position can be preset to 0 when a floating reference position is registered.

11.1.8 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 No. 3111 to 1.

Procedure for displaying the operating monitor

Procedure

1 Press function key POS to display a current position display screen.

2 Press the continuous–menu key ▷.

3 Press soft key **[MONI]**.

```
OPERATING MONITOR                         O0001 N00001
 (LOAD METER)

 X :        * * *  80%   S1 :               01%
 Y : * * * * *  0%        (SPEED METER RPM)
 Z : * * * * *  0%       S1 :        * * *  1500

                          PART COUNT      5
   RUN TIME   0H15M       CYCLE TIME  0H 0M38S
   ACT.F     3000 MM/M

MEM STRT MTN ***              09:06:35
[ MONI ] [      ] [      ] [      ] [ (OPRT) ]
```

Explanations

- **Display of the servo axes**

The reading on the load meter can be displayed for up to eight servo axes by setting parameters Nos. 3151 to 3158.
When all these parameters are set to 0, data is displayed only to the 3rd axis.

- **Unit of graph**

The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%).

- **Load meter**

The reading on the load meter depends on servo parameter No. 2086.

- **Color of graph**

If the value of a load meter exceeds 100%, the bar graph turns purple.

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 III–4.5 for the program restart screen.
See III–4.6 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) ]
```

Explanations

- **12 soft keys type**

On the 12 soft keys type, the contents of the program are displayed on the right half of the screen or on the entire screen (switched each time soft key **[PRGRM]** is pressed).

```
PROGRAM                                          O0006  N00000

O0003 ;                                  N015  G99G82X550.0Y–450.0
N001  G92X0Y0Z0;                               Z–130.0R–97.0P300F70;
N002  G90 G00 Z250.0 T11 M6;             N016  G98Y–650.0;
N003  G43 Z0 H11;                        N017  G99X1050.0;
N004  S30 M3                             N018  G98Y–450.0;
N005  G99 G81X400.0 R Y–350.0            N019  G00X0Y0M5;
      Z–153.0R–97.0 F120;                N020  G49Z250.0T31M6;
N006  Y–550.0;                           N021  G43Z0H31;
N007  G98Y–750.0;                        N022  S10M3;
N008  G99X1200.0;                        N023  G85G99X800.0Y–350.0
N009  Y–550.0;                                 Z–153.0R47.0F50;
N010  G98Y–350.0;                        N024  G91Y–200.0K2;
N011  G00X0Y0M5;                         N025  G28X0Y0M5;
N012  G49Z250.0T15M6;                    N026  G49Z0;
N013  G43Z0H15;                          N027  M0;
N014  S20M3;

                                         EDIT **** *** ***     07:12:55
|  |  |  |  |  |  |                      |  | O SRH | SRH↑ | SRH↓ | REWIND | + |
```

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

Procedure

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
                  G40 .1   G21  H  2  D
                  G25      G41
                           G49  T
                           G80
                           G98  S
                           G50
>_                                   S   0  T0000
MEM STRT    ***             16:05:59
[ PRGRM ] [ CHECK ] [ CURRNT ] [ NEXT ] [ (OPRT) ]
```

Explanations

- **12 soft keys type**

The current block display screen is not provided for 12 soft keys type. Press soft key **[PRGRM]** to display the contents of the program on the right half of the screen. The block currently being executed is indicated by the cursor. Modal data is displayed on the left half of the screen. The screen displays up to 18 modal G codes.

```
ACTUAL POSITION                   O3001  N00000
   (ABSOLUTE)                   F               0  MM/MIN
X        0.000
Y        0.000                 PROGRAM
Z       30.000                 O3001 ;
                               G40 ;
                               G49 M06 T9 ;
                               G0 G54 G90 X0 Y0 ;
        (MODAL)                G43 Z30. H5 S6000 M3 ;
G00 G40 G54  F     500  M    3 M0 ;
G17 G43 G64                    X17.5 Y–22 ;
G90 G80 G69  H  5              Z–6.5 ;
G22 G90 G15  D          T    9 G10 P11 R0.995 F500 ;
G94 G50 G25                    M30 ;
G21 G67   S  6000              %
            SACT     0
                               >_
                               MEM **** *** ***     07:07:40

| ABS | REL | ALL |   |   |    | PRGRM |   |   | 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

Procedure

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 beam, and modal data in the MEMORY mode.

Procedure for displaying the program check screen

Procedure

1 Press function key PROG .

2 Press chapter selection soft key **[CHECK]**.
The program currently being executed, current position of the beam, and modal data are displayed.

- **Display with one–path control (7 soft keys type)**

```
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]**.
When there are six or more controlled axes, pressing the **[ABS]** soft key toggles the display between the absolute coordinates for the first to fifth axes and those for the sixth to eighth axes. Pressing the **[REL]** soft key toggles the relative coordinate display in the same way.

- **Modal G codes**

Up to 12 modal G codes are displayed.

- **Display during automatic operation**

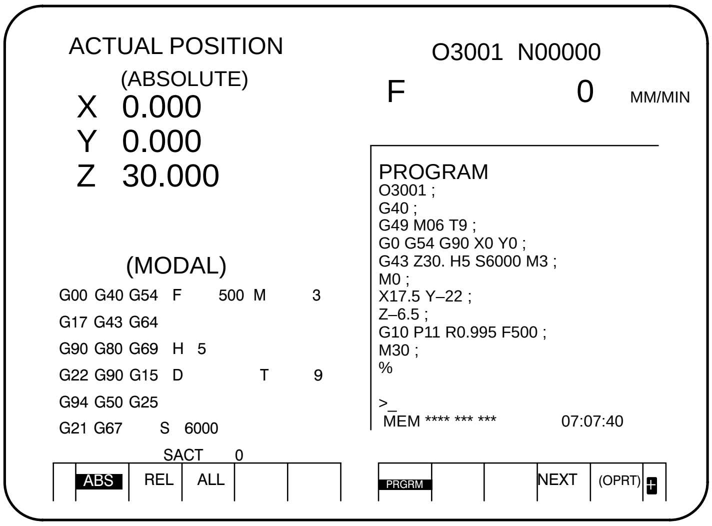
During automatic operation, the actual speed, SACT, and repeat count are displayed. The key input prompt (>_) is displayed otherwise.

- **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.

- **12 soft keys type**

The program check screen is not provided for 12 soft keys type. Press soft key **[PRGRM]** to display the contents of the program on the right half of the screen. The block currently being executed is indicated by the cursor. The current position of the beam and modal data are displayed on the left half of the screen.
Up to 18 modal G codes are displayed.



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 chapter selection soft key **[MDI]**.
The program input from the MDI and modal data are displayed.

Program

Modalinformation

```
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 III–4.2 for MDI operation.

- **Modal information**

The modal data is displayed when bit 7 (MDL) of parameter No. 3107 is set to 1. Up to 16 modal G codes are displayed. On the 12 soft keys type, however, the contents of the program are displayed on the right half of the screen and the modal data is displayed on the left half of the screen, regardless of this parameter.

- **Displaying during automatic operation**

During automatic operation, the actual speed, SACT, and repeat count are displayed. The key input prompt (>_) is displayed otherwise.

11.2.6 Stamping the Machining Time

When a machining program is executed, the machining time of the main program is displayed on the program machining time display screen. The machining times of up to ten main programs are displayed in hours/minutes/seconds. When more than ten programs are executed, data for the oldest programs is discarded.

Procedure for Stamping Machining Time

Procedure 1 Machining time calculation and display

1 Select the memory operation mode, then press the RESET key.

2 Select the program screen, then select a program whose machining time is to be calculated.

3 Execute the program to perform actual machining.

4 When the RESET key is pressed, or M02 or M30 is executed, the machining time count operation stops. When the machining time display screen is selected, the program number of the stopped main program and its machining time are displayed.
To display the machining time display screen, use the procedure below. (Machining time data can be displayed in any mode and during background editing.)

- Press the function key PROG .
- Press the rightmost soft key once or twice to display soft key **[TIME]**.
- Press soft key **[TIME]**. The machining time display screen appears.

Machining time display screen

```
PROGRAM (TIME)                                  O0010 N00002

   NO.                 TIME
   O0020               12H48M02S





>_
EDIT **** *** ***                                   16:52:13
[ TIME ]  [      ]  [      ]  [      ]  [ (OPRT) ]
```

5 To calculate the machining times of additional programs, repeat the above procedure. The machining time display screen displays the executed main program numbers and their machining times sequentially.
Note, that machining time data cannot be displayed for more than ten main programs. When more than ten programs are executed, data for the oldest programs is discarded. The screens below show how the screen display changes from the initial state where the machining times of ten main programs (O0020, O0040, ..., and O0200) are displayed to the state where the machining time of the main program O0220 is calculated.

```
PROGRAM (TIME)                          O0000  N00000

        NO.              TIME
       O0020           12H48M01S
       O0040            0H48M01S
       O0060            4H16M01S
       O0080            0H16M01S
       O0100            1H20M01S
       O0120            2H08M02S
       O0140            2H32M01S
       O0160            0H51M01S
       O0180           15H04M01S
       O0200            0H56M01S

>_
 EDIT **** *** ***        16:52:13
[ TIME ][      ][      ][      ][(OPRT)]
```

```
PROGRAM  (TIME)                         O0000  N00000

        NO.              TIME
       O0040            0H48M01S
       O0060            4H16M01S
       O0080            0H16M01S
       O0100            1H20M01S
       O0120            2H08M02S
       O0140            2H32M01S
       O0160            0H51M01S
       O0180           15H04M01S
       O0200            0H56M01S
       O0220            0H03M01S

>_
 EDIT **** *** ***        16:52:20
[ TIME ][      ][      ][      ][(OPRT)]
```

Procedure 2 Stamping machining time

1 To insert the calculated machining time of a program in a program as a comment, the machining time of the program must be displayed on the machining time display screen. Before stamping the machining time of the program, check that the machining time display screen shows the program number

2 Set the part program storage and edit mode or background edit state and select the program screen. Then select the program whose machining time is to be inserted.

3 Suppose that the machining time of O0100 is displayed on the machining time display screen.Press soft key **[(OPRT)]** to display the operation soft keys. Then, hold down the rightmost soft key until soft key **[TIME–INSERT]** appears. When soft key **[TIME–INSERT]** is pressed, the cursor moves to the start of the program, and the machining time of the program is inserted after the program number.

```
PROGRAM                                   O0100 N00000

 O0100 ;
 N10 G92 X100. Z10. ;
 N20 S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 X–10. F25. ;
 N50 G02 X–16.5 Z–12. R2. ;
 N60 G01 X40. ;
 N70 X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;
 N100 X80. ;

  EDIT  **** *** ***                    16:05:59
  [ INS–TM ][        ][        ][        ][        ]
```

⇩

```
PROGRAM                                   O0100 N00000

 O0100 (001H20M01S) ;
 N10 G92 X100. Z10. ;
 N20 S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 Z–10. F25. ;
 N50 G02 X16.5 Z–12. R2. ;
 N60 G01 X40. ;
 N70 X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;
 N100 X80. ;

 EDIT  **** *** ***              16:05:59
 [ INS–TM ][        ][        ][        ][        ]
```

4 If a comment already exists in the block containing the program number of a program whose machining time is to be inserted, the machining time is inserted after the existing comment.

```
PROGRAM                                   O0100 0N0000
 O0100    (SHAFT XSF001) ;
 N10   G92   X100. Z10. ;
 N20   S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 X–10. F25. ;
 N50 G02 X16.5 Z–12. R2. ;
 N60 G01 X40. ;
     X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;
 N100 X80. ;

EDIT   ****  ***  ***          16:52:13
[ INS–TM ]  [      ]  [      ]  [      ]  [      ]
```

⇩

```
PROGRAM                                  O0100 N00000

 O0100 (SHAFT XSF001)  001H20M01S)
 N10 G92 X100. Z10. ;
 N20 S1500 M03 ;
 N30 G00 X20.5 Z5. T0101 ;
 N40 G01 Z–10. F25. ;
 N50 G02 X16.5 Z–12. R2. ;
 N60 G01 X40. ;
 N70 X42. Z–13. ;
 N80 Z–50. ;
 N90 X44. Z–51. ;
 N100 X80. ;

EDIT   ****  ***  ***          16:52:13
[ INS–TM ] [      ]  [      ]  [      ]  [      ]
```

5 The machining time of a program inserted as a comment can be displayed after an existing program comment on the program directory screen.

```
PROGRAM DIRECTORY                    O0001 N00010

        PROGRAM (NUM.)      MEMORY (CHAR.)
   USED:          60                3321
   FREE:           2                 429

  O0020  (GEAR XGR001 ):(012H48M01S)
  O0002  (GEAR XGR002 ):(000H48M01S)
  O0010  (BOLT YBT001  ):(004H16M01S)
  O0020  (BOLT YBT002  ):(000H16M01S)
  O0040  (SHAFT XSF001):(001H20M01S)
  O0050  (SHAFT XSF002):(002H08M01S)
  O0100  (SHAFT XSF011 ):(002H32M02S)
  O0200  (PLATE XPL100 ):(000H51M01S)
 >_
 EDIT **** *** ***     14:46:09
 [ PRGRM ] [  DIR  ] [      ] [      ] [ (OPRT) ]
```

Explanations

- **Machining time**

Machining time is counted from the initial start after a reset in memory operation mode to the next reset. If a reset does not occur during operation, machining time is counted from the start to M02 (or M30). However, note that the time during which operation is held is not counted, but the time used to wait for completion of M, T, and/or B functions is counted.

- **Stamping the machining time**

The displayed machining time can be inserted (stamped) as a comment in a program stored in memory. Machining time is inserted as a comment after the program number.

- **Program directory**

The machining time inserted after a program number can be displayed on the program directory screen by setting bit 0 (NAM) of parameter No. 3107 to 1. This lets the user know the machining time of each program. This information is useful as reference data when planning processing.

Limitations

- **Alarm**

When program execution is terminated by an alarm during the machining time count, the machining time until the alarm is released is counted.

- **M02**

If the user specifies that M02 does not reset the CNC but returns completion signal FIN to the CNC to restart the program from the beginning successively (with bit 5 (M02) of parameter No. 3404 set to 0), the machining time count stops when M02 returns completion signal FIN.

- **Stamping the machining time**

When the machining time of a program to be stamped is not displayed on the machining time display screen, the machining time cannot be inserted into the program even if soft key **[TIME–INSERT]** is pressed.

- **Program directory**

When the machining time inserted into a program is displayed on the program directory screen and the comment after the program number consists of only machining time data, the machining time is displayed in both the program name display field and machining time display field.
If machining time data is inserted into a program as shown below, the program directory screen does not display the data or displays only part of the data.

Example 1: Program directory screen when a program name longer than 16 characters

```
PROGRAM                                          O0100 N00000

  O0240 (SHAFT XSF301 MATERIAL=FC25)
  (001H20M01S);
  N10 G92 X100. Z10. ;
  N20 S1500 M03 ;
  N30 G00 X20.5 Z5. T0101 ;
  N40 G01 Z–10. F25. ;
  N50 G02 X16.5 Z–12. R2. ;
  N60 G01 X40. ;
  N70 X42. Z–13. ;
  N80 Z–50. ;
  N90 X44. Z–51. ;


  EDIT **** *** ***                  16:52:13
  [ INS–TM ]  [        ]  [        ]  [        ]  [        ]
```

⇩

All characters after the first 16 characters of the program comment are discarded and the machining time display field is left blank.

```
PROGRAM DIRECTORY                       O0001 N00010

          PROGRAM (NUM.)       MEMORY (CHAR.)
   USED:          60                 3321
   FREE:           2                  429

  O0240   (SHAFT XSF301   )  :  (              )




>_
EDIT **** *** ***                      16:52:13
[ PRGRM ]  [  DIR  ]  [        ]  [        ]  [ (OPRT) ]
```

Example 2: Program directory screen when two or more machining times are stamped.

```
PROGRAM                                          O0260 N00000

  O0260 (SHAFT XSF302) (001H15M59S)
  (001H20M01S)
  N10 G92 X100. Z10. ;
  N20 S1500 M03 ;
  N30 G00 X20.5 Z5. T0101 ;
  N40 G01 Z–10. F25. ;
  N50 G02 X16.5 Z–12. R2. ;
  N60 G01 X40. ;
  N70 X42. Z–13. ;
  N80 Z–50. ;
  N90 X44. Z–51. ;

   EDIT  **** *** ***                        16:52:13
   [ INS–TM ]  [        ]  [         ]  [        ]  [        ]
```

⇩

Only the first machining time is displayed.

```
 PROGRAM DIRECTORY                          O0001 N00010

              PROGRAM (NUM.)      MEMORY (CHAR.)
    USED:              60                 3321
    FREE:               2                  429

   O0260   (SHAFT XSF302   )   :  (001H15M59S)





>_
EDIT **** *** ***        16:52:13
[ PRGRM ]  [  DIR  ]  [         ]  [        ]  [ (OPRT) ]
```

Example 3: **Program directory screen when inserted machining time data does not conform to the format hhhHmmMssS (3–digit number followed by H, 2–digit number followed by M, and 2–digit number followed by S, in this order)**

```
PROGRAM                                       O0280 N00000

  O0280 (SHAFT XSF303)  1H10M59S)
  N10 G92 X100. Z10. ;
  N20 S1500 M03 ;
  N30 G00 X20.5 Z5. T0101 ;
  N40 G01 Z–10. F25. ;
  N50 G02 X16.5 Z–12. R2. ;
  N60 G01 X40. ;
  N70 X42. Z–13. ;
  N80 Z–50. ;
  N90 X44. Z–51. ;
  N100 X80. ;


   EDIT  **** *** ***                    16:52:13
   [ INS–TM ] [        ] [         ] [        ] [        ]
```

The machining time display field is blank.

```
PROGRAM DIRECTORY                          O0001 N00010

             PROGRAM (NUM.)        MEMORY (CHAR.)
     USED:              60                   3321
     FREE:               2                    429

   O0260   (SHAFT XSF302   )  :  (001H15M59S)
   O0280   (SHAFT XSF303   )  :  (          )




>_
EDIT **** *** ***       16:52:13
[ PRGRM ] [ DIR   ] [        ] [        ] [ (OPRT) ]
```

- **Correcting the machining time**

If an incorrect machining time is calculated (such as when a reset occurs during program execution), reexecute the program to calculate the correct machining time. If the machining time display screen displays multiple programs with the same program number, select the machining time of the latest program number for insertion into the program.

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 list screen (displays memory used and a list of programs). Pressing function key PROG in the EDIT mode can also display the conversational graphics programming screen and the floppy file directory screen. See III–9 for the program editing screen and conversational graphics programming screen. See III–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

Procedure

1 Select the **EDIT** mode.

2 Press function key PROG .

3 Press chapter selection soft key **[DIR]**.

```
PROGRAM DIRECTORY                     O0001 N00010

          PROGRAM (NUM.)      MEMORY (CHAR.)
  USED:          60                3321
  FREE:           2                 429

 O0010 O0001 O0003 O0002 O0555 O0999
 O0062 O0004 O0005 O1111 O0969 O6666
 O0021 O1234 O0588 O0020 O0040



>_                    S   0  T0000
MDI **** *** ***                   16:05:59
[ PRGRM ]  [ DIR ]  [        ]  [        ]  [ (OPRT) ]
```

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.
Also, the program name, size, and updated date can be displayed in the program table by setting parameter NAM (No. 3107#0) to 1.

To toggle between the program name list (Fig.11.3.1 (a)) and the program size and program update date list (Fig.11.3.1 (b)), click the **[List]** soft key.

The update date is also changed when the program number is changed.

```
PROGRAM DIRECTORY                    O0001 N00010
          PROGRAM (NUM.)      MEMORY (CHAR.)
    USED:         60                3321
    FREE:          2                 429

  O0001  (MACRO–GCODE.MAIN)
  O0002  (MACRO–GCODE.SUB1)
  O0010  (TEST–PROGRAM.ARTHMETIC NO.1)
  O0020  (TEST–PROGRAM.F10–MACRO)
  O0040  (TEST–PROGRAM.OFFSET)
  O0050
  O0100  (INCH/MM CONVERT CHECK NO.1)
  O0200  (MACRO–MCODE.MAIN)

> _
EDIT **** *** ***    16:05:59
[ PRGRM ] [  DIR  ] [        ] [        ] [ (OPRT) ]
```

Fig.11.3.1 (a)

```
PROGRAM DIRECTORY                    O0001 N00010
          PROGRAM (NUM.)      MEMORY (CHAR.)
    USED:          60                 3321
    FREE:           2                  429

  O0001        360          1996–06–12   14:40
  O0002        240          1996–06–12   14:55
  O0010        420          1996–07–01   11:02
  O0020        180          1996–08–14   09:40
  O0040      1,140          1996–03–25   18:40
  O0050         60          1996–08–26   16:40
  O0100        120          1996–04–03   13:11


> _
EDIT **** *** ***    16:05:59
[ PRGRM ] [ DIR+ ] [        ] [        ] [ (OPRT) ]
```

Fig.11.3.1 (b)

When the NAM parameter (No.3107#0) is 0, only program numbers are displayed.

- **Program name**

Always enter a program name between the control out and control in codes immediately after the program number.
Up to 31 characters can be used for naming a program within the parentheses. If 31 characters are exceeded, the exceeded characters are not displayed.
Only program number is displayed for the program without any program name.

```
○ □□□□   (ΔΔΔ…Δ)     ;
```

Program number — Program name (up to 31 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**

When no program has been deleted from the list, each program is registered at the end of 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

11.3.2 Displaying a Program List for a Specified Group

In addition to the normal listing of the numbers and names of CNC programs stored in memory, programs can be listed in units of groups, according to the product to be machined, for example.

To assign CNC programs to the same group, assign names to those programs, beginning each name with the same character string.

By searching through the program names for a specified character string, the program numbers and names of all the programs having names including that string are listed.

Procedure for Displaying a Program List for a Specified Group

Procedure

1 Enter EDIT or background editing mode.

2 Press the PROG function key.

3 Press the PROG function key or **[DIR]** soft key to display the program list.

```
PROGRAM DIRECTORY                    O0001 N00010
        PROGRAM (NUM.)      MEMORY (CHAR.)
   USED:          60              3321
   FREE:           2               429

  O0020   (GEAR–1000 MAIN)
  O0040   (GEAR–1000 SUB–1)
  O0060   (SHAFT–2000 MAIN)
  O0100   (SHAFT–2000 SUB–1)
  O0200   (GEAR–1000 SUB–2)
  O1000   (FRANGE–3000 MAIN)
  O2000   (GEAR–1000 SUB–3)
  O3000   (SHAFT–2000 SUB–2)

> _
EDIT  **** *** ***        16:52:13
[ PRGRM ]  [ DIR ]  [         ]  [         ]  [ (OPRT) ]
```

```
( BG–EDT ) ( O–SRH ) (        ) (        ) ( GROUP )
(        ) (        ) ( NAME ) ( PR–GRP ) (        )
```

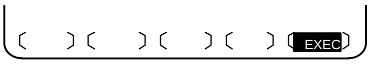
4 Press the **[(OPRT)]** operation soft key.

5 Press the **[GROUP]** operation soft key.

6 Press the **[NAME]** operation soft key.

7 Enter the character string corresponding to the group for which a search is to be made, using the MDI keys. No restrictions are imposed on the length of a program name. Note, however, that search is made based on only the first 32 characters.

Example: To search for those CNC programs having names that begin with character string "GEAR–1000," enter the following:
>GEAR–1000*_



8 Pressing the **[EXEC]** operation soft key displays the group–unit program list screen, listing all those programs whose name includes the specified character string.

```
PROGRAM DIRECTORY (GROUP)       O0001 N00010
          PROGRAM (NUM.)      MEMORY (CHAR.)
    USED:            60                 3321
    FREE:             2                  429

  O0020  (GEAR–1000 MAIN)
  O0040  (GEAR–1000 SUB–1)
  O0200  (GEAR–1000 SUB–2)
  O2000  (GEAR–1000 SUB–3)





> _
EDIT **** *** ***        16:52:25
[ PRGRM ] [  DIR  ] [        ] [        ] [ (OPRT) ]
```

[Group–unit program list screen displayed when a search is made for “GEAR–1000*”]

When the program list consists of two or more pages, the pages can be changed by using a page key.

Explanations

● * and ?

In the above example, the asterisk (*) must not be omitted. The asterisk indicates an arbitrary character string (wild card specification).

“GEAR–1000*” indicates that the first nine characters of the target program names must be “GEAR–1000,” followed by an arbitrary character string. If only “GEAR–1000” is entered, a search is made only for those CNC programs having the nine–character name “GEAR–1000.”

A question mark (?) can be used to specify a single arbitrary character. For example, entering “????–1000” enables a search to be made for programs having names which start with four arbitrary characters, followed by “–1000”.

[Example of using wild cards]

(Entered character string)		(Group for which the search will be made)
(a)	"*"	CNC programs having any name
(b)	"*ABC"	CNC programs having names which end with "ABC"
(c)	"ABC*"	CNC programs having names which start with "ABC"
(d)	"*ABC*"	CNC programs having names which include "ABC"
(e)	"?A?C"	CNC programs having four–character names, the second and fourth characters of which are A and C, respectively
(f)	"??A?C"	CNC programs having five–character names, the third and fifth characters of which are A and C, respectively
(g)	"123*456"	CNC programs having names which start with "123" and which end with "456"

- **When the specified character string cannot be found**

If no program is located as a result of a search for an entered character string, warning message "DATA NOT FOUND" is displayed on the program list screen.

- **Holding the group for which a search is made**

A group–unit program list, generated by a search, is held until the power is turned off or until another search is performed.

- **Group for which previous search was made**

After changing the screen from the group–unit program list to another screen, pressing the **[PR–GRP]** operation soft key (displayed in step 6) redisplays the group–unit program list screen, on which the program names for the previously searched group are listed. Using this soft key eliminates the need to enter the relevant character string again to redisplay the search results after changing the screen.

Examples

Assume that the main programs and subprograms for machining gear part number 1000 all have names which include character string "GEAR–1000." The numbers and names of those programs can be listed by searching through the names of all CNC programs for character string "GEAR–1000." This function facilitates the management of the CNC programs stored in large–capacity memory.

11.4 SCREENS DISPLAYED BY FUNCTION KEY OFFSET SETTING

Press function key OFFSET SETTING to display or set cutter 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. Pattern menu and pattern data
7. Software operator's panel

This section also describes measurement of beam length and the sequence number comparison and stop function.
The pattern menu, pattern data, software operator's panel depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

11.4.1 Setting and Displaying the Tool Offset Value

Tool offset values, beam 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 tool offset value

Procedure

1 Press function key OFFSET SETTING .

2 Press chapter selection soft key **[OFFSET]** or press OFFSET SETTING several times until the cutter compensation screen is displayed.
The screen varies according to the type of tool offset memory.

```
OFFSET                                    O0001 N00000
     NO.         DATA          NO.       DATA
     001         .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) ]
```

Tool offset memory A

```
OFFSET                      O0001 N00000
     NO.   GEOM(H)          WEAR(H)      GEOM(D)      WEAR(D)
     001                     0.000        0.000        0.000
     002    –1.000           0.000        0.000        0.000
     003     0.000           0.000        0.000        0.000
     004    20.000           0.000        0.000        0.000
     005     0.000           0.000        0.000        0.000
     006     0.000           0.000        0.000        0.000
     007     0.000           0.000        0.000        0.000
     008     0.000           0.000        0.000        0.000
ACTUAL POSITION (RELATIVE)
          X     0.000            Y            0.000
          Z     0.000
> _
MDI **** *** ***                16:05:59
[ OFFSET ] [ SETING ] [ WORK ] [          ] [ (OPRT) ]
```

Tool offset memory C

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 setting method**

An external input/output device can be used to input or output a tool offset value. See III–8.

- **Tool offset memory**

There are tool offset memories A, B, and C, which are classified as follows:

Tool offset memory A
D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated the same.

Tool offset memory B
D codes and H codes are treated the same. Tool geometry compensation and tool wear compensation are treated differently.

Tool offset memory C
D codes and H codes are treated differently. Tool geometry compensation and tool wear compensation are treated differently.

- **Disabling entry of compensation values**

The entry of compensation values may be disabled by setting bit 0 (WOF) and bit 1 (GOF) of parameter 3290 (not applied to tool offset memory A). And then, the input of cutter 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.

- **12 soft keys type**

OFFSET **O0000 N00000**

NO.	DATA	NO.	DATA
001	0.000	017	0.000
002	0.000	018	0.000
003	0.000	019	0.000
004	0.000	020	0.000
005	0.000	021	0.000
006	0.000	022	0.000
007	0.000	023	0.000
008	0.000	024	0.000
009	0.000	025	0.000
010	0.000	026	0.000
011	0.000	027	0.000
012	0.000	028	0.000
013	0.000	029	0.000
014	0.000	030	0.000
015	0.000	031	0.000
016	0.000	032	0.000

ACTUAL POSITION (RELATIVE)

X–12345.678
Y–12345.678
Z–12345.678
A–12345.678
B–12345.678
C–12345.678
7–12345.678
8–12345.678

>_

MDI **** *** *** 20:45:00

OFFSET SETTING (OPRT)

Offset memory A

OFFSET **O0000 N00000**

	(LENGTH)	(RADIUS)	ACTUAL POSITION			
NO.	GEOM	WEAR	GEOM	WEAR	(RELATIVE)	
001	0.000	0.000	0.000	0.000	X	0.000
002	0.000	0.000	0.000	0.000	Y	0.000
003	0.000	0.000	0.000	0.000	Z	0.000
004	0.000	0.000	0.000	0.000	A	0.000
005	0.000	0.000	0.000	0.000	B	0.000
006	0.000	0.000	0.000	0.000	C	0.000
007	0.000	0.000	0.000	0.000	7	0.000
008	0.000	0.000	0.000	0.000	8	0.000
009	0.000	0.000	0.000	0.000		
010	0.000	0.000	0.000	0.000		
011	0.000	0.000	0.000	0.000		
012	0.000	0.000	0.000	0.000		
013	0.000	0.000	0.000	0.000		
014	0.000	0.000	0.000	0.000		
015	0.000	0.000	0.000	0.000		
016	0.000	0.000	0.000	0.000		

>_

MDI **** *** *** 20:45:00

OFFSET SETTING (OPRT)

Offset memory C

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, and perform settings for the sequence number comparison and stop function.
See III–10.2 for automatic insertion of sequence numbers.
See III–11.4.3 for the sequence number comparison and stop function.
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 (0–3:CHANNEL NO.)
  SEQUENCE NO.       =  0 (0:OFF   1:ON)
  TAPE FORMAT        =  0 (0:NO CNV  1:F15)
  SEQUENCE STOP      =        0 (PROGRAM NO.)
  SEQUENCE STOP      =        0 (SEQUENCE NO.)

> _
MDI **** *** ***                     16:05:59
[ OFFSET ] [ SETING ] [ WORK ] [      ] [ (OPRT) ]
```

```
SETTING (HANDY)                                O0001 N00000

  MIRROR IMAGE         X  =  1 (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
3 : Channel 3

- **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.

- **TAPE FORMAT**

Setting the F15 tape format conversion.
0 : Tape format is not converted.
1 : Tape format is converted.
See II. PROGRAMMING for the F15 tape format.

- **SEQUENCE STOP**

Setting the sequence number with which the operation stops for the sequence number comparison and stop function and the number of the program to which the sequence number belongs

- **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 III–11.4.5 for this screen.

11.4.3 Sequence Number Comparison and Stop

If a block containing a specified sequence number appears in the program being executed, operation enters single block mode after the block is executed.

Procedure for sequence number comparison and stop

Procedure

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 (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  (0–3:CHANNEL NO.)
  SEQUENCE NO.      =  0  (0:OFF    1:ON)
  TAPE FORMAT       =  0  (0:NO CNV  1:F10/11)
  SEQUENCE STOP     =         0  (PROGRAM NO.)
  SEQUENCE STOP     =        11  (SEQUENCE NO.)


> _
MDI **** *** ***                      16:05:59
[ OFFSET ] [ SETING ] [ WORK ] [        ] [ (OPRT) ]
```

5 Enter in (PROGRAM NO.) for SEQUENCE STOP the number (1 to 9999) of the program containing the sequence number with which operation stops.

6 Enter in (SEQUENCE NO.) for SEQUENCE STOP (with five or less digits) the sequence number with which operation is stopped.

7 When automatic operation is executed, operation enters single block mode at the block containing the sequence number which has been set.

Explanations

- **Sequence number after the program is executed**

After the specified sequence number is found during the execution of the program, the sequence number set for sequence number compensation and stop is decremented by one. When the power is turned on, the setting of the sequence number is 0.

- **Exceptional blocks**

If the predetermined sequence number is found in a block in which all commands are those to be processed within the CNC control unit, the execution does not stop at that block.

Example

```
N1 #1=1 ;
N2 IF [#1 EQ 1] GOTO 08 ;
N3 GOTO 09 ;
N4 M98 P1000 ;
N5 M99 ;
```

In the example shown above, if the predetermined sequence number is found, the execution of the program does not stop.

- **When the same sequence number is found several times in the program**

If the predetermined sequence number appears twice or more in a program, the execution of the program stops after the block in which the predetermined sequence number is found for the first time is executed.

- **Block to be repeated a specified number of times**

If the predetermined sequence number is found in a block which is to be executed repeatedly, the execution of the program stops after the block is executed specified times.

11.4.4 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

Procedure

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     =    1993/07/05
        TIME=         11:32:52

> _
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 6710.

- **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

11.4.5 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59, G54.1 P1 to G54.1 P48 and G54.1 P1 to G54.1 P300) 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

Procedure

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, P1 to P48 : workpiece coordinate systems G54.1 P1 to G54.1 P48, P1 to P300 : workpiece coordinate systems G54.1 P1 to G54.1 P300) 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.

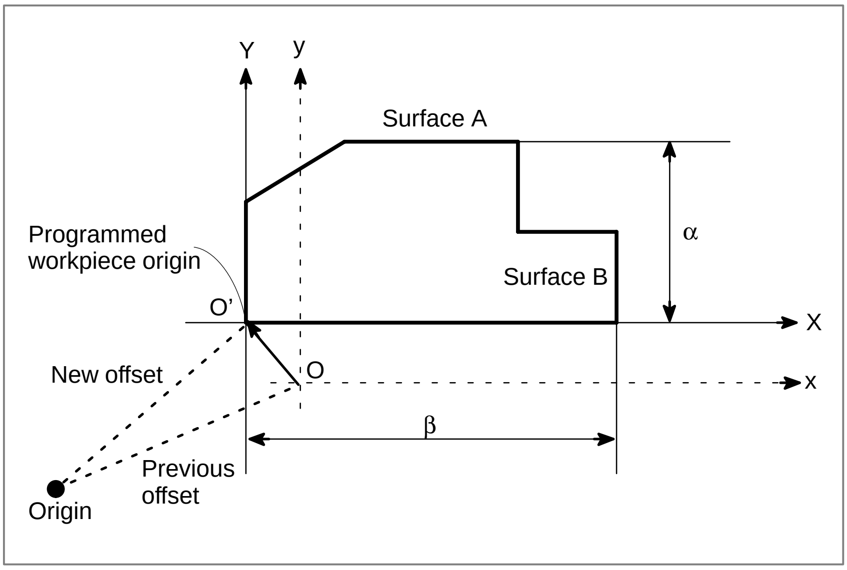
8 Turn on the data protection key to disable writing.

11.4.6 Direct 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 Direct Inputting of Measured Workpiece Origin Offsets

Procedure



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 beam 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 beam manually until it touches surface B of the workpiece.

10 Retract the beam 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**

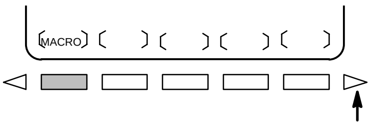
This function cannot be used while a program is being executed.

11.4.7 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the screen. 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

Procedure



Continuous menu key

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 relative coordinate in a variable, press address key [X], [Y], or [Z], then press soft key **[INP.C.]**.

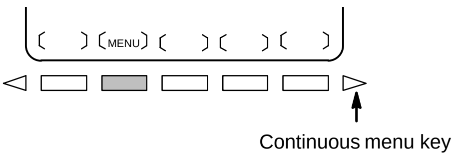
6 To set a blank in a variable, just press soft key **[INPUT]**. The value field for the variable becomes blank.

11.4.8 Displaying Pattern Data and Pattern Menu

This subsection uses an example to describe how to display or set machining menus (pattern menus) created by the machine tool builder. Refer to the manual issued by the machine tool builder for the actual pattern menus and pattern data. See II. PROGRAMMING for the pattern data entry function.

Procedure for displaying the pattern data and the pattern menu

Procedure



1 Press function key OFFSET SETTING .

2 Press the continuous menu key ▷ , then press chapter selection soft key **[MENU]**.
The following screen (pattern menu screen) is displayed:

```
MENU : HOLE PATTERN                          O0000 N00000
        1.  TAPPING
        2.  DRILLING
        3.  BORING
        4.  POCKET
        5.  BOLT HOLE
        6.  LINE ANGLE
        7.  GRID
        8.  PECK
        9.
       10.

> _
MDI **** *** ***     16:05:59
[ MACRO ] [ MENU ] [ OPR ] [      ] [ (OPRT) ]
```

3 Enter a pattern number and press soft key **[SELECT]**.
In this example, press 5 , then press **[SELECT]**.
The following screen (pattern data screen) is displayed:

```
VAR. : BOLT HOLE                              O0001 N00000
  NO.     NAME            DATA     COMMENT
  500     TOOL            0.000
  501     STANDARD X      0.000    *BOLT HOLE
  502     STANDARD Y      0.000    CIRCLE*
  503     RADIUS          0.000    SET PATTERN
  504     S. ANGL         0.000    DATA TO VAR.
  505     HOLES NO        0.000    NO.500–505.
  506                     0.000
  507                     0.000
 ACTUAL POSITION  (RELATIVE)
    X     0.000                   Y     0.000
> _ Z     0.000
MDI **** *** ***                   16:05:59
[ OFFSET ] [SETING] [       ] [      ] [ (OPRT) ]
```

4 Enter necessary pattern data and press INPUT .

5 After entering all necessary data, enter the **MEMORY** mode and press the cycle start button to start machining.

Explanations

- **Explanation of the pattern menu screen**

HOLE PATTERN : Menu title
An optional character string can be displayed within 12 characters.

BOLE HOLE : Pattern name
An optional character string can be displayed within 10 characters.

The machine tool builder should program character strings of menu title and pattern name by custom macro, and load them into the program memory.

- **Explanation of the pattern data screen**

BOLT HOLE : Pattern data title
An optional character string can be displayed within 12 characters.

TOOL : Variable name
An optional character string can be displayed within 10 characters.

BOLT HOLE CIRCLE : Comment statement
An optional character string comment can be displayed up to 12 characters/line by 8 lines.

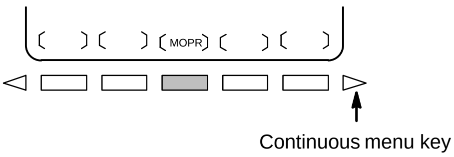
The machine tool builder should program the character strings of variable name and comment statement by custom macro, and load them into the program memory.

11.4.9 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 MDI panel.
This means that mode selection and jog feed override selection can be performed on the MDI panel and the corresponding switches on the machine operator's panel can be omitted.
Jog feed can be performed using numeric keys.

Procedure for displaying and setting the software operator's panel

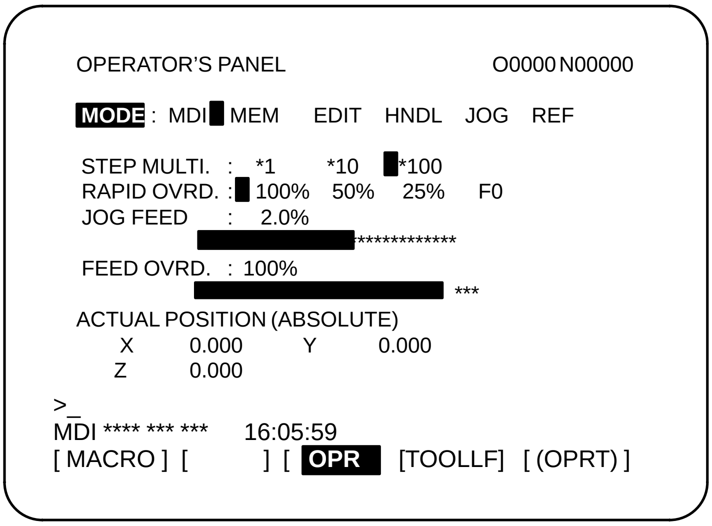
Procedure

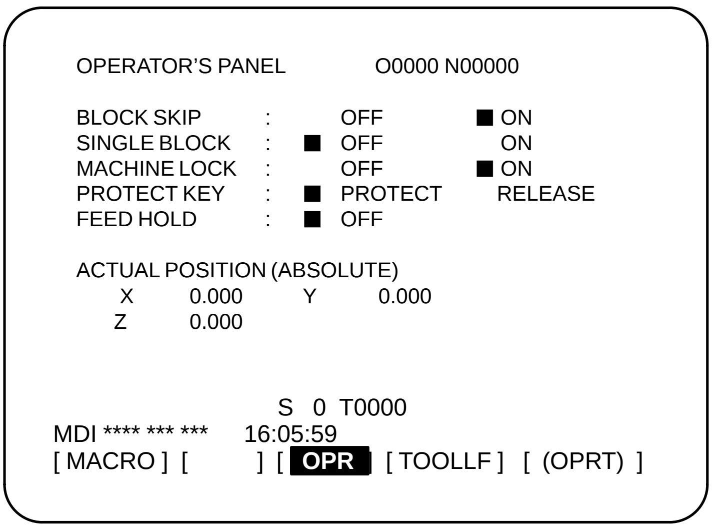


1 Press function key OFFSET SETTING .

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.

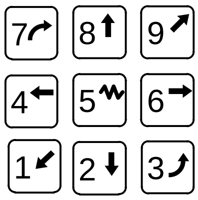




4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓ .

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.



Explanations

- **Valid operations**

The valid operations on the software operator's panel are shown below. Whether to use the CRT/MDI panel 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 III–8).
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 III–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

Procedure

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
  0001                                    FCV
        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
  0022                                          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  (0–3:CHANNEL NO.)
  SEQUENCE NO.      =  0  (0:OFF   1:ON)
  TAPE FORMAT       =  0  (0:NO CNV  1:F10/11)
  SEQUENCE STOP     =  0  (PROGRAM NO.)
  SEQUENCE STOP     =  11 (SEQUENCE NO.)


> _                                              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 P/S alarm No. 000 has occurred, however, turn off the power supply and then turn it on, otherwise the P/S alarm is not released.

Explanations

- **Setting parameters with external input/output devices**

See III–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 P/S alarm 000. In this case, turn off the power, then turn it on again.

- **Parameter list**

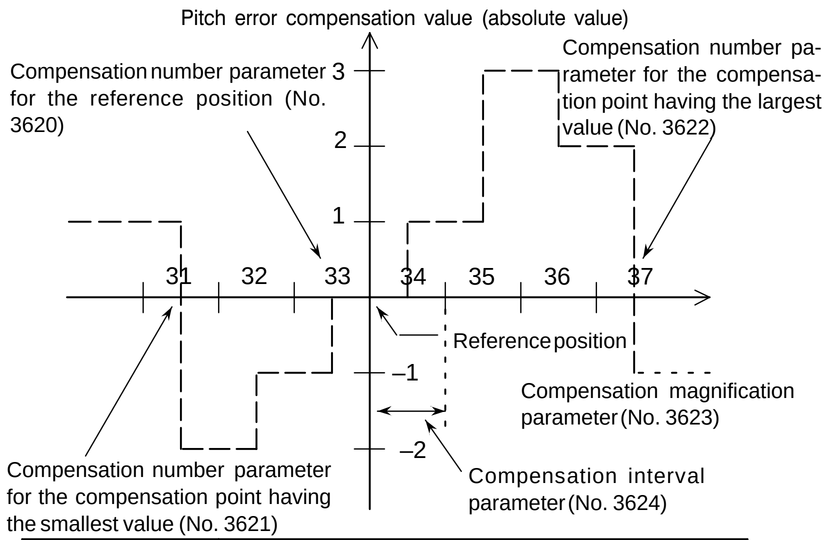
Refer to the FANUC Series 16*i*/18*i*/160*i*/180*i*–B Parameter Manual (B–63530EN) or FANUC Series 16*i*/160*i*–LB Parameter Manual (B–63670EN) for the parameter list.

- **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 III–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 No. 3620
- Number of the pitch error compensation point having the smallest value (for each axis) : Parameter No. 3621
- Number of the pitch error compensation point having the largest value (for each axis) : Parameter No. 3622
- Pitch error compensation magnification (for each axis) : Parameter No. 3623
- Interval of the pitch error compensation points (for each axis) : Parameter No. 3624

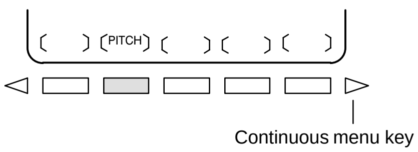
Procedure for displaying and setting the pitch error compensation data

Procedure

1 Set the following parameters:

- Number of the pitch error compensation point at the reference position (for each axis): Parameter No. 3620
- Number of the pitch error compensation point having the smallest value (for each axis): Parameter No. 3621
- Number of the pitch error compensation point having the largest value (for each axis): Parameter No. 3622
- Pitch error compensation magnification (for each axis): Parameter No. 3623
- Interval of the pitch error compensation points (for each axis): Parameter No. 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 →.

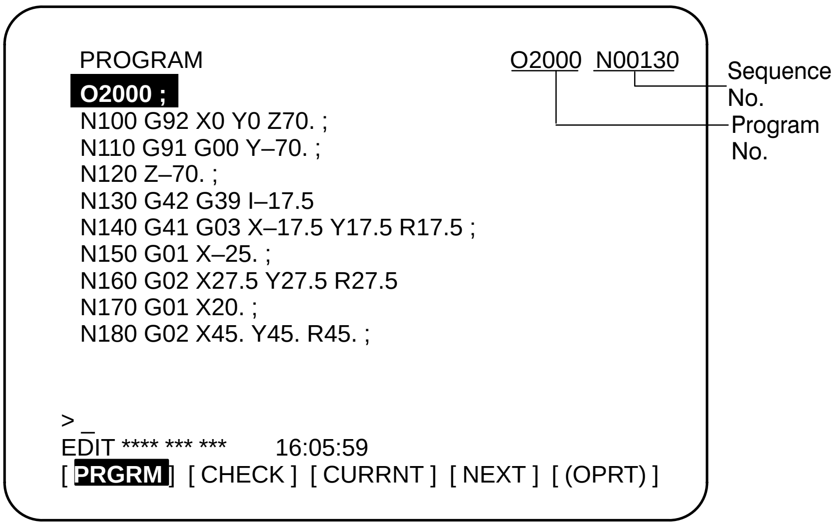
5 Enter a value with numeric keys and press the **[INPUT]** soft key.

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 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.



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 :

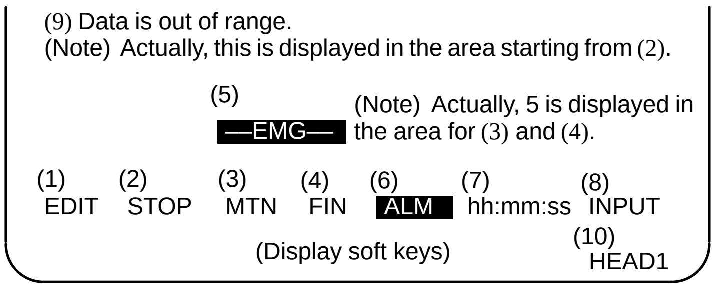
Immediately after the program No. search and sequence No. search, the program No. and the sequence No. searched are indicated.

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 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 screen. This prevents invalid data setting and input/output errors.

Explanations

Description of each display



> **NOTE**
> Actually, (10) is displayed at the position where (8) is now displayed.

(1) Current mode

MDI : Manual data input, MDI operation
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
INC : Manual incremental feed
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)
```

11.7 SCREENS DISPLAYED BY FUNCTION KEY MESSAGE

By pressing the function key MESSAGE, data such as alarms, alarm history data, and external messages can be displayed.
For information relating to alarm display, see Section III–7.1. For information relating to alarm history display, see Section III–7.2.
For information relating to external message display, see the relevant manual supplied by the machine tool builder.

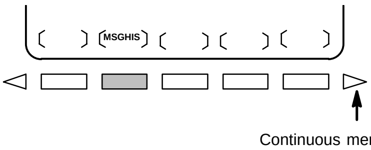
11.7.1 External Operator Message History Display

External operator messages can be preserved as history data.
Preserved history data can be displayed on the external operator message history screen.

Procedure for external operator message history display

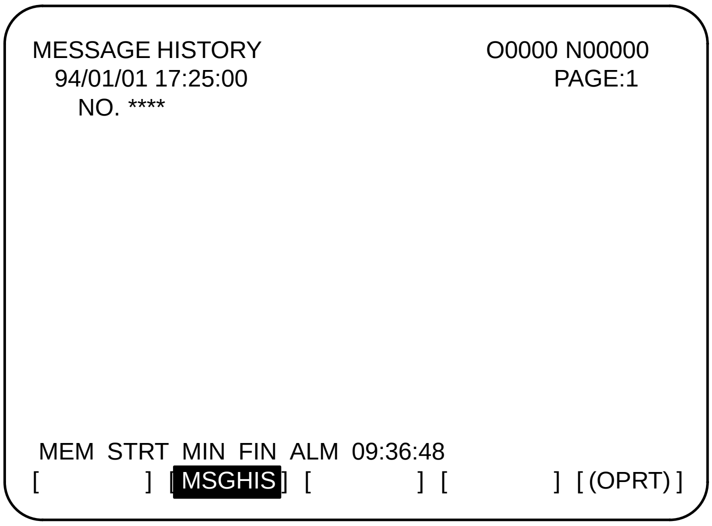
Procedure

1 Press the MESSAGE function key.



2 Press the continuous menu key ▷, then press the chapter selection soft key **[MSGHIS]**. The screen shown below appears.

Date and Page number
Message nuumber
Display range
(Up to 255 characters)



NOTE
Up to 255 characters can be specified for an external operator message. By setting MS1 and MS0 (bits 7 and 6 of parameter No. 3113), however, the number of characters that can be preserved as external operator message history data can be restricted, and the number of history data items selected.

Explanations

- **Updating external operator message history data**

When an external operator message number is specified, updating of the external operator message history data is started; this updating is continued until a new external operator message number is specified or deletion of the external operator message history data is specified.

- **Clearing external operator message history data**

To clear external operator message history data, press the **[CLEAR]** soft key. This clears all external operator message history data. (Set MSGCR (bit 0 of parameter No. 3113) to 1.)
Note that when MS1 and MS0 (bits 7 and 6 of parameter No. 3113), used to specify the number of external operator message history data items to be displayed, are changed, all existing external operator message history data is cleared.

Limitations

- **Option**

Before this function can be used, the external data input function or optional external message function must be selected.

11.8 CLEARING THE SCREEN

Displaying the same characters in the same positions on the screen causes a LCD to degrade relatively quickly. To help prevent this, the screen can be cleared by pressing specific keys. It is also possible to specify the automatic clearing of the screen if no keys are pressed during a period specified with a parameter.
But, the life of the back light may be contracted all the more when the clearing of screen and re–indication of screen are repeated beyond the necessity.
This effect can be expected when a screen is cleared for more than one hour.

11.8.1 Erase Screen Display

Holding down the CAN key and pressing an arbitrary function key clears the screen.

Procedure for erase screen display

Procedure

- **Clearing the screen**

Hold down the CAN key and press an arbitrary function key (such as POS and PROG).

- **Restoring the screen**

Press an arbitrary function key.

11.8.2 Automatic Erase Screen Display

The CNC screen is automatically cleared if no keys are pressed during the period (in minutes) specified with a parameter. The CNC screen is restored by pressing any key.

Procedure for automatic erase screen display

- **Clearing the screen**

The CNC screen is cleared once the period (minutes) specified with parameter No. 3123 has elapsed, provided the following conditions are satisfied:

Conditions for clearing the CNC screen

- Parameter No. 3123 is set to other than 0.
- None of the following keys have been pressed:
  MDI keys
  Soft keys
  External input keys
- No alarm has been issued.

- **Restoring the screen**

The cleared CNC screen is restored once at least one of the following conditions is satisfied:

Conditions for restoring the CNC screen

- Any of the following keys has been pressed:
  MDI keys
  Soft keys
  Externally input keys
- An alarm has been issued.

Some machines feature a special key for restoring the screen. For an explanation of the location and use of this key, refer to the corresponding manual, supplied by the machine tool builder.

Explanations

- **Clearing the screen using CAN + function key**

If parameter No. 3123 is set to 0, clearing of the screen using the CAN key and a function key (III–11.8.1) is disabled.

CAUTION

Pressing any key while the screen is being cleared restores the screen. In such a case, however, the function assigned to the pressed key is initiated. Do not press the DELET, INSERT, or ALTER key to restore the screen, therefore.

12 GRAPHICS FUNCTION

Two graphic functions are available. One is a graphic display function, and the other is a dynamic graphic display function.

The graphic display function can draw the beam path specified by a program being executed on a screen. The graphic display function also allows enlargement and reduction of the display.

The dynamic graphic display function can draw a beam path and machining profile.
In beam path drawing, automatic scaling and solid drawing are possible.
In machining profile drawing, the status of machining in progress can be drawn through simulation. Blank figures can also be drawn.

This chapter mainly explains drawing procedures and drawing parameters for the following:

1. Drawing the beam path specified by a program being executed, with the graphic display function
2. Drawing the beam path with the dynamic graphic display function
3. Drawing the machining profile with the dynamic graphic display function

12.1 GRAPHICS DISPLAY

It is possible to draw the programmed beam path on the screen, which makes it possible to check the progress of machining, while observing the path on the screen.
In addition, it is also possible to enlarge/reduce the screen.
Before drawing, graphic parameters must be set.
When the dynamic graphics function is used, the graphics function described in this section cannot be used. See Section 12.2 for the dynamic graphics function.

Graphics display procedure

Procedure

1 Press function key GRAPH . Press CUSTOM GRAPH for a small MDI unit.

The graphic parameter screen shown below appears. (If this screen does not appear, press soft key **[PARAM]**.)

```
GRAPHIC PARAMETER                                    O0000 N00000

   AXES        P=    4
       (XY=0.YZ=1,ZY=2, XZ=3, XYZ=4, ZXY=5)
   RANGE       (MAX.)
   X=    115000        Y=    150000      Z=       0
   RANGE       (MIN.)
   X=         0        Y=         0      Z=       0
   SCALE               K=        70
   GRAPHIC CENTER
   X=     57500        Y=     75000      Z=       0
   PROGRAM STOP        N=         0
   AUTO ERASE          A=         1

MDI   ****  ***  ***                 14 : 23 : 54
( PARAM )( GRAPH )(       )(       )(       )
```

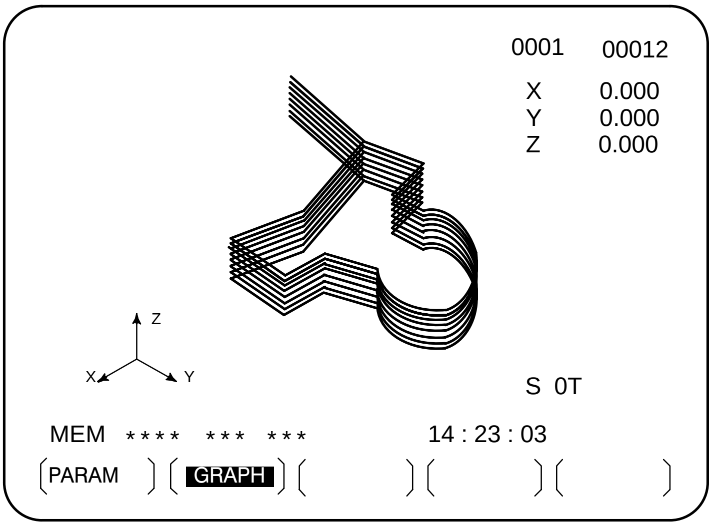
2 Move the cursor with the cursor keys to a parameter to set.

3 Enter data, then press the INPUT key.

4 Repeat steps 2 and 3 until all required parameters are specified.

5 Press soft key **[GRAPH]**.

6 Automatic operation is started and machine movement is drawn on the screen.



Explanation

- **RANGE (Actual graphic range)**

The size of the graphic screen will be as follows:

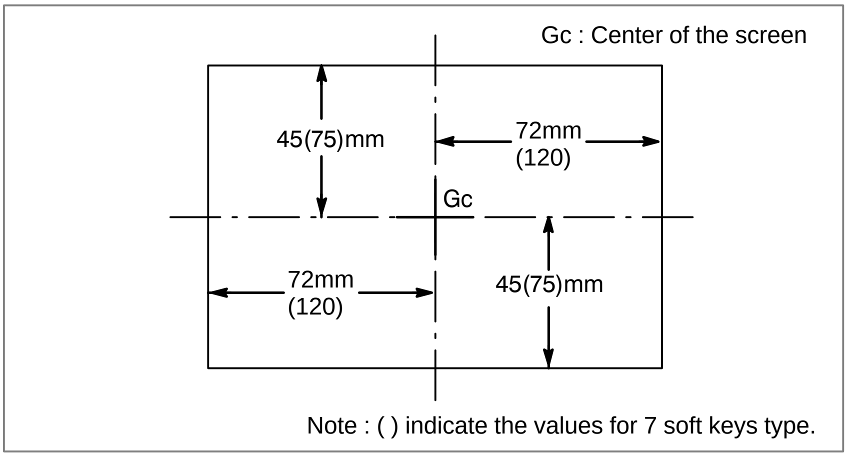


Fig.12.1 (a) Graphic range

As shown in Fig.12.1 (a), the maximum graphics range is an area of approx. 144 mm (width) × 90 mm (height) for 7 soft keys type and approx. 240 mm (width) ×150 mm (height) for 12 soft keys type.

- **Setting the graphics range**

To draw a section of the program within the actual graphics range, set the graphics range using one of the following two methods:

1 . Set the center coordinates of the range and the magnification.

2 . Set the maximum and minimum coordinates for the range in the program.

Whether 1 or 2 is used depends on which parameters are set last. A graphics range which has been set is retained when the power is turned off.

1. Setting the center coordinate of the graphics range and graphics magnification

Set the center of the graphic range to the center of the screen. If the drawing range in the program can be contained in the above actual graphics range, set the magnification to 1 (actual value set is 100).

When the drawing range is larger than the maximum graphics range or much smaller than the maximum graphics range, the graphics magnification should be changed. The graphics magnification is 0.01 to 100.00 times, which is usually determined as follows;

Graphics magnification=Graphics magnification (**H**), or graphics magnifications (**V**), whichever is smaller

Graphics magnification **H** =α/(length on program to horizontal direction axis)

Graphics magnification **V**=β/(length on program to vertical direction axis)

α:144mm (for 7 soft keys type)
α:240mm (for 12 soft keys type)
β:90mm (for 7 soft keys type)
β:150mm (for 12 soft keys type)

The graphics magnification is always based on the center of the screen.

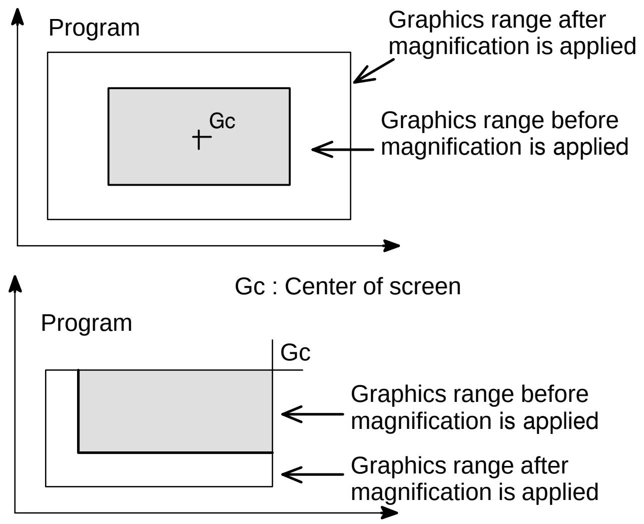


Fig.12.1 (b) Applying graphics magnification (Example of enlargement)

2. Setting the maximum and minimum coordinates for the drawing range in the program

When the actual beam path is not near the center of the screen, method 1 will cause the beam path to be drawn out of the geaphics range if graphics magnification is not set properly.
To avoid such cases, the following six graphic parameters are prepared;

Graphic range (Max.) X
Graphic range (Max.) Y
Graphic range (Max.) Z
Graphic range (Min.) X
Graphic range (Min.) Y
Graphic range (Min.) Z

With the above parameters, the center of screen (Gcx, Gcy, Gcz) is determined by the CNC as follows;

Gcx = (X (MAX.) + X (MIN.)) /2
Gcy = (Y (MAX.) + Y (MIN.)) /2
Gcz = (Z (MAX.) + Z (MIN.)) / 2

The unit of the value will be 0.001 mm or 0.0001 inch depending on the input unit.
Graphics magnification is applied automatically. When the graphics range is specified, the center coordinates and magnification do not need to be calculated.

- **Work coordinate system and graphics**

The graphic origin and graphic center point will not be changed even if the workpiece coordinate origin is changed.
In other words, the workpiece coordinate origin is always consistent with the graphic origin.

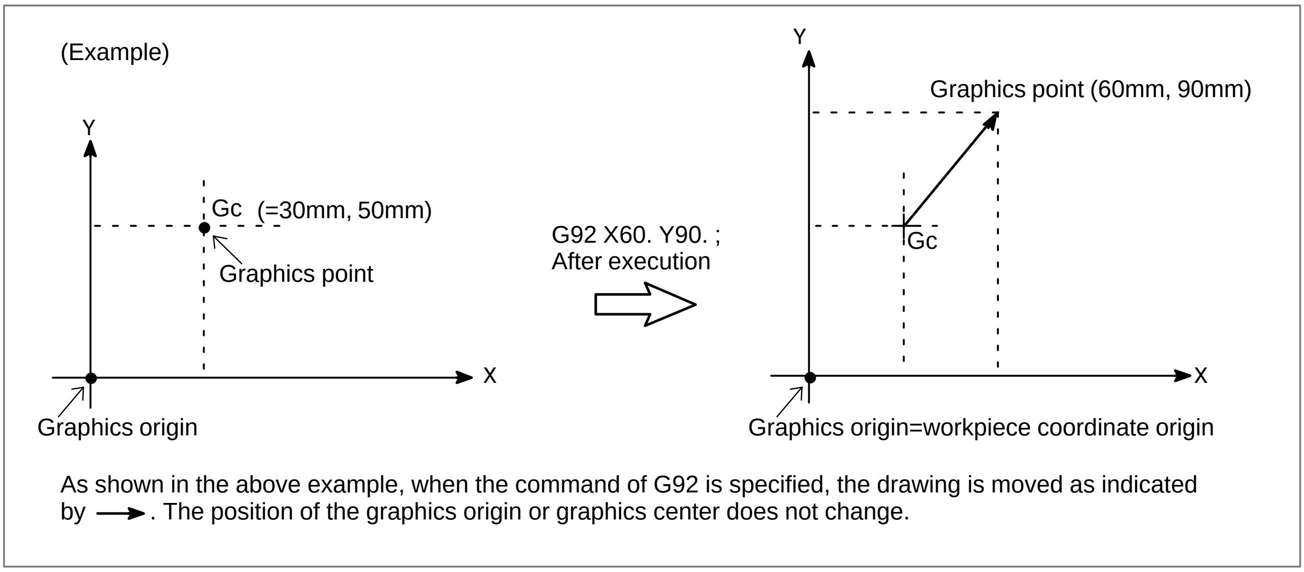
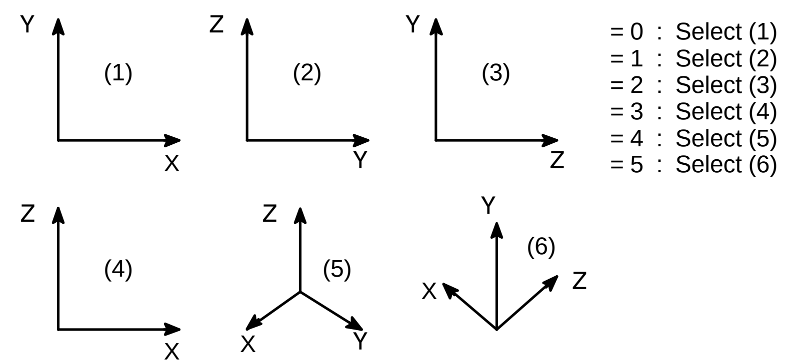


Fig.12.1 (c) Workpiece coordinate origin and graphics origin

• Graphics parameter

· **AXES**

Specify the plane to use for drawing. The user can choose from the following six coordinate systems.



The rotating angle (horizontal, vertical) for isometric display of 5) and 6) is fixed at 45°in both cases.

Fig.12.1 (d) Coordinate system

· **RANGE (Max., Min.)**

Set the graphic range displayed on the screen by specifying maximum and minimum values along each axis.

X=Maximum value X=Minimum value
Y=Maximum value Y=Minimum value
Z=Maximum value Z=Minimum value
Valid range: 0 to ±9999999

NOTE

1 The units are 0.001 mm or 0.0001 inch. Note that the maximum value must be greater than the minimum value for each axis.
2 When setting the graphics range with the graphics parameters for the maximum and minimum values, do not set the parameters for the magnification and screen center coordinates afterwards. Only the parameters set last are effective.

· **SCALE**

Set the graphic magnification
The setting range is 0 to 10000 (unit:0.01 time).

· **GRAPHIC CENTER**

X=_
Y=_
Z=_
Set the coordinate value on the workpiece coordinate system at graphic center.

> **NOTE**
> 1 When MAX. and MIN. of RANGE are set, the values will be set automatically once drawing is executed
> 2 When setting the graphics range with the graphics parameters for the magnification and screen center coordinates, do not set the parameters for the maximum and minimum values afterward. Only the parameters set last are effective.

· **PROGRAM STOP**

N=_
Set the sequence No. of the end block when necessary to partially display.
This value is automatically cancelled and set to –1 once drawing is executed.

· **AUTO ERASE**

1 : Erase the previous drawing automatically when the automatic operation is started under reset condition.
0 : Not erase automatically.

- **Executing drawing only**

Since the graphic drawing is done when coordinate value is renewed during automatic operation, etc., it is necessary to start the program by automatic operation. To execute drawing without moving the machine, therefore, enter the machine lock state.

- **Deleting the previous drawing**

When the AUTO operation is started under reset condition, the program is executed after deleting the previous drawing automatically (Automatic deleting=1). It is possible not to delete the previous drawing by graphic parameter (Automatic deleting=0).

- **Drawing a part of a program**

When necessary to display a part of a program, search the starting block to be drawn by the sequence No. search, and set the sequence No. of the end block to the PROGRAM STOP N= of the graphic parameter before starting the program under cycle operation mode.

- **Drawing using dashed lines and solid lines**

The tool path is shown with a dashed line (- - - -) for rapid traverse and with a solid line (——) for cutting feed.

Limitations

- **Feedrate**

In case the feed rate is considerably high, drawing may not be executed correctly, decrease the speed by dry–run, etc. to execute drawing.

12.2 DYNAMIC GRAPHIC DISPLAY

There are the following two functions in Dynamic Graphics.

Path graphic	This is used to draw the path of beam center commanded by the part program.
Solid graphic	This is used to draw the workpiece figure machined by tool movement commanded by the part program.

The path graphic function is used to precisely check the part program for drawing the beam path with a line. The solid graphic function is used to draw the workpiece figure to be machined with a program. Thus, it is easy to recognize roughly the part program. These two functions can be used freely by switching them.

12.2.1 Path Drawing

The path graphic feature calls a program from memory and draws the tool path specified by the program. This feature provides the following functions.

1. Drawing plane

The user can choose the drawing plane from four types of plane views, two types of isometric projection views, and biplane view.

2. Drawing rotation

When an isometric projection view is used, the drawing can be rotated horizontally and vertically.

3. Drawing enlargement and reduction

A drawing can be enlarged or reduced by specifying a magnification from 0.01 to 100 with respect to the actual size. In addition, a drawing can be automatically enlarged or reduced by setting maximum and minimum values.

4. Partial drawing

A range of the program can be drawn by specifying a starting sequence number and ending sequence number.

5. Programmed path and beam path drawing

The user can specify whether to apply length offset and cutter compensation to drawing. This way, either the actual programmed path or the beam path can be drawn.

6. Color

When a beam path is drawn on a screen, the colors used can be chosen from seven colors including white. The color of the beam path can be changed according to the T code.

7. Automatic scaling

The CNC automatically determines the maximum and minimum drawing coordinates for each program. This means that drawing can be performed with a magnification automatically determined according to these maximum and minimum values.

8. Partial enlargement drawing

Except for biplane views the user can enlarge all types of drawings by a factor of up to 100 while looking at the drawing that has been made.

9. Indicating the current beam position with a mark

The current nozzle position can be displayed on the screen.

10. Indicating the coordinates of the current position

The current position can also be indicated using coordinates.

11. Displaying coordinate axes and actual size dimensions lines

Coordinate axes and actual size dimension lines are displayed together with the drawing so that actual size can be referenced.

The first six functions above (**1.** to **6.**) are available by setting the graphic parameters. The seventh to ninth functions (**7.** to **9.**) are mainly executed using soft keys after drawing has been setup. The tenth function (**10.**) is enabled by setting a parameter. The eleventh function (**11.**) can be used at any time.

Path drawing procedure

Procedure

1 To draw a beam path, necessary data must be set beforehand.
So press the function button GRAPH some times (CUSTOM GRAPH for the small MDI).The “PATH GRAPHIC (PARAMETER)” is displayed.

```
PATH GRAPHIC (PARAMETER–1)          O0000 N00002

AXES         P=        4
(XY=0, YZ=1, ZY=2, XZ=3, XYZ=4, ZXY=5, 2P=6)
ANGLE
 ROTATION          A=           0
 TILTING           A=           0
SCALE              K=        0.00
CENTER OR MAX./MIN.
  X=130.000        Y=    110.000      Z=  50.000
   I=  0.000       J=    –10.000      K=   0.000
START SEQ. NO.     N=           0
END SEQ. NO.       N=           0
NO.    A=

MDI   ****  ***  ***           14 : 25 : 07
[ PARAM ][ EXEC ][ SCALE ][ POS ][      ]
```

```
PATH GRAPHIC (PARAMETER–2)  O0000 N00001

TOOL. COMP         P=       0
COLOR (0123456)
 PATH              P=       0
 TOOL              Q=       0
 AUTO CHANGE       R=       0

MDI   ****  ***  ***           14 : 25 : 51
[ PARAM ][ EXEC ][ SCALE ][ POS ][      ]
```

2 There are two screens for setting drawing parameters.
Press the page key according to the setting items for selecting screens.

3 Set the cursor to an item to be set by cursor keys.

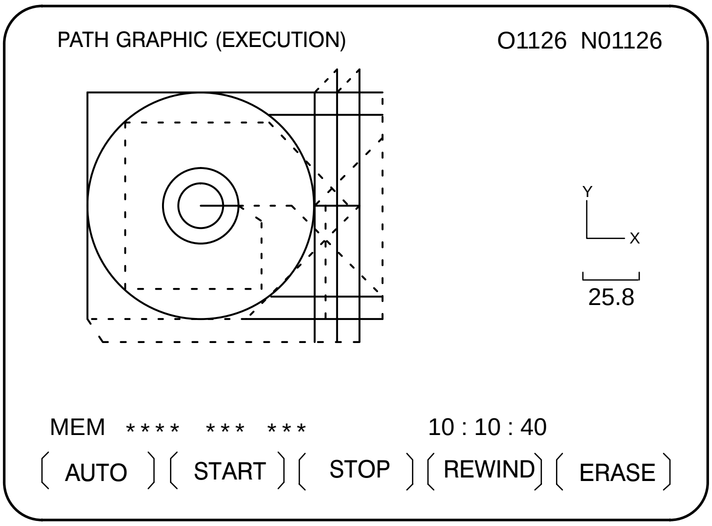
4 Input numerics by numeric keys.

5 Press the INPUT key.

The input numerics are set by these operations and the cursor automatically moves to the next setting items. The set data is held even after the power is turned off.

6 Set the operation mode to the memory mode, press function key PROG, and call the part program which should be drawn.

7 Press function key GRAPH (CUSTOM GRAPH for a small MDI) several times to redisplay the PATH GRAPHIC (PARAMETER) screen, then press soft key **[EXEC]** to display the PATH GRAPHIC (EXECUTION) screen.



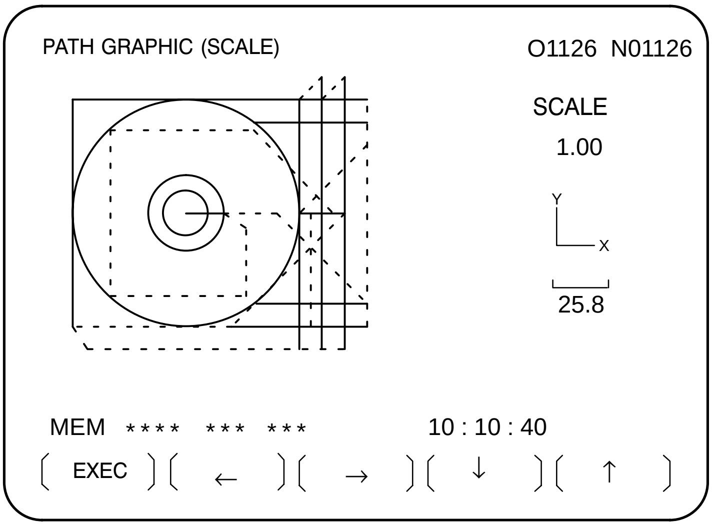
8 Press soft key **[(OPRT)],** then press soft key **[AUTO]** or **[START]**. Pressing **[AUTO]** enables automatic scaling. See item **7** in introduction of path drawing and the description of soft key **[AUTO]** in Explanations for details. Drawing is now started. During drawing, the message ”DRAWING” blinks at the lower–right corner of the CRT screen.

9 Press soft key **[STOP]** to pause drawing. The indication of “STOP” blinks at the lower right corner on the CRT screen. Press soft key **[START]** to start drawing. In addition, press soft key **[REWIND]** to redraw from the top of program before pressing soft key **[START]**.

10 Execute the last of part program (M02/M30) to end drawing. This will cause, blinking of the “DRAWING” light to turn off. The beam path view drawn can be retained until the power is turned off unless a new beam path view is drawn.

Partial enlargement

11 **For partial drawing enlargement,** display the PATH GRAPHIC (SCALE) screen by pressing the soft key **[ZOOM]** on the PATH GRAPHIC (PARAMETER) screen of **step 1** above. The beam path is displayed. Next, press soft key **[(OPRT)]**.



12 Perform positioning of marks displayed at the center of the screen to the center of the part enlarged using soft keys [←], [→], [↓], and [↑].

13 Set the relative magnification rate for the beam path view which is being drawn using the address keys “P” and “M”. When you press address key P or M, the following results:

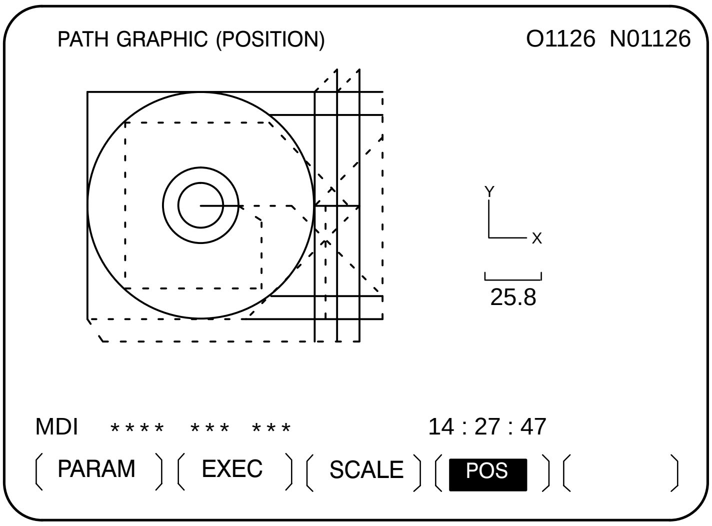
Address key	Function
P	The relative magnification rate increases by 0.1.
M	The relative magnification rate decreases by 0.1.

The relative magnification rate is continuously changed by keeping the address keys depressed. It is possible to magnify up to 100 times in reference to the actual dimensions.

14 Press the soft key **[EXEC]** after setting the relative magnification rate. Then, the screen automatically changes to “BEAM PATH (EXECUTION)” and the drawing of set partial enlargement view starts. The set partial enlargement status is valid until soft key **[AUTO]** or **[ERASE]** is pressed.

Mark display

15 **To display a mark at the current nozzle position,** display the PATH GRAPHIC (POSITION) screen by pressing soft key **[POS]** on the PATH GRAPHIC (PARAMETER) screen of **step 1** above. This mark blinks at the current nozzle center position on the beam path.



Explanations

• AXES

The relationship between the setting value and drawing screen is as shown below:

Setting value	Drawing screen
0	Plane view (XY)
1	Plane view (YZ)
2	Plane view (ZY)
3	Plane view (XZ)
4	Isometric projection (XYZ)
5	Isometric projection (ZXY)
6	Biplane view (XY,XZ)

• Plane view (XY,YZ,ZY,XZ)

The following coordinate systems are selected.

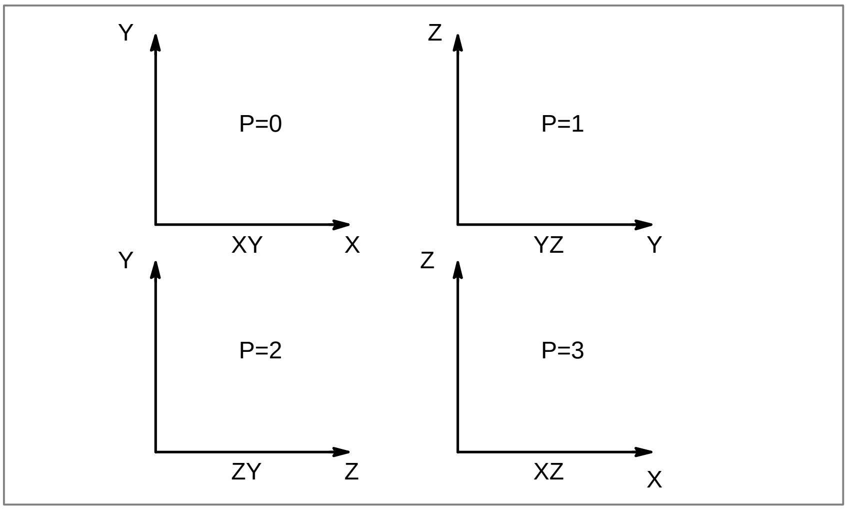


Fig.12.2.1 (a) Coordinate systems for the plane view

- **Isometric projection (XYZ,ZXY)**

Projector view by isometric can be drawn.

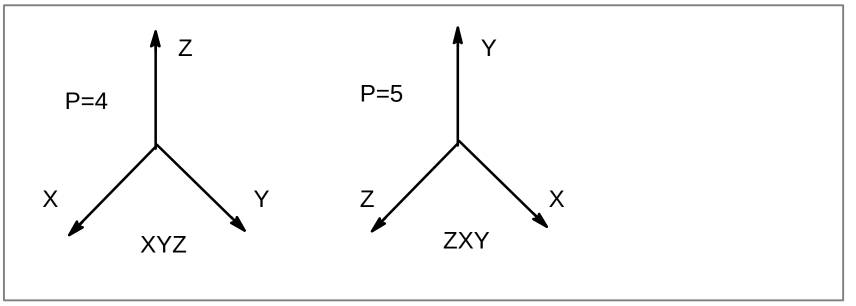


Fig.12.2.1 (b) Coordinate systems for the isometric projection

- **Biplane view**

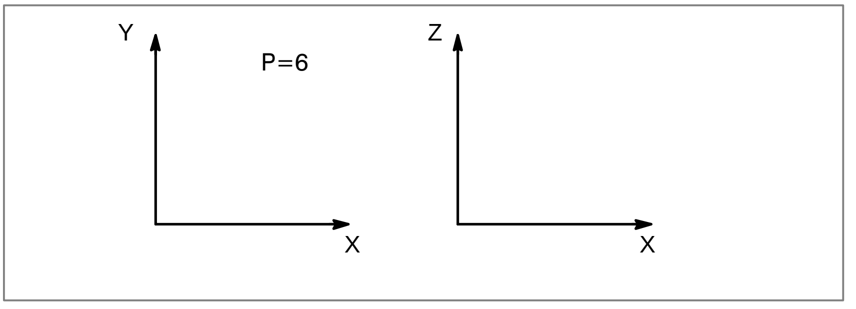


Fig.12.2.1 (c) Coordinate systems for the biplane view

Biplanes (XY and XZ) can be drawn simultaneously. The maximum and minimum coordinate values must be set to draw the biplane view. The maximum and minimum coordinate values can also be set by performing automatic scaling

- **ANGLE**

The direction of the coordinate axis is set when the isometric projection is the setting of the drawing screen. The direction is set by horizontal and vertical rotation angles. The unit is expressed in degrees.

- **ROTATION**

The horizontal rotation angle is set in the range of –180°to +180°in reference to the vertical axis. Set a positive value for clockwise rotation of the coordinate axis. Thus, the direction of projection (visual field) becomes counterclockwise.

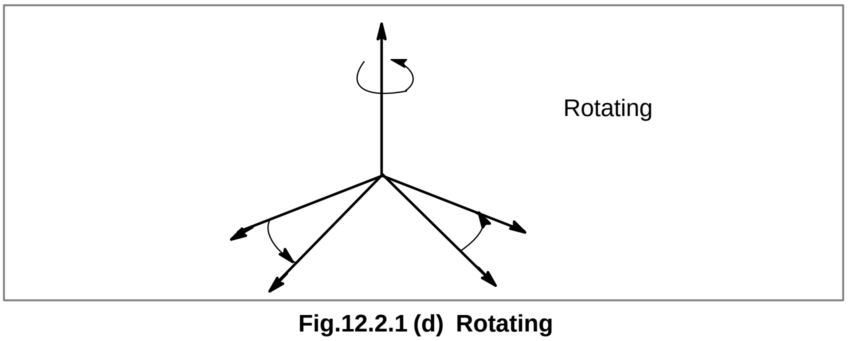


Fig.12.2.1 (d) Rotating

- **TILTING**

The tilting angle of the vertical axis is set in the range of –90°to +90°in reference to the horizontal axis crossing the vertical axis at a right angle. When a positive value is set, the vertical axis slants to the other side of the graphic screen. Thus, the projection direction (visual field) becomes the horizontal direction.

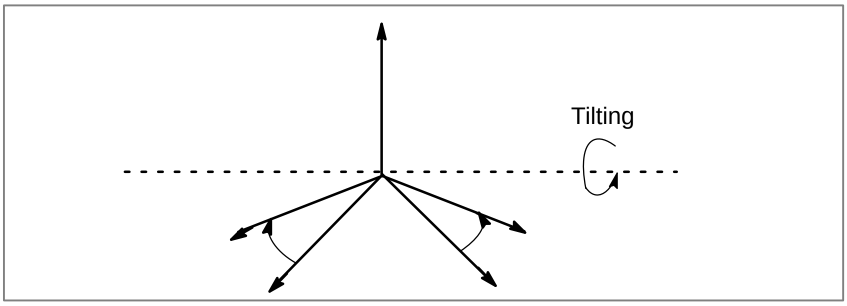


Fig.12.2.1 (e) Tilting

- **SCALE**

Set the magnification rate of drawing from 0.01 to 100.00. When 1.0 is set, drawing is carried out in actual dimensions. When 0 is set, the drawing magnification rate is automatically set based on the setting of maximum and minimum coordinate values of drawing.

- **CENTER OR MAX./MIN.**

When a graphics (drawing) magnification of 0 is set, maximum coordinates on the X–axis, Y–axis, and Z–axis in the workpiece coordinate system must be set in addresses X, Y, and Z, and minimum coordinates must be set in addresses I, J, and K, to specify the graphics (drawing) range. For biplane view drawing, maximum and minimum coordinates for drawing must be specified.
When a drawing magnification other than 0 is set, the X, Y, and Z coordinates of the drawing center in the workpiece coordinate system must be set in addresses X, Y, and Z. Addresses I, J, and K are not used.
The table below summarizes the setting requirements described above.

Setting the drawing magnification rate	Setting	
	Address X/Y/Z	Address I/J/K
Other than 0	Drawing center coordinate value of X, Y, and Z axes	Ignored
0 or biplane view drawing	Drawing maximum coordinate value of X, Y, and Z axes	Drawing minimum coordinate value of X, Y, and Z axes

- **START SEQ. NO. and END SEQ. NO.**

Set the start and end sequence numbers of drawing in five digits each. The part program for drawing is executed from the head and only the part enclosed by the start sequence and end sequence numbers is drawn. When 0 is commanded as the start sequence number, drawing is performed from the head of the program. In addition, when 0 is commanded as the end sequence number, drawing is performed up to the end of program. The sequence number is referred to regardless of either main program or subprogram.

- **Cutter compensation**

It is possible to set whether the beam path is drawn by making the beam length offset or cutter compensation valid or invalid.

Setting value	Cutter compensation
0	Perform drawing by making cutter compensation valid (An actual beam path is drawn.)
1	Perform drawing by making cutter compensation invalid (A programmed path is drawn.)

Always set 0 before drawing when indicating the mark of the current nozzle position.

- **COLOR**

Specify the color of the beam path. In the case of monochrome it is not required to set it. The relationship between the setting value and color is as shown below:

Setting value	Color
0	White
1	Red
2	Green
3	Yellow
4	Blue
5	Purple
6	Light blue

· **PATH** Specify the color of the beam path.
· **TOOL** Specify the color of the current position mark of the beam.
· **AUTO CHANGE** Set if for changing the color of the beam path automatically according to the T –code command.

Setting value	Function
0	The color of the beam path is not changed.
1	The color of the beam path is changed automatically.

When 1 is set, the setting value of the color designation of PATH is incremented by 1 every time the T code is commanded. At the same time, the color of the beam path changes. If the setting value exceeds 6, it returns to 0.

- **Soft key functions on the “PATH GRAPHIC [EXECUTION]”screen**

Software key	Function
[AUTO]	Automatic scaling is performed. Obtain the maximum and minimum coordinates of the part program before performing drawing, specify them for the maximum and minimum values of drawing parameters, and set the drawing magnification rate to 0 before starting drawing. Thus, the beam path view is properly laid out on the screen.
[START]	Drawing starts. When the **[START]** is pressed while the drawing is not in STOP, the part program starts from the top of the part program. Press the **[START]** while the drawing is in stop to allow drawing to be carried out continuously.
[STOP]	Stop drawing. (Single block stop)
[REWIND]	Press this key to start drawing from the top of part program. Searches for the beginning of a part program.
[ERASE]	Erase the beam path view which has been drawn.

- **Graphic program**

No part program which has not been registered in memory can be drawn. Also, it is necessary that the M02 or M30 should be commanded at the end of the part program.

- **Mark for the nozzle current position**

The period of mark blinking is short when the nozzle is moving and becomes longer when the nozzle stops.
The mark indicating the current position of nozzle is displayed on the XY plane view when the biplane drawing is performed.

- **Position mark**

Parameter No. 6501 (CSR, bit 5) is used to specify whether to use ■ or x as the mark for indicating the current beam position and the center of a partially enlarged drawing.

- **Display of the coordinate value**

Parameter No. 6500 (DPO, bit 5) is used to specify whether to display the coordinates of the current position on the beam path drawing screen.

- **Changing the coordinate system**

If a program specifies a coordinate system change, parameter 6501 (ORG, bit 0) is used to specify whether to draw without changing the coordinate system or to draw by regarding the current drawing position as the current position in the new coordinate system.

Restrictions

- **Graphic condition**

If machine operation is not allowed, no drawing can be carried out. No drawing can be made during machine operation. The setting data and switches required for drawing are as shown below:

Setting data and switch	Status
Tool offset amount	Set it properly when performing drawing while the tool offset amount becomes valid.
Single block	Off
Optional block skip	Set it properly.
Feed hold	Off

- **Partial enlargement**

The partial enlargement can be carried out on the plane view and isometric projection view. No partial enlargement can be made in the drawing of the biplane view.

- **Nozzle current position**

In dynamic graphics display, drawing cannot be executed while the machine is operating even though this is possible in ordinary graphics display (see III–12.1). However, after drawing is executed, the operator can see how the beam moves along the beam path by operating the machine while displaying the mark for the current position of the nozzle. It is necessary that the setting data and switches related to the machine operation should be the same status between drawing operation and machining operation for properly displaying the current position of nozzle on the drawn beam path.

12.2.2 Solid Graphics

The solid graphics draws the figure of a workpieces machined by the movement of a beam.
The following graphic functions are provided :

1. Solid model graphic

Solid model graphic is drawn by surfaces so that the machined figure can be recognized concretely.

2. Blank figure graphics

It is possible to draw a blank figure before machining.
A rectangular parallelepiped and a circular column or cylinder can be drawn. A circular column or cylinder parallel to the X–axis, Y–axis, or Z–axis can be selected.

3. Drawing of machining progress

It is possible to draw the progress of machining by simulation.

4. Drawing of final machined figure

It is possible to draw the final finish machined figure.

5. Changing of drawing direction

The user can choose from four drawing directions and eight tilting angles.

6. Plane view graphics

It is possible to draw XY plane views as well as solid model views. Height of the workpiece is discriminated by color for color or brightness for monochrome.

7. Triplane view graphic

In addition to a solid drawing, a triplane view can be drawn. The user can choose from four types of plane view and side view positions. The user can freely change the cross–section position of a side view.

8. Horizontal hole machining

It is possible to install nozzle in the direction which is parallel to the X or Y axis as well as the Z axis.

Solid graphics drawing procedure

Procedure

1 To draw a machining profile, necessary data must be set beforehand. So press the function key GRAPH (CUSTOM GRAPH for the small MDI).
The screen of "SOLID GRAPHIC (PARAMETER)" is displayed.

```
SOLID GRAPHIC (PARAMETER)                      O0000 N00003
BLANK FORM              P=
  X=    0.000     Y=    0.000      Z=    0.000
  I=  120.000     J=  100.000      K=   40.000
TOOL FORM         P=        2
                  R=    7.500      K=    0.000
PROJECTION        P=        3
                  Q=        1      R=        0
INTENSITY         P=        4
(0123456)         Q=        2      R=        1
START SEQ.NO.     N=        0
END SEQ.NO.       N=        0
ANIM. SPEED       N=        0

 >_
 MDI   ****  ***  ***              14 : 42 : 17
[ PARAM ] [ BLANK ] [ EXEC ] [ REVIEW ] [       ]
```

2 Use a cursor key to move the cursor to an item to be set.

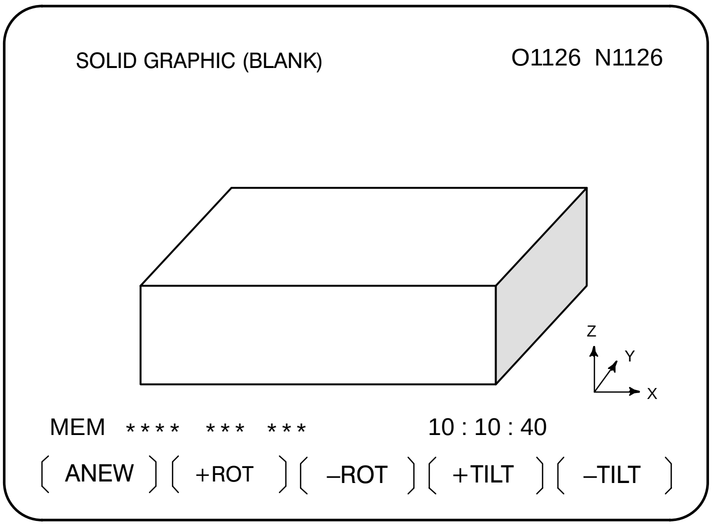
3 Input numerics for the item at the cursor using the numeric key.

4 Press the INPUT .

Input numerics can be set by these operations and the cursor moves to the next setting item automatically. The set data is retained even if the power is turned off.
See Explanations for details on settings.

- **SOLID GRAPHIC (BLANK)**

5 **To draw a blank figure,** display the SOLID GRAPHIC (BLANK) screen by pressing soft key **[BLANK]** on the SOLID GRAPHIC (PARAMETER) screen of **step 1** above.



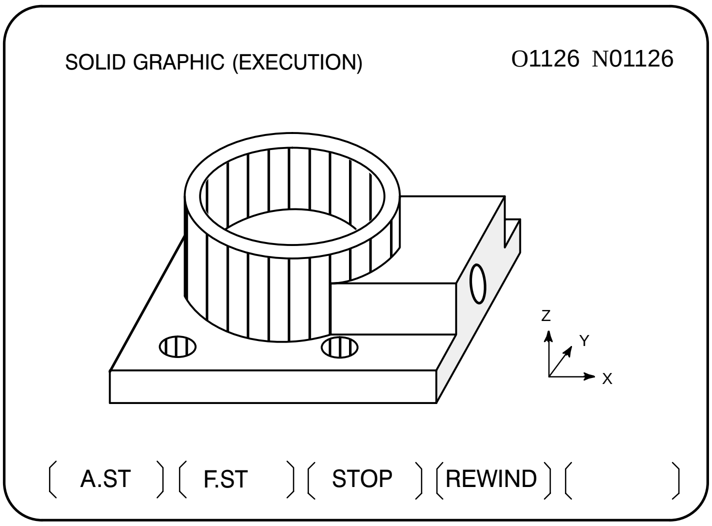
6 Press soft key **[ANEW]**. This allows the blank figure drawing to be performed based on the blank figure data set.

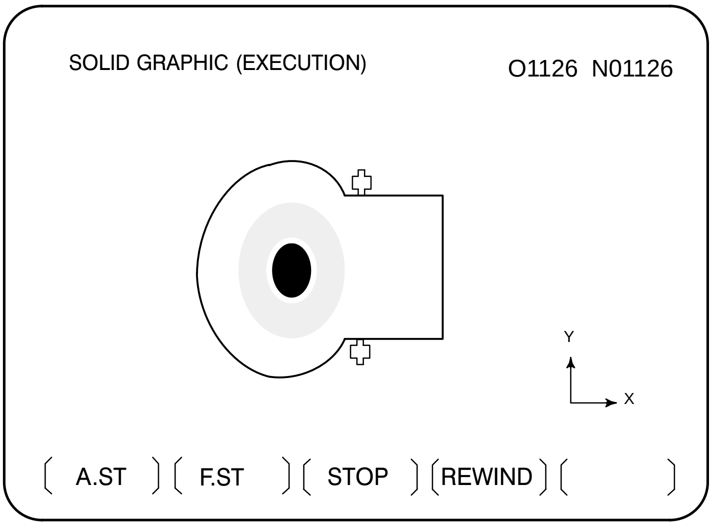
7 Press soft keys **[+ROT] [–ROT] [+TILT]**, and **[–TILT]**, when performing drawing by changing the drawing directions. Parameters P and Q for the drawing direction are changed and the figure is redrawn with the new parameters.

- **SOLID GRAPHICS (EXECUTION)**

8 Set the operation mode to the memory mode, press function key PROG, and call the subject part program of drawing.

9 **To draw a machining profile,** display the SOLID GRAPHIC (EXECUTION) screen by pressing soft key **[EXEC]** on the SOLID GRAPHIC (PARAMETER) screen of **step 1** above.





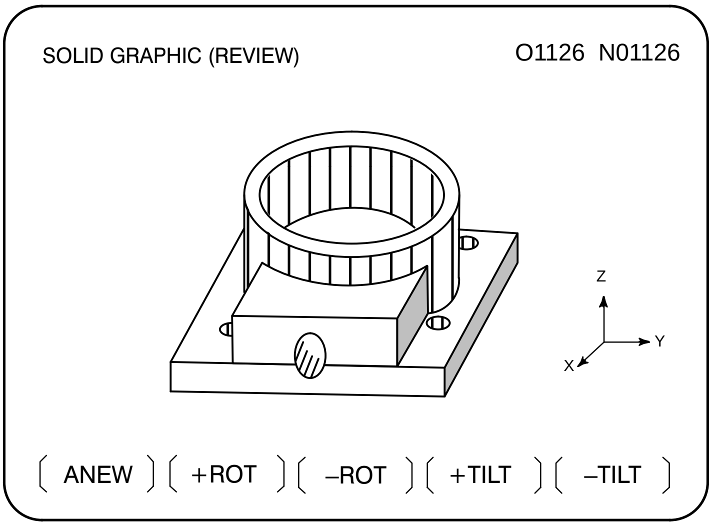
10 Press soft key **[(OPRT)]** and press either soft key **[A.ST]** or **[F.ST]**. When **[A.ST]** is pressed, the status of machining in progress is drawn by simulation. When **[F.ST]** is pressed, the profile during machining is not drawn. Only the finished profile produced by the program is drawn. This allows drawing to be started.
When “STOP” is not displayed at the lower right corner of the screen, the program is executed from its head. “DRAWING” blinks at the lower right corner of CRT screen during drawing.

11 Press soft key **[STOP]** to stop drawing temporarily. Drawing is stopped after drawing the current block and "STOP" blinks at the lower right corner of CRT screen. Press soft key **[A.ST]** or **[F.ST]** when restarting drawing. Press soft key **[REWIND]** and then the **[A.ST]** or **[F.ST]** if redrawing from the head. It is possible to continue drawing after changing the solid graphic parameters in temporary stop.

12 When the end of program (M02 or M03) is executed, the drawing ends and the blinking of "DRAWING" stops. Then, the final finish figure is drawn on the CRT screen. The drawn figure view is retained until the power is turned off as long as a new machine figure view is drawn.

- **REVIEW**

13 The color, intensity, or drawing direction of a machining figure which has been drawn can be changed and the figure redrawn.
To redraw the figure, first change the parameters for the color, intensity, or drawing direction on the SOLID GRAPHIC (PARAMETER) screen shown in **step 1**, then press soft key **[REVIEW]** to display the SOLID GRAPHIC (REVIEW) screen.

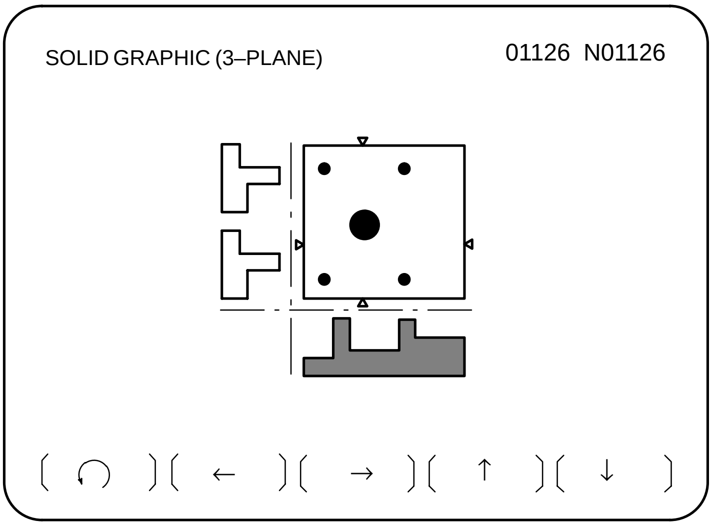


14 Press soft key **[(OPRT)]**, then press soft key **[ANEW]**. The machining figure is redrawn with the color, intensity, or drawing direction set in **step 13**.

15 **To redraw the figure in a different mode,** press soft key **[+ROT]**, **[–ROT]**, **[+TILT]**, or **[–TILT].** Parameters P and Q for the drawing direction are changed and the figure is redrawn with the new paramaters.

- **Triplane view drawing**

16 The machined figure can be drawn on the tri–plane view.
To draw a triplane view, press the rightmost soft key (next–menu key) on the SOLID GRAPHIC (PARAMETER) screen of **step 1** above, then press soft key **[3–PLN]** and **[(OPRT)]**. The SOLID GRAPHIC (3–PLANE) screen appears.



17 Each time soft key [ ] is pressed, the side–view drawings displayed change as follows.

Right view and rear view
↓
Rear view and left view
↓
Left view and front view
↓
Front view and right view

18 The sectional position of side view can be changed by the soft keys [←], [→], [↑], and [↓].
With the sectional position of the left/right side view, the marks ▲ and ▼ indicating the sectional position can be moved using the soft keys [←] and [→].
With the sectional position of rear/front side view, the marks ▶ and ◀ indicating the sectional position can be moved using the soft keys [↑], and [↓]. Keep the keys depressed to change sectional/views continuously.

Explanations

GRAPHICS PARAMETER

- **BLANK FORM**

♦ BLANK FORM (P)

Set the type of blank figure under P. The relationship between the setting value and figure is as follows:

P	Blank figure
0	Rectangular parallelepiped (Cubed)
1	Column or cylinder (parallel to Z–axis)

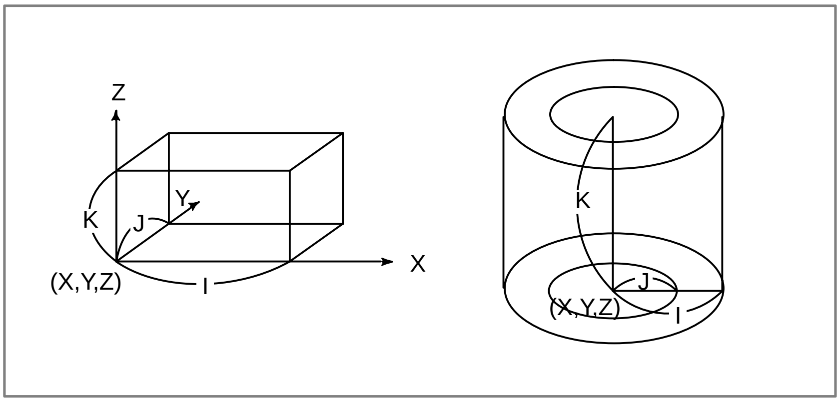
♦ Material positions (X,Y,Z)

Set the X–axis, Y–axis, and Z–axis coordinate values of standard point of materials in workpiece coordinate system to the addresses X, Y, and Z. The standard point of materials is the corner point in the negative direction in the case of rectangular parallelepiped blank figure and the center point of bottom in the case of column and cylinder materials.

♦ Material dimensions (I,J,K)

Set the dimensions of materials. The relationship between the addresses I, J, and K and setting value is as shown below:

Material	I	J	K
Rectangular	Length in X–axis direction	Length in Y–axis direction	Length in Z–axis direction
Column	Radius of circle	0	Length of column
Cylinder	Radius of external circle	Radius of internal circle	Length of cylinder



• NOZZLE FORM

♦ Machining nozzle orientation (P)

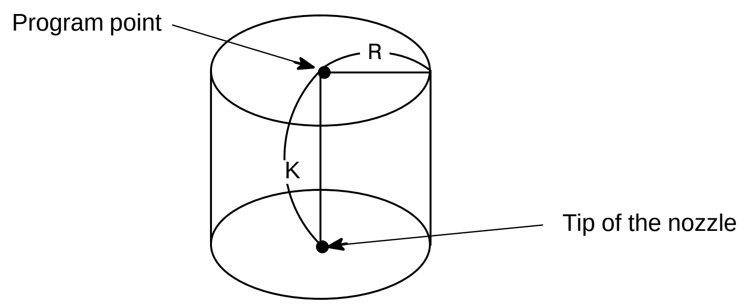
Set the machining direction of nozzle. The relationship between the setting value and machining direction is as shown below.

P	Machining direction of tools
0,1	Parallel to the Z–axis (perform machining from the + direction)

♦ Dimensions of beam (R,K)

Set the dimensions of beam. The relationship between the displayed address and setting value is as shown below:

Address	Setting numerics
R	Radius of beam
K	Distance from the program point to nozzle tip (normally 0)



• PROJECTION

♦ Graphics method and direction (P)

The relationship between graphic method and direction and setting value is as shown below:

P	Graphic method and direction
0, 4	Oblique projection view (+ X–axis)
1, 5	Oblique projection view (+ Y–axis)
2, 6	Oblique projection view (– X–axis)
3, 7	Oblique projection view (–Y–axis)

This setting value can also be incremented or decremented by the soft keys **[+ROT]** or **[– ROT]**. In this case, if the setting value exceeds 7, it returns to 0. If it is smaller than 0, it becomes 7.

In addition, pressing the soft key **[+TILT]** displays a top view and the soft key **[– TILT]** displays a solid drawing.

• VERTICAL AXIS

The vertical axis is fixed to the Z–axis.

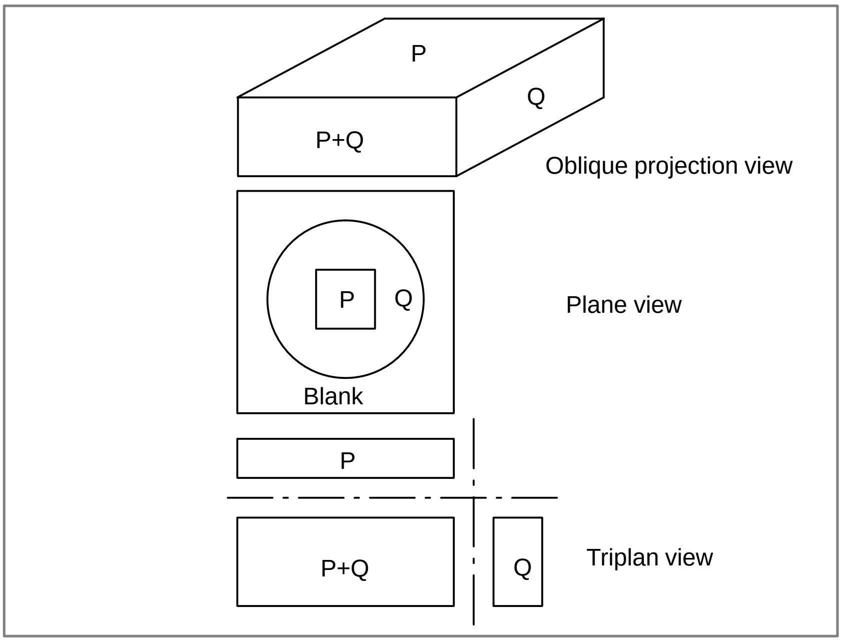
- **INTENSITY**

Specify the intensity of the drawing screen when performing drawing on the monochrome, and the color of the drawing screen when performing drawing on the color screen. The relationship between the setting, intensity, and color is as shown below:
However, when the plane view is displayed on the monochrome. The brighter surface, whichever is specified by P or Q becomes the top surface.

Setting value	Intensity	Color
0	Maximum brightness	White
1	Light ↓ Dark	Red
2		Green
3		Yellow
4		Blue
5		Purple
6		Light blue

The relationship between the display address, surface, and line on the machined figure view is as shown below:

Address	Oblique projection view	Plane view	Triplane view
P	Upper surface	Upper surface	Upper/lower surface
Q	Side surface	Middle surface	Left/right surface
R	Ridge	Ridge	Ridge
Remarks	The intensity/color of front surface is between P and Q	Lower surface is blank	The intensity/color of plane view is between P and Q



- **START SEQ. NO. and END SEQ. NO.**

Specify the start sequence number and end sequence number of each drawing in a five–digit numeric. The subject part program is executed from the head. But only the part enclosed by the start sequence number and end sequence numeric is drawn. When 0 is commanded as the start sequence number, the program is drawn from its head. When 0 is commanded as the end sequence number, the program is drawn to its end. The comparison of sequence number is performed regardless of main program and subprogram.

- **ANIM. SPEED**

Set interval of animated simulation drawing ranging from 0 to 255. Every time the machining proceeds by the number set, the drawing is repeated. If 0 is set, drawing is repeated at every 1 block execution.

- **Soft key functions on the "SOLID GRAPHIC (EXECUTION)"screen**

Soft key	Function
[A.ST]	Simulate and draw the progress of machining.
[F.ST]	No figure during machining is drawn and only the final finish figure by that program is drawn.
[STOP]	When pressed, stops drawing at the end of block (single block stop).
[REWIND]	Press this key to perform drawing from the head of part program. Heading is performed automatically after execution of program end (M02/M30).

- **Graphics program**

No part program which has not been registered in memory can be drawn. It is also necessary that the M02 or M30 be commanded at the end of the part program.

- **Specifying the blank form and beam form in the part program**

It is possible to specify BLANK FORM and BEAM FORM in the part program. The command format is as shown below. If it is commanded during execution of drawing, the item corresponding to the screen of "SOLID GRAPHIC (PARAMETER)" is set and drawing continues with the set data.

- Command of BLANK FORM

```
G10 L90 P_X_Y_Z_I_J_K_ ;
```

The command value succeeding the address is the same as the numeric set to the address being displayed at the item of BLANK FORM of "SOLID GRAPHIC (PARAMETER)". If BLANK FORM is commanded, drawing continues after a new blank figure is drawn.

- Command of BEAM FORM

```
G10 L91 P_R_K_ ;
```

The command value succeeding the address is the same as the numeric set to the address being displayed at the item of BEAM FORM of screen "SOLID GRAPHIC (PARAMETER)". If 0 is commanded with the beam radius value, no machining simulation is performed thereafter.

- **Display of the coordinate value**

Parameter No. 6500 (DPO, bit 5) is used to specify whether to display the coordinates of the current position on SOLID GRAPHIC screen.

- **Graphic method**

Parameter No. 6501 (3PL, bit 2) is used to select whether to draw a triplane view with the third–angle or first–angle projection.

- **Ridge drawing**

Parameter No. 6501 (RID, bit 3) is used to specify whether to draw ridges in plane view drawing.

- **Display mode**

Parameter No. 6501 (FIM, bit 4) is used to specify whether to display a solid graphics in the rough mode or in the fine mode. When a solid graphics is drawn in the fine mode, the drawing speed is slower than when drawn in the rough mode.

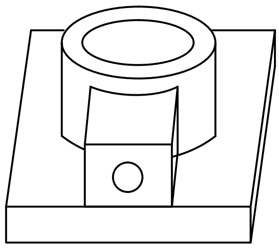
- **Cross section position change**

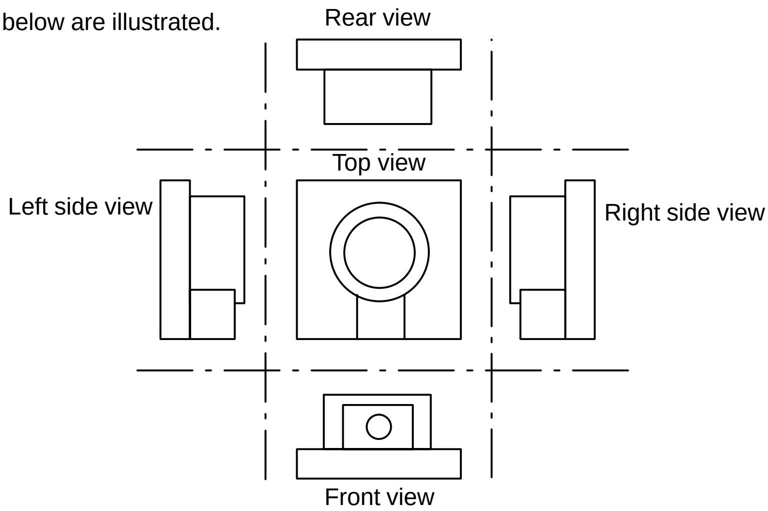
In triplane drawing, a value can be specified for changing the position of the cross section while the soft key is held down. A value from 0 to 10 can be set. When 0 is specified, a value of 1 is used. (Parameter No. 6515)

Examples

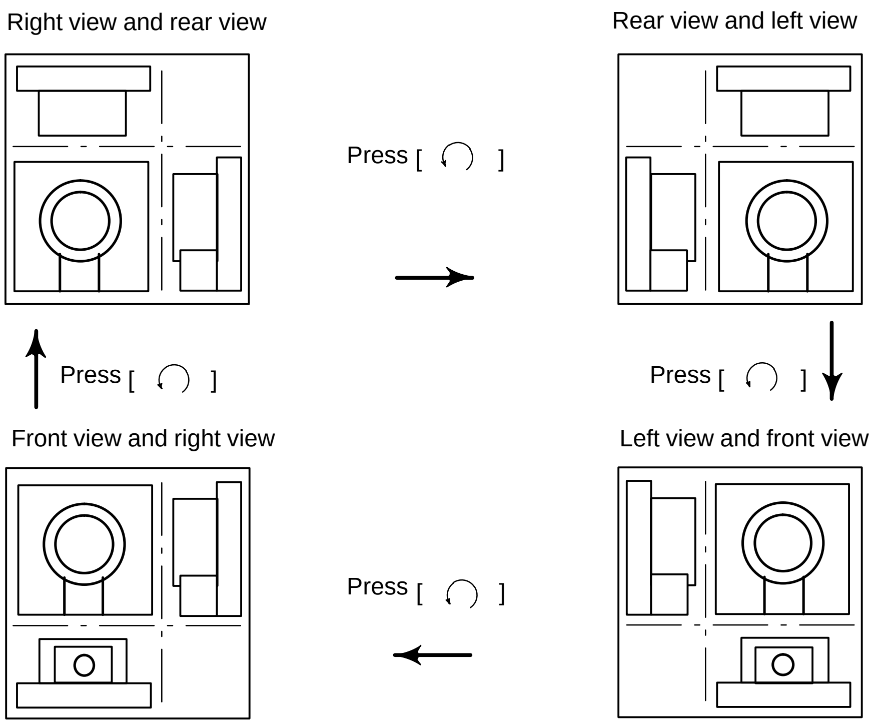
- **Side view selection in triplane drawing**

Example) The side views of the figure below are illustrated.



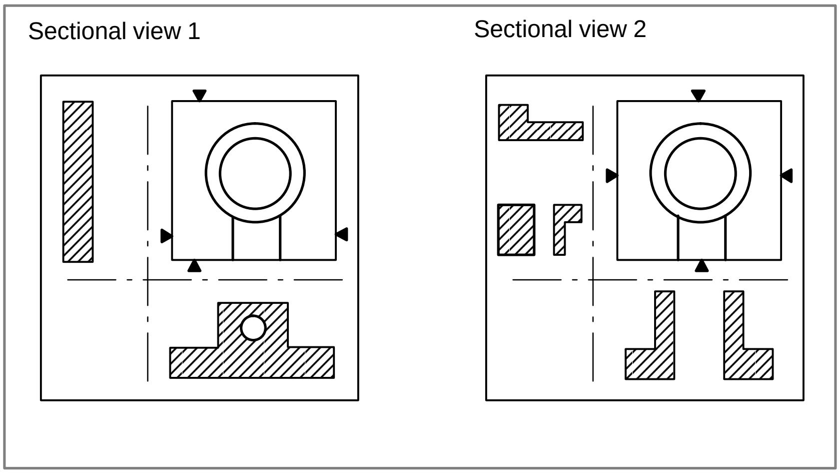


In the above figure, the side views displayed are switched as follows.



- **Cross section position selection in triplane drawing**

Some examples of cross–sectional views are given below for the left view and front view shown on the previous page.



Limitations

- **Graphic condition**

If the machine operation is not enabled, no drawing can be made. No drawing can be made during operation of machine. The main setting data and switches needed on drawing are as shown below:

Setting data/switch	Status needed for drawing
Tool offset value	It is necessary to set the cutter compensation value properly. The beam length offset is ignored.
Single block	Off
Optional block skip	Properly set it.
Feed hold	Off

- **Helical interpolation**

In solid graphics, paths based on helical interpolation cannot be drawn.

13 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  **** *** ***               10:12:25
 [ ALARM ]   [OPERAT]   [ PARAM ]   [        ]   [        ]
```

Fig.13 (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 **[ALARM]** on the HELP (INITIAL MENU) screen to display detailed information about an alarm currently being raised.

```
HELP (ALARM DETAIL)                          O0010 N00001

NUMBER : 027
M'SAGE  : NO AXES COMMANDED IN G43/G44
FUNCTION  : BEAM LENGTH COMPENSATION C
ALARM  :
     IN BEAM LENGTH COMPENSATION TYPE C,
     NO AXIS IS DESIGNATED IN G43 & G44
     BLOCKS. IN BEAM LENGTH COMPENSATION
     TYPE C, IT TRIES TO LATCH ON TO
     ANOTHER AXIS WITHOUT OFFSET CANCE–
     LING.

>100                                     S    0 T0000
MEM **** *** ***              10:12:25
[ ALARM ]   [OPERAT]   [ PARAM ]   [        ]  [        ]
```

Alarm No.
Normal explana–tion on alarm
Function classification
Alarm details

Fig.13 (b) ALARM DETAIL Screen when Alarm P/S 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.

```
HELP  (ALARM DETAIL)                         O1234 N00001


NUMBER       :
M'SAGE       :
FUNCTION     :
ALARM       :

              <<ALARM IS NOT GENERATED>>

   ENTER THE DETAIL–REQUIRED ALARM NUMBER,
    AND PRESS [SELECT] KEY

>100                                     S    0 T0000
MEM **** *** ***              10:12:25
[ ALARM ]   [OPERAT]   [ PARAM ]   [        ]  [        ]
```

Fig.13 (c) ALARM DETAIL Screen when No Alarm is issued

Limitations

- **Laser alarms**

Details on laser alarms are not displayed.

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.

```
>100                                        S    0 T0000
MEM **** *** ***             10:12:25
[         ] [         ] [         ] [         ]  [ SELECT ]
```

Fig.13 (d) How to select each ALARM DETAILS

The following is the screen when P/S alarm 100 is selected as example.

```
HELP (ALARM DETAIL)                         O1234 N00001


NUMBER      :  100
M'SAGE      :  PARAMETER WRITE ENABLE
FUNCTION    :
ALARM       :

                <<ALARM IS NOT GENERATED>>




>100                                        S    0 T0000
MEM **** *** ***             10:12:25
[        ]  [         ]  [         ]  [         ]  [ SELECT ]
```

Fig.13 (e) ALARM DETAIL Screen when P/S 100 is selected

OPERATION METHOD screen

4 To determine an operating procedure for the CNC, press the soft key **[OPERAT]** key on the HELP (INITIAL MENU) screen. The OPERATION METHOD menu screen is then displayed.

```
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
  [ ALARM ]   [OPERAT]   [ PARAM ]   [         ]   [ (OPRT) ]
```

Fig.13 (f) 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  **** *** ***                   10:12:25
[        ] [        ]  [         ] [        ]  [ SELECT ]
```

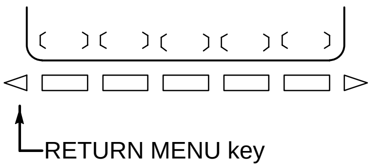
Fig.13 (g) How to select each OPERATION METHOD screen

When "1. PROGRAM EDIT" is selected, for example, the screen in Figure 13 (h) 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.

```
HELP (OPERATION METHOD)                01234 N00001      --> Each item
<< 1.  PROGRAM EDIT >>                          1/4      --> Page/maximum page
*DELETE ALL PROGRAMS                                     --> Operation
  MODE        :  EDIT                                    --> Set mode
  SCREEN      :  PROGRAM                                 --> Operation location
  OPR         :  (O–9999) – <DELETE>                     --> Operating procedure

*DELETE ONE PROGRAM
  MODE        :  EDIT
  SCREEN      :  PROGRAM
  OPR         :  (O+PROGRAM NUMBER) – <DELETE>


>_                                    S    0 T0000
MEM    ****  ***  ***            10:12:25
[ ALARM ]   [OPERAT]   [ PARAM ]   [        ]  [        ]
```

Fig.13 (h) Selected OPERATION METHOD screen



5 To return to the OPERATION METHOD menu screen, press the RETURN MENU key to display "**[OPERAT]**" again, and then press the **[OPERAT]** key again.

To directly select another OPERATION METHOD screen on the screen shown in Figure 13 (h), enter an item No. from the keyboard and press the **[SELECT]** key.

```
>3                                         S    0 T0000
MEM    **** *** ***                 10:12:25
[        ] [        ] [         ] [         ] [ SELECT ]
```

Fig.13 (i) 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 **[PARAM]** key on the HELP (INITIAL MENU) screen. A list of parameter Nos. for each function is displayed. (See Figure 13 (j).)

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.

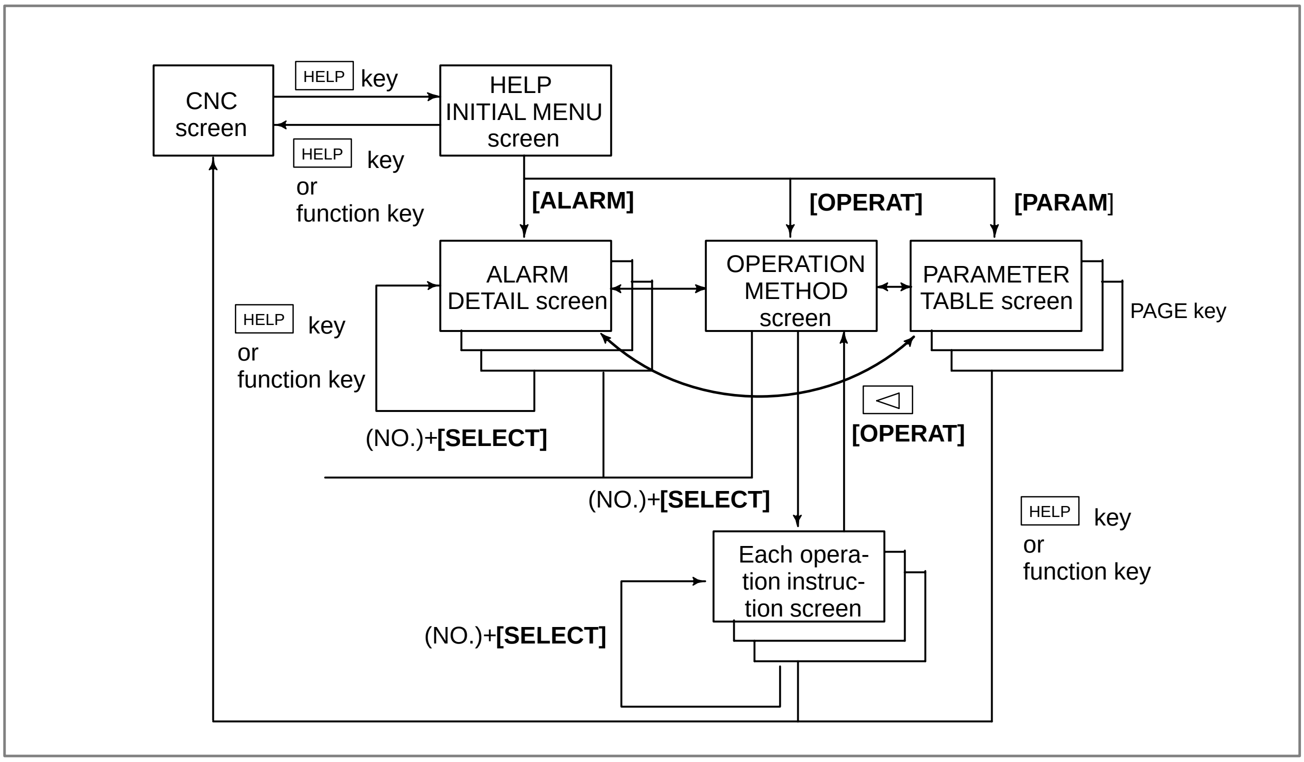
```
HELP (PARAMETER TABLE)                      01234  N00001
                                                      1/4

* SETTEING                                    (No. 0000~)
* READER/PUNCHER INTERFACE                    (No. 0100~)
* AXIS CONTROL
  /SETTING UNIT                               (No. 1000~)
* COORDINATE SYSTEM                           (No. 1200~)
* STROKE LIMIT                                (No. 1300~)
* FEED RATE                                   (No. 1400~)
* ACCEL/DECELERATION CTRL                     (No. 1600~)
* SERVORELATED                                (No. 1800~)
* DI/DO                                       (No. 3000~)

>_                                        S      0 T0000
MEM  ****  ***  ***            10:12:25
[ ALARM ]  [OPERAT]  [ PARAM ]  [        ]  [        ]
```

Fig.13 (j) PARAMETER TABLE screen

7 To exit from the help screen, press the HELP key or another function key.

Explanation

- **Configuration of the Help Screen**



14 SCREEN HARDCOPY

The screen hardcopy function outputs the information displayed on the CNC screen as 640*480–dot bitmap data. This function makes it possible to produce a hard copy of a still image displayed on the CNC.
The created bitmap data can be displayed on a PC.

Screen Hardcopy Procedure

1 Check the parameter settings. To use the screen hardcopy function, set bit 7 of parameter 3301 to 1 and parameter 20 (I/O channel selection) to 4 (memory card I/F). Set other relative parameters (bits 0, 2, and 3 of parameter 3301) as needed. In a multipath system, set the parameters for each path.

2 Insert a memory card.

3 To start the function, set hardcopy start signal HDREQ (G67#7) to 1. Alternatively, hold down the **[SHIFT]** key for five seconds.

4 To end the function, press the **[CAN]** key. Alternatively, set hard copy stop signal HDABT (G67#6) to 1.

5 While the screen hardcopy operation is in progress, the hardcopy in progress signal (F061#3) is tied to 1. For several tens of seconds (or several seconds, for a monochrome LCD) before the hardcopy operation ends, the screen image stands still.

6 When the screen hardcopy operation is completed, the hardcopy in progress signal (F061#3) goes 0.

Supplementary explanation

While the screen hardcopy operation is in progress, the screen image keeps still. This means that the clock displayed on the screen indicates the beginning and end of the operation. When the clock stops counting seconds, the hard copy operation starts. The clock resumes counting seconds when the screen hard copy operation ends.

NOTE

1. During the screen hardcopy operation, key input is disabled for several tens of seconds. Until the screen hardcopy operation ends, the screen image lies still. During this period, the hardcopy in progress signal (F061#3) is tied to 1. No other signal is output. Avoid turning off the power indiscriminately during that period.
2. If the **[SHIFT]** or **[CAN]** key is customized by C executor, for instance, the screen hardcopy operation may disable the **[SHIFT]** or **[CAN]** key.
3. A normal hard copy may not be produced while the screen image is moving.

Limitations

A hard copy of the following screens cannot be produced.

1 Screen of the FS–160*i* (CNC with personal computer function)

2 System alarm screen

3 Screen while RS–232–C is being used

4 Screen during automatic or manual operation (A hard copy can be produced in a rest of the operation.)

File name

The bitmap files created by the screen hardcopy function are named as follows, in order in which they are created after power–up:

‘HDCPY000.BMP’ (Name of the first hardcopy file created after power–up)

‘HDCPY001.BMP’ (Name of the second hardcopy file created after power–up)

:

:

‘HDCPY099.BMP’

NOTE

1. A screen hardcopy file produced after the file HDCPY099. BMP is output will be named as HDCPY000.BMP.
2. If a BMP file output by the screen hardcopy function has the same name as a file present on a memory card, the file on the memory card is overwritten unconditionally.
3. When the hardcopy function is carried out after power is turned off and on again, the first output file is named as HDCPY000.BMP again. If the inserted memory card has a file having the same name, the file is unconditionally overwritten. Note this when continuously producing hard copies of various screens.

Colors of data

The number of colors used in created bitmap data depend on the display control card, the LCD hardware, and the display mode of the CNC screen. Table 14 (a) indicates the relationships.

Table 14 (a) Colors of BMP data created by the screen hardcopy function

	LCD hardware	CNC screen display mode	Colors displayed on CNC	Colors used in created BMP data	Remarks
VGA card	Monochrome LCD	—	2 colors	2 colors	Shades of gray are not supported.
	Color LCD	VGA–compatible mode	Character: 16 colors Graphic: 16 colors	When bit 0 of parameter 3301 is set to 0: 256 colors When bit 0 of parameter 3301 is set to 1: 16 colors	Most CNC screens use this mode. Note that colors may not be normally displayed in the 16–color mode.
		VGA mode	256 colors	256 colors	A special screen can be prepared by C executor, for instance.

Data size

Table 14 (b) indicates the sizes of bitmap data created by the screen hardcopy function.

Table 14 (b) Sizes of bitmap data created by the screen hardcopy function

Bitmap colors	File size (bytes)
Monochrome (2 colors)	38,462
Color (16 colors)	153,718
Color (256 colors)	308,278

Alarm message

If bit 2 of parameter 3301 is set to 1, an alarm message can be output when the hardcopy operation ends in failure.
(P/S alarm Nos. 5212 to 5214)

15 LASER FUNCTION

15.1 LASER POWER SCREEN

The following data items are displayed on the laser power screen.

- Current output power
- Actual output power
- Pulse frequency
- Pulse duty
- Error

Procedure

1 Press the OFFSET SETTING function key.

2 Press the **[POWER]** soft key.

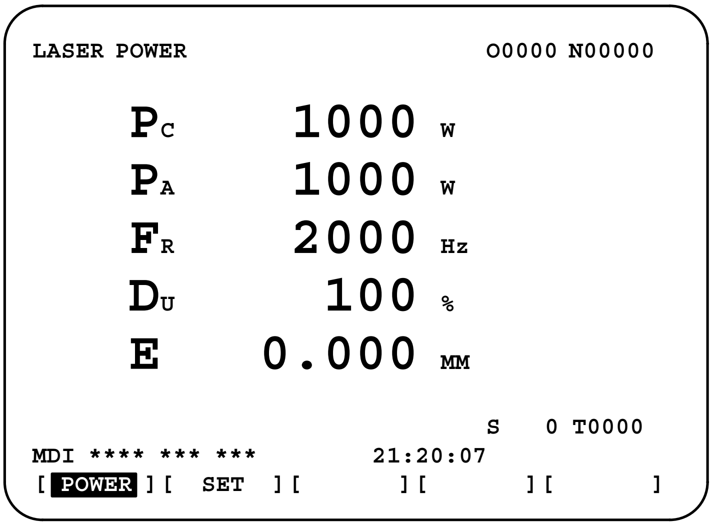


Fig.15.1

Explanations

- **Data items**

Current output power (P_C) : Displays the currently specified output power in Watt (W) . The data indicates the value before a power override is applied.

Actual output power (P_A) : Displays the actual output power in Watt (W) .

Pulse frequency (F_R) : Displays the currently specified pulse frequency in Hertz (Hz) .

Pulse duty (D_U) : Displays the currently specified pulse duty as a percentage (%) .

Error (E) : Displays the error of Z-axis gap control in mm.

- **Piercing data**

Data items, P_C , F_R , and D_U normally show contouring data. During piercing, they show piercing data.

15.2 LASER SETTING SCREEN

The data necessary to laser processing can be set on these screens.

* For operating the tracing setting screen, see the description of Z-axis gap control.

Procedure

1 Press the OFFSET SETTING function key.

2 Press the **[SET]** soft key.

3 Press the page key to select a desired screen.

4 Move the cursor to the item to be changed using the cursor keys.

5 Enter the value and press the INPUT key.

```
LASER SETTING                              O0000 N00000

CONTOURING POWER            =  1000 W
           FREQUENCY        =  2000 Hz
           DUTY             =   100 %
PIERCING   POW4R            =   800 W
           FREQUENCY        =  1000 Hz
           DUTY             =    50 %
POWER SELECT                = 0 (0:ALL,1:HLF)
POWER CONTROL               = 0 (0:OFF,1:ON)
         MIN. DUTY          =    10 %
ASSIST GAS SELECT           =     0
GAS FLOW SELECT             =     1
>_                                 S 0 T0000
MDI **** *** ***          21:12:20
[ SET ][ TRACE ][      ][ xxxxx ][       ]
```

Fig.15.2 (a)

```
LASER SETTING                              O0000 N00000

GAS FLOW 1 PRE-TIME         =  1.00 S
           PRE-PRES.        =   100
           WORK-PRES.       =   200
           AFTER-TIME       =  1.00 S
           AFTER-PRES.      =   100
GAS FLOW 2 PRE-TIME         =  1.00 S
           PRE-PRES.        =   100
           WORK-PRE         =   200
           AFTER-TIME       =  1.00 S
           AFTER-PRE        =   100

>_                                 S 0 T0000
MDI **** *** ***          21:12:59
[ POWER ][ SET ][      ][ AGING ][       ]
```

Fig.15.2 (b)

```
LASER SETTING                         O0000 N00000

GAS FLOW 3  PRE-TIME      =    1.00 S
            PRE-PRES.     =     100
            WORK-PRES.    =     200
            AFTER-TIME    =    1.00 S
            AFTER-PRES.   =     100






>_                                S    0 T0000
MDI **** *** ***          21:14:54
[ POWER ] [  SET  ] [      ] [ AGING ] [       ]
```

Fig.15.2 (c)

Explanations

- **Laser power for contouring**

Specify the laser power items for contouring (power pulse frequency and pulse duty) .

Parameter	Item at cursor	Setting unit	Setting range
Output power	(POWER)	W (Watt)	0 to 7000 W
Pulse frequency	(FREQUENCY)	Hz (Hertz)	5 to 2000 Hz
Pulse duty	(DUTY)	%	0 to 100 %

When new values for the laser power are specified in the machining mode block (G01, G02, or G03) in the part program, the program execution overwrites the old values with the new ones.

- **Laser power for piercing**

Specify the laser power items for piercing. Their respective setting ranges and units are the same as those for contouring. When new values for the laser power are specified with the piercing command (G24) in the part program, the program execution overwrites the old values with the new ones.

- **Power selection**

Specify the number of power devices (the number of effective discharge tubes) . Either the number of all tubes or half the number can be specified.

Setting 0 : All tubes are used.
1 : Half of the tubes are used.

To change the setting, turn off the HV, then turn it on. The number of effective discharge tubes is then changed.

- **Power control (on/off)**

To enable laser power control (function for controlling the pulse duty according to the feedrate) , set this parameter to 1.
When the laser power control function is enabled, laser power is controlled with a G63 P1 command or by turning on the corresponding PMC signal, PWCTL.

- **Power control (minimum duty)**

Specify the minimum duty to be clamped for laser power control. When the calculated pulse duty is less than this setting during laser power control, this setting is used.

- **Assist gas selection**

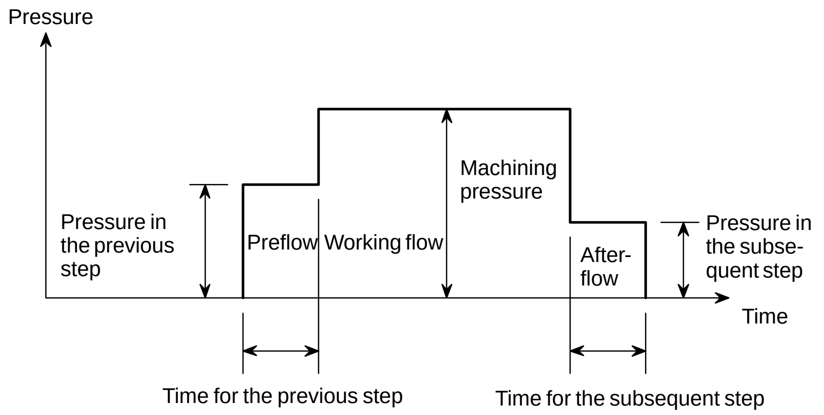
Specify the type of assist gas to be used 0 to 3 or 0 to 7 (which depends: on the setting of PRM15001#0) can be specified. When 0 is specified, no assist gas is selected. When address P is specified for the assist gas select command (G32) in the program, the program execution overwrites the old setting with the new setting.
For the correspondence between settings 1 to 7 and actual assist gas types, refer to the relevant manual of the machine tool builder.
The setting assist gas selection is set to 0 by reset.

- **Gas flow selection**

Specify the supply pattern of the assist gas flow.
1 to 3 can be specified. One of gas flows 1 to 3 is selected accordingly.

- **Gas flow 1**

Specify the supply pattern of gas flow 1. The supply patterns of gas flow corresponds to the following 5 settings.



The units and setting ranges of time and pressure are as follows.

	Setting unit	Setting range
Time	0. 01sec	0 to 9999 sec
Pres	Refer to the relevant manual of the machine tool builder.	0 to 255

- **Gas flow 2**

Specify the supply pattern of gas flow 2.
The settings are the same as those of gas flow 1.

- **Gas flow 3**

Specify the supply pattern of gas flow 3.
The settings are the same as those of gas flow 1.

Limitations

Laser setting is possible in any modes.
It is also possible while the machine operates.

The protect key switch does not work for laser setting.

15.3 POWER COMPENSATION

The power compensation function prevents the actual output power from becoming less than the specified power because of a dirty mirror or other reasons.

Procedure for power compensation

Procedure

1 Setting the parameters

No. 15000#4 : Power compensation enable/ disable
No. 15200 : Power compensation command output when half of all discharge tubes are used.
No. 15201 : Power compensation command output when all the discharge tubes are used.
No. 15203 : Power compensation limit.
No. 15204 : Power compensation coefficient
No. 15205 : Power compensation time.

Please refer the parameter sheet attached the laser oscillator.

2 Generating the laser oscillator

If the laser is discharged by setting the HV on, the power specified in parameter No. 15200 or No. 15201 becomes effective.
The function compares the specified power with the actual power and calculates a power compensation coefficient.

Explanations

- **Power compensation coefficient**

Calculating the Power compensation coefficient as follows.

Cp= (Pc/Pa) ∗1024

Cp : Power compensation coefficient
Pa : Actual Power
Pc : Command power (PRM No. 15200 or No. 15201)

Outputting laser beam with shutter close by setting power for setting time, and read the actual laser power from the power sensor when the laser is generated.

- **Commanding the power**

The commanding power to laser oscillator becomes as follows.

Pl=Cp∗Pp/ 1024

Pl : Commanding power to laser oscillator
Cp : Power compensation coefficient
Pp : Power commanded in a part program

- **Requiring mirror clean**

If the calculated power compensation coefficient exceeds the power compensation limit (PRM No. 15203), the value is clamped at the limit. When this occurs, the system displays a message indicating that the mirror needs to be cleaned and outputs the MWRN signal (F220#6) to report that it is time to clean the mirror.
The alarm can be released by resetting the system, but the MWRN signal is continued to output till the system power off.

Limitations

When the function is invalidated with the parameter, the power compensation coefficient is set 1024.

15.4 AUTOMATIC AGING FUNCTION SETTING SCREEN

The automatic aging function setting screen shows a list of settings required for the automatic aging function. On this screen, settings required for aging operation and power compensation can be referenced and changed at a time. This screen is displayed regardless of whether the automatic aging function is enabled or disabled.

Procedure

This screen can be displayed by following the procedure below.

1 Press the “OFRFSET” MDI key several times to display the laser screen.

2 Press the [SET] soft key.

3 Press the [AGING] soft key to display the aging screen.

POWER	SET	DATA	3D.TRN	
SET	TRACE	W–AXIS	AGING	

Soft key operation: Aging setting screen

Explanations

The screen consists of three pages in total. The first page and second page are related to aging operation. The third page is related to power compensation operation.

Screens

Screen display First page

```
AGING FUNCTION

[AGING SETTING 1]
   AGING (0:OFF,1:ON)          =      *
   REMAINING CYCLE             =    ***
   YEAR DATA                   =   ****
   MONTH DATA                  =   ****
   DAT DATA                    =   ****
   HOUR DATA                   =   ****

   TIME LEFT                   =   **** MIN.
   MANUAL MODE (0:OFF,1:ON)    =      *
   SET MANUAL TIMES            =    ***


[       ] [       ] [       ] [       ] [       ]
```

Corresponding parameter Nos.

- AGING (0:OFF,1:ON) → No.15008#6
- REMAINING CYCLE → No.15334
- YEAR DATA → No.15335
- MONTH DATA → No.15336
- DAT DATA → No.15337
- HOUR DATA → No.15338
- SET MANUAL TIMES → No.15340

The items displayed on the first page are listed below. The valid data range in the following applies when a value is input on this screen.

Name	Aging (0: Off, 1: On)
Meaning	Disables or enables the automatic aging function.
Parameter	No.15008#6
Valid data range	0 or 1

Name	Rest
Meaning	The rest of the CNC–determined number of times the aging operation is performed
Parameter	No.15334
Valid data range	0 to 255

Name	Year data
Meaning	Time data(NOTE)
Parameter	No.15335
Valid data range	0 to 2100

Name	Month data
Meaning	Time data(NOTE)
Parameter	No.15336
Valid data range	0 to 12

Name	Day data
Meaning	Time data(NOTE)
Parameter	No.15337
Valid data range	0 to 31

Name	Hour data
Meaning	Time data(NOTE)
Parameter	No.15338
Valid data range	0 to 23

NOTE

The time data is updated when the oscillator is in the LSTR and subsequent later stages. When LSTR is turned off, the values present at this point of time are maintained. When the automatic aging function is disabled (bit 6 of parameter No. 15008 is set to 0), however, the time data is not updated.

Name	Remaining time
Meaning	An estimated remaining time of the aging operation including the power compensation operation is indicated in minutes. Because the indicated value is an estimated value, it may differ from the actual operation time.
Parameter	—
Valid data range	This item can be referenced only. No value can be set.

Name	Manual mode (0: Off, 1: On)
Meaning	This item is set to 0 at CNC power–up. When RUN–ON is performed after this item is set to 1, the aging operation is performed as many times as the number set as the “No. of manual operations” regardless of whether the aging function is enabled or disabled and of the period in which the oscillator has been stopped. When aging is completed, and the LSTR state is entered, this item is set to 0 automatically. Even when this item is set to 1 after RUN–ON, this item does not become effective, and manual aging by this item is not started. The value set in this item is determined at the time of RUN–ON, so even when the setting of this item is changed during aging operation, the operation does not change.
Parameter	—
Valid data range	0 or 1

Name	No. of manual operations
Meaning	Set the number of times aging is to be performed when the aging operation is started manually. If bit 4 of parameter No. 15014 is set to 0, the CNC sets this item to 2. If bit 4 of parameter No. 15014 is set to 1, a value from 2 to 255 can be set.
Parameter	No.15340
Valid data range	2 to 255

Screen display Second page

```
AGING FUNCTION

[AGING SETTING 2]
   POWER                          =     * W
   FREQUENCY                      =   *** HZ
   DUTY                           =  **** %
   CAL. TIME                      =  **** S
   GAS PRESS.(50Hz)               =  ****
   GAS PRESS.(60Hz)               =  ****
   INTERVAL                       =  **** H




[       ][       ][       ][       ][       ]
```

Corresponding parameter Nos.

No.15326
No.15327
No.15328
No.15329
No.15330
No.15331
No.15339

The items displayed on the second page are listed below. The valid data range in the following applies when a value is input on this screen.

Name	**Power**
Meaning	Laser power during aging operation
Parameter	No.15326
Valid data range	0 to 9999

Name	**Frequency**
Meaning	Frequency during aging operation
Parameter	No.15327
Valid data range	0 to 2000

Name	**Duty**
Meaning	Duty cycle during aging operation
Parameter	No.15328
Valid data range	0 to 100

Name	**Compensation time**
Meaning	Power compensation time during aging operation
Parameter	No.15329
Valid data range	0 to 32767

Name	Gas pressure (50 Hz)
Meaning	Value of pressure in tubes during aging operation
Parameter	No.15330
Valid data range	0 to 32767

Name	Gas pressure (60 Hz)
Meaning	Value of pressure in tubes during aging operation
Parameter	No.15331
Valid data range	0 to 32767

Name	Stop time
Meaning	Oscillator stop time that requires aging operation
Parameter	No.15339
Valid data range	0 to 32767 (valid range: 30 to 120)

Screen display Third page

```
AGING FUNCTION

[POWER CALIBRATION]
   CALIB. (0:OFF,1:ON)        =     *
   POWER (HALF)               =     * W
   POWER (ALL)                =     * W
   FREQUENCY                  =   *** HZ
   DUTY                       =   *** %
   CAL. TIME                  =  **** S
   GAS PRESS.(50Hz)           =  ****
   GAS PRESS.(60Hz)           =  ****




[       ][       ][       ][       ][       ]
```

Corresponding parameter Nos.

- CALIB. (0:OFF,1:ON) → No.15000#4
- POWER (HALF) → No.15200
- POWER (ALL) → No.15201
- FREQUENCY → No.15321
- DUTY → No.15322
- CAL. TIME → No.15205
- GAS PRESS.(50Hz) → No.15242
- GAS PRESS.(60Hz) → No.15243

Name	Compensation (0: Off, 1: On)
Meaning	Disables or enables power compensation.
Parameter	No.15000#4
Valid data range	0 or 1

Name	Power (half)
Meaning	Specified power compensation output value when half of discharge tubes are used
Parameter	No.15200
Valid data range	0 to 9999

Name	Power (all)
Meaning	Specified power compensation output value when all discharge tubes are used
Parameter	No.15201
Valid data range	0 to 9999

Name	Frequency
Meaning	Specified frequency during power compensation
Parameter	No.15321
Valid data range	0 to 2000

Name	Duty
Meaning	Specified duty cycle during power compensation
Parameter	No.15322
Valid data range	0 to 100

Name	Compensation time
Meaning	Power compensation time setting
Parameter	No.15205
Valid data range	0 to 32767

Name	Gas pressure (50 Hz)
Meaning	Value of pressure in tubes during oscillation
Parameter	No.15242
Valid data range	0 to 32767

Name	Gas pressure (60 Hz)
Meaning	Value of pressure in tubes during oscillation
Parameter	No.15243
Valid data range	0 to 32767

Starting the aging operation manually

When the aging operation is to be started manually at the time of maintenance of the oscillator, it can be started by setting on the aging setting screen. At power–up of the CNC, the "Manual mode" item on the first page of the aging function setting screen is initially set to 0. When this item is set to 1 then RUN–ON is performed, the aging operation is performed as many times as the number set in "No. of manual operations" regardless of whether the aging function is enabled or disabled and of the period for which the oscillator has been stopped. When the LSTR state is entered after aging is completed, the "Manual mode" is set to 0 automatically. Even when this item is set to 1 after RUN–ON, this item does not become effective, and manual aging by this item is not started. The value set in this item is determined at the time of RUN–ON, so even when the setting of this item is changed during aging operation, the operation does not change.

Ending aging

As the aging operation starts, the gas pressure and laser output condition settings are automatically rewritten for aging. When the aging operation is in progress, to stop the aging operation and to execute the ordinary start sequence, follow the steps explained below. (The same steps are used for both manual aging and automatic aging.)

When the unit does not allow HV–ON (high–voltage ON) operation

1 Stop the laser oscillator (RUN–OFF) to place the oscillator in the purge state.

2 On the first page of the screen, set "Manual mode" to 0.

3 On the first page of the screen, set "Aging" to 1.

4 On the first page of the screen, set the current time in the time data items.

5 After changing the settings, restart the oscillator (RUN–ON). The parameters for specifying the gas pressure values (parameter Nos. 15242 and 15243) are changed to the values set in parameter Nos. 15324 and 15325, and the power compensation conditions (parameter Nos. 15201, 15332, 15333, and 15205) are changed to the values set in parameter Nos. 15320 to 15323 automatically.

When the unit allows HV–ON (high–voltage ON)

1 Perform HV–OFF.

2 On the first page of the screen, set the "Rest" to 0.

3 Perform HV–ON again. The parameters for specifying the gas pressure values (parameter Nos. 15242 and 15243) are changed to the values set in parameter Nos. 15324 and 15325, and the power compensation conditions (parameter Nos. 15201, 15332, 15333, and 15205) are changed to the values set in parameter Nos. 15320 to 15323 automatically.

Changing the screen display

The way of displaying the above screen items can be changed by parameter setting. If bit 4 of parameter No. 15014 is set to 0, only the first page of the above screen is displayed, allowing change of the "Manual mode" setting only. The second and third pages are not displayed.
If bit 4 of parameter No. 15014 is set to 1, all pages and items are displayed.

Parameter

	#7	#6	#5	#4	#3	#2	#1	#0
15014				MAG				

[Data type] Bit

MAG On the aging setting screen:

1 : All items on all pages are displayed. As the No. of manual operations, a value from 2 to 255 may be set.

0 : Only the first page is displayed, and the on/off setting for the manual mode may be changed. The No. of manual operations is always set to 2.

15.5 NEAR–POINT SEARCH FUNCTION

The block satisfied as follows is searched for , and the laser nozzle is positioned to the start point of the block and the machine is set in the feed hold status.

1 Where the distance Ls between the start point of the block and the current position of the laser nozzle does not exceed the distance specified in a parameter.

2 Where the distance Le between the end point of the block and the current position of the laser nozzle exceeds Ls.

Procedure for near-point search function

Procedure

1 Change the mode to MEM.

2 Find the beginning of the program.

3 Turn on the DI signal for near-point search (G226#0 NRSRH).

4 Push the start button.

5 The near-point search is worked above operation.
The near-point search busy signal (F225#1 LNSR) and cycle start lamp signal (F000#5 STL) is output in searching.

6 The cycle start lamp signal is turned off when the searching is finished.
Turn off the DI signal for near-point search and on the start button after finishing the search.
And the near-point search busy signal is turned off and the program is re-started.

Explanations

• Near-point distance

Set the near- point distance in parameter No. 15635.
The laser nozzle is positioned to the start point of the block in case that the distance between the start point and the cerrent position of the laser nozzle does not exceed the near- point distance.

Limitations

(1) The program that is to execute the near–point search function must be the one that ran by memory operation just before a near–point search was made. If a near–point search is made without running the program by memory operation, positioning cannot be performed correctly. This is because the machine position at the start of a normal operation is memorized as the reference position for a near–point search. The buffer used for this purpose is provided to memorize the machine position related to the program that performed an automatic operation (MEM, MDI, or DNC) most recently but not for individual programs. Therefore, no near–point search can be made after MDI or DNC operation.

(2) When the power was turned off and on gain, it is impossible to make a near–point search for the program executed before the power was turned off.

(3) A near–point search cannot be triggered again when it is already in progress. Once a near–point search was made and a block was found, that is, bit 0 of G226 (NRSCH) = 1, bit 1 of F225 (LNSR) = 1, and bit 5 of F000 (STL) = 0, executing a cycle start again results in alarm PS4000. Before making a re–search, effect a reset.

(4) If there is no block in which the distance Ls between the current nozzle position and block start point is shorter than the value set in parameter 15635 during a near–point search, alarm PS4000 occurs.

(5) If the current nozzle position is near the memorized near–point search reference position during a near–point search, the nozzle is moved to the reference position.

(6) No dwell command is executed during a near–point search.

(7) During a near–point search, all M and T codes are output. The PMC must handle them.

(8) No laser power is output during a near–point search.

(9) If a program to be subjected to a search contains a command related to a reference position return, include an absolute command after it. During the search, G27, G28, G29, and G30 are ignored, and the absolute command is used to re–set the coordinates.

(10) Positioning at the block found in a search is performed for all axes at a time. When positioning is completed, the nozzle keeps its attitude for the start point of the block.

(11) In a system provided with attitude control B, alarm PS4002 occurs if the G53 operation has not been set to the fixed–tip mode (bit 3 of PRM.15600 (MIA) = 0).

(12) During the G33 mode, positioning is made at the coordinates specified in the program but not at the end point of an automatically inserted arc.

(13) Do not use gap control for any axis other than the W–axis during a near–point search.
Specifying Z–axis gap control results in alarm PS4001.

(14) For programs that are to be subjected to a near–point search, it is impossible to change a work coordinate system.

(15) For programs that are to be subjected to a near–point search, all manual operations must be performed with ”manual absolute” turned on no matter whether machining has not been started or has been finished.

(16) Once a near–point search is made, manual intervention is impossible before the program is restarted.

(17) Machine coordinate system–based move commands (G27, G28, G29, G30, and G53) are ignored during a near–point search.

(18) If the end point of a block is a near point, and the block is followed by a block with G27, G28, G29, G30, or G53, the latter block is assumed to be at a near point.

15.6 GAP CONTROL AXIS SWITCHING

If the three–dimensional cutting function supports W–axis gap control, setting the gap control feed signal to “1” selects the Z–axis as a gap control axis. Activating Z–axis gap control enables the Z–axis to gap control movement along the X– and Y–axes when they are operated manually.

Procedure for gap control axis switching

Procedure

1 Turn on the gap control axis switching signal (G225#0 ZTRM).

2 Turn on the gap control start signal (G225#3 TCST).

3 Change the mode to JOG or HNDL and X and/or Y axes are moved by manual operation.

Limitations

This function cannot be executed when the manual operation in hand coordinate system.

When the gap control feed function is being executed, the Z-axis jog feed signal is ignored.

15.7 APPROACH FEED FUNCTION

The approach feed mode is ON , if the gap control feed signal is turned on when the W-axis tracing control function is added.
The signal from the gap sennsor is monitored in approach feed mode.
If the signal from the sensor is 0 or more (too close to the workpiece), the jog feed in the X, Y and Z directions and the nozzle direction is stopped.

Procedure for approach feed function

Procedure

1 Turn on the approach feed signal (G225#1 ZAPR).

2 The jog feed is stopped automatically, if the nozzle is too close to the workpiece when the nozzle is feeded in jog mode.

3 To release the machine from stopped status, set the approach feed signal to 0 and the jog feed select signal to 0, then select the approach feed function again.

15.8 MANUAL OPERATION IN HAND COORDINATE SYSTEM

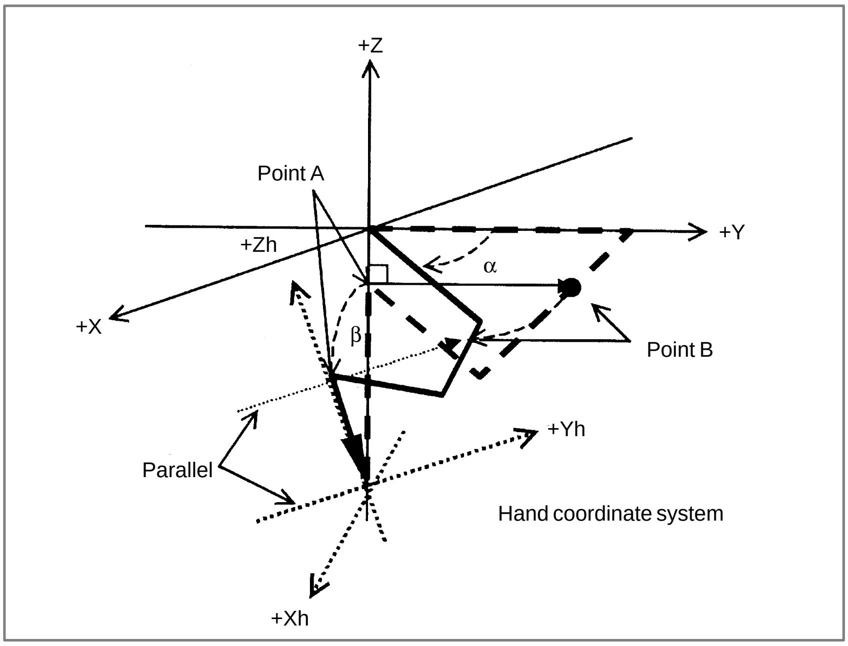
The term ”hand coordinate system” refers to a coordinate system defined based on the nozzle attitude with the nozzle tip position regarded as the origin in attitude control A or B. In a hand coordinate system defined in an arbitrary space, axis movement is implemented by moving an axis or more at a time using the feed signal for the X–, Y– or Z–axes.
Using this function makes it easy to position the nozzle during program creation by teaching.

Procedure for manual operation in hand coordinate system

Definition of the hand coordinate

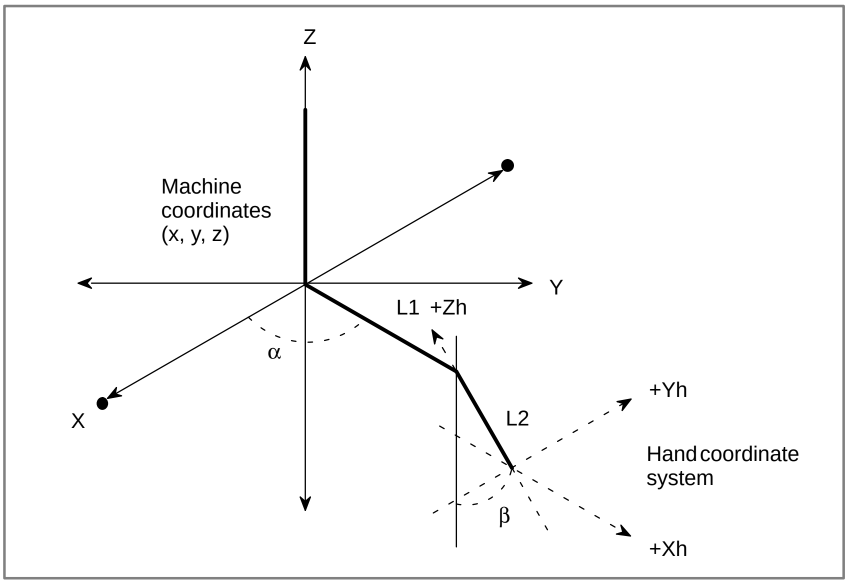
(1) Attitude control A

- Letting +Zh be a direction normal to the nozzle;
- Letting B be a point viewed from basic–attitude point A in parallel to the +Y direction, and letting +Yh be a direction parallel to line AB at the nozzle tip; and
- With the thumb and the first and second fingers of the left hand extended at right angles to one another, letting +Yh and +Zh be the directions of the first finger and thumb, respectively; let +Zh be the direction of the second finger.



(2) Attitude control B

- Letting +Zh be a direction normal to the nozzle;
- Letting +Xh be the α–axis arm direction;
- With the thumb and the first and second fingers of the left hand extended at right angles to one another, letting +Xh and +Zh be the directions of the second finger and thumb, respectively; let +Yh be the direction of the first finger.



Procedure

1 Turn on the manual operation in hand coordinate system signal (G226#3 HNDCD).

2 Jog feed
Each axis moves in the ± Xh-axis direction using jog feed axis direction signal ± X. They also move in the ± Yh-axis direction using a ± Y signal and the ± Zh-axis direction using a ± Z signal.
For the jog feed in the hand coordinate system, one-axis operation (including α and β axes) is performed at the same time. When jog feed axis direction signal ± α and ± β are specified, the rotarion axis operates in the nozzle tip fixed mode. If execution of the rotation axis command stops, the hand coordinate system is updated to a new coordinate system.
A linear axis command is executed in the updated hand coordinate system.

3 Handle feed
Each axis can be moved in the ± Xh-axis direction using a manual pulse generator when manual handle feed axis select signal HX turned on. They also move in the ± Yh-axis direction using ± HY signal and the ± Zh-axis direction using a ± HZ signal.

Limitations

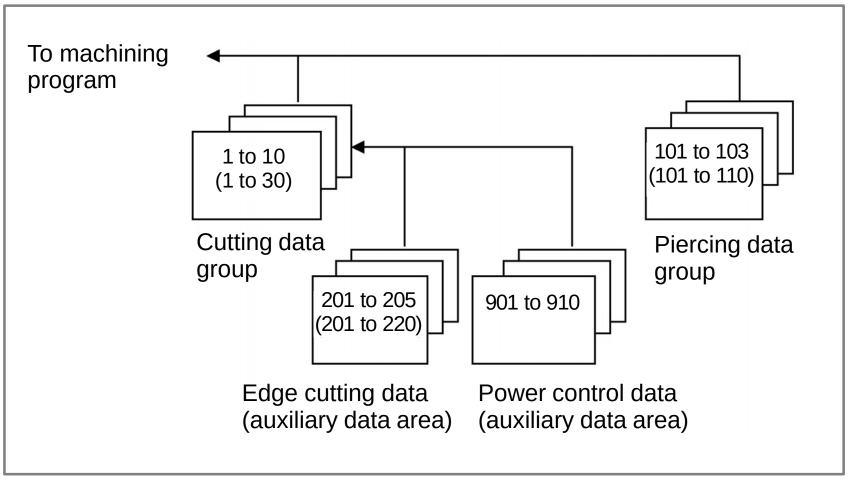
H4 (4th axis: α axis) and H5 (5th axis: β axis) handle feeds cannot be accepted when hand coordinate mode select signal HNDCD is set to 1.
To move the α and β axes by the handle feed, set the HNDCD signal to 0. When manual handle feed axis select signal H4 or H5 is selected, set a PMC ladder so that the HNDCD signal is “0”. This facilitates the operation.
Set all the X–, Y–, and Z–axes to the same manual feedrate.

15.9 CUTTING CONDITION SETTING FUNCTION

Overview

The sets of cutting data of the relevant items are registered, under the title of the proper number in the corresponding data area, which can be called and used for cutting when commanded by that number in a part program. This data area has an enough capacity for holding data required for cutting one type of workpiece (material and plate thickness), so conditions required for cutting can be set just by specifying a data number. High–speed access to this area is possible, so data can be modified manually and easily even during program execution. Furthermore, modified data can be registered, so cutting conditions can be updated easily.
When the option for additional cutting condition registration is specified, the specifications of the conventional cutting condition setting function can be expanded.

- **Structure of the cutting condition data area**



1) Cutting data items make up a set, sets make up a group, and groups further make up a cutting condition data area.
2) The cutting condition data area consists of three types of groups including piercing data groups, cutting data groups, and auxiliary data groups.
3) Piercing data groups and cutting data groups can be specified from a program by adding a set number to address E.

   a. Without the option for additional cutting condition registration

E×××;

The number of cutting data set (1 to 10)
The number of piercing data set (101 to 103)

b. With the option for additional cutting condition registration

E×××;
The number of cutting data set (1 to 30)
The number of piercing data set (101 to 110)

4) Auxiliary data groups are specified from data items in cutting data groups. It is impossible to specify auxiliary data groups directly with address E.

Items of cutting condition data groups

• Piercing data group

Piercing data consists of the items listed below.
For the operation of piercing, see Section 18.2, “PIERCING FUNCTION”, in Part II, “PROGRAMMING”.

1) Setting items

Data item	Setting range	Unit
Peak power	0 to 9999	W
Initial frequency	5 to 2000	Hz
Initial duty	0 to 100	%
Step frequency	0 to 2000	Hz
Step duty	0 to 99	%
Step time	0 to 9.999	Sec
Step number	0 to 99	
Piercing time	0.001 to 999.999	Sec
Assist gas pressure	0 to 9.99	MPa
Assist gas select	1 to 7	
Gas settling time	0 to 9.9	Sec
Reference displace	–9.999 to 9.999	mm

2) Size
A group consists of three sets, each consisting of the above items, when the option for additional cutting condition registration is not provided, or a group consists of ten sets when the option for additional cutting condition registration is provided.

3) Calling method
If a piercing data set number is specified with address E in a program, the data set with the specified number is read as active data.

E×××;
The number of piercing data set (101 to 103(*1), 101 to 110(*2))

*1 Without the option for additional cutting condition registration
*2 With the option for additional cutting condition registration

- **Cutting data group**

Cutting data consists of the items listed below.
For the meaning of cutting data, see Section 18.1, "CONTOUR MACHINING", in Part II, "PROGRAMMING".

1) Setting items

Data item	Setting range		Unit
	Without additional cutting condition registration	With additional cutting condition registration	
Feedrate	0 to 99999	0 to 99999	mm/min
Cutting peak power	0 to 9999	0 to 9999	W
Cutting frequency	5 to 2000	5 to 2000	Hz
Cutting duty	0 to 100	0 to 100	%
Assist gas pressure	0 to 9.99	0 to 9.99	MPa
Assist gas select	1 to 7	1 to 7	
Gas settling time	0 to 9.9	0 to 9.9	Sec
Reference displace	–9.999 to 9.999	–9.999 to 9.999	mm
Cutter compensation	–9.999 to 9.999	–9.999 to 9.999	mm
Edge cutting select	0/201 to 205	0/201 to 215	Aux. data No.
Startup select	0/201 to 205	0/201 to 215	Aux. data No.
Power control select(*3)	0/901 to 910	0/901 to 910	Aux. data No.

*3 Supported by the FS16*i*–LB and later

2) Size
A group consists of 10 sets, each consisting of the above items, when the option for additional cutting condition registration is not provided, or a group consists of 30 sets when the option for additional cutting condition registration is provided.

3) Calling method
If a cutting data set number is specified with address E in a program, the data set with the specified number is read as active data and used for control when the cutting feed command is executed.

E××× ;

The number of cutting data set (1 to 10(*4), 1 to 30(*5))

*4 Without the option for additional cutting condition registration
*5 With the option for additional cutting condition registration

- **Auxiliary data group**

Edge cutting data groups and power control data groups are provided as auxiliary data groups.

1. Edge cutting data group

Edge cutting data is used for edge cutting, which is performed to cut a corner sharply, and for using the startup function, which operates when contouring is started after piercing. When used, edge cutting data is called from "Edge cutting select" or "Startup select" of a cutting data group. For the operations of edge cutting and startup, see Subsection 18.7.3, "Edge Machining Function", and Subsection 18.7.4, "Startup Machining Function", in Part II, "PROGRAMMING".

1) Setting items

Data item	Setting range	Unit
Judge angle for edge	0 to 180	Degree
Piercing power	0 to 9999	W
Piercing frequency	5 to 2000	Hz
Piercing duty	0 to 100	%
Piercing time	0 to 999.999	Sec
Piercing gas press	0 to 9.99	MPa
Piercing gas select	0 to 99	
Recovery distance	0 to 99.999	mm
Recovery feedrate	0 to 9999	mm/min
Recovery frequency	5 to 2000	Hz
Recovery duty	0 to 100	%

2) Size
A group consists of five sets, each consisting of the above items, when the option for additional cutting condition registration is not provided, or a group consists of 20 sets when the option for additional cutting condition registration is provided.

3) Calling method
Data is called indirectly.
In "Edge cutting select" or "Startup select" of cutting data groups, set edge cutting data numbers from 201 to 205 in advance if the option for additional cutting condition registration is not provided, or set edge cutting data numbers from 201 to 220 in advance if the option for additional cutting condition registration is provided. Then, when cutting data is read, edge cutting data is called at the same time.

(1) Specified peak power

[Data type] Word

[Unit of data] W

[Valid data range] 0 to 7000

Set a peak power to be specified for each data table.

(2) Specified frequency

[Data type] Word

[Unit of data] Hz

[Valid data range] 0 to 2000

Set a frequency to be specified for each data table.

(3) Specified duty cycle

[Data type] Word

[Unit of data] %

[Valid data range] 0 to 100

Set a duty cycle to be specified for each data table.

2. Power control data group

Power control data consists of the items listed below.
For the operation of power control, see Section 18.3, "LASER POWER CONTROL", in Part II, "PROGRAMMING".

1) Setting items

Data item	Setting range	Unit
Minimum peak power (clamp value)	0 to 9999	W
Peak power at feedrate (F) of 0	0 to 9999	W
Minimum pulse frequency (clamp value)	5 to 2000	Hz
Pulse frequency at feedrate (F) of 0	5 to 2000	Hz
Minimum pulse duty cycle (clamp value)	0 to 100	%
Duty cycle at feedrate (F) of 0	0 to 100	%
Allowable feedrate variation width (dead–band)	0 to 32767	mm/min or %

2) Size
A group consists of 10 sets, each consisting of the above items.

3) Calling method
Data is called indirectly.
In "Power control select" of cutting data groups, set power control condition numbers from 901 to 210 in advance. Then, when cutting data is read, power control data is called at the same time.

Displaying cutting condition data area

1) Displaying each cutting data group registration screen
   a. Press the OFFSET/SETTING function key several times. Then, the following soft keys are displayed:

POWER	SET	DATA	3D.TRN	

   b. Press the [DATA] soft key. Then, the following soft keys are displayed:

CUT	PIERCE	EDGE	PWR.CT	OPRT

   c. Press the soft key of a desired cutting data group. Then the data group is displayed.

Cutting data group	>	[CUT]
Piercing data group	>	[PIERCE]
Edge data group	>	[EDGE]
Power control data group	>	[POWER C]

2) Indication of the name of displayed data
   The name of the displayed data is indicated at the beginning of the screen.

   Example:xxxxx

3) Indication of the active data number
   Following the name of the displayed data, the numbers of the cutting data set and piercing data set that are currently active are indicated.

   Example:xxxxx

   To clear the E number by reset operation, set bit 6 of parameter No. 15004 to 1.

Cutting data screen

```
[CUTTING]      ACTIVE DATA NO. CUTTING = 5     PIERCING = 103

NO.   FEED     PWR   FREQ     DUTY     GAS    GAS   GAS   GAP   OFSET     EDGE START    POWER
                                       PRESS.KIND  TIME                        UP       CTL

 1    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
 2    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
 3    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
 4    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
 5    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***

 6    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
 7    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
 8    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
 9    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
10    ***.**   ****  ****     ***      **.*    *    *.*  **.****  ***.****  *** ***      ***
```

Piercing data screen

```
[PIERCING]     ACTIVE DATA NO. CUTTING = 5     PIERCING = 103

No.  PWR.    [INITIAL]          [INCREM.]        [STEP]     PIRC [ASSIST GAS]   DEF
          FREQ.      DUTY    FREQ.      DUTY  TIME   CNT  TIME PRESS. KIND TIME

101  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
102  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
103  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
104  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
105  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***

106  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
107  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
108  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
109  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
110  **** ****       ***     ****       ***   *.***  ** ***.***  **.* **   *.* -*.***
```

Edge cutting data screen

```
[EDGE]     ACTIVE DATA NO. CUTTING = 5     PIERCING = 103

No. ANGL. PWR  FREQ    DUTY    PIRC  GAS    GAS  STRT STRT STRT    STRT     DEF
                               TIME  PRESS. KIND DIST FEED FREQ.   DUTY

201  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
202  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
203  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
204  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
205  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***

206  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
207  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
208  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
209  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
210  ***  **** ****    ***    ***.***  **.* **  **.*** **** ****    ****    -*.***
```

Power control data screen

```
[POWER CONTROL]     ACTIVE DATA NO. CUTTING = 5     PIERCING = 103

  No.        POWER          FREQUENCY          DUTY      PWR./SPEED VAR.
          MIN     F=0      MIN     F=0      MIN     F=0

  901     ****    ****     ****    ****     ***     ***        *****
  902     ****    ****     ****    ****     ***     ***        *****
  903     ****    ****     ****    ****     ***     ***        *****
  904     ****    ****     ****    ****     ***     ***        *****
  905     ****    ****     ****    ****     ***     ***        *****

  906     ****    ****     ****    ****     ***     ***        *****
  907     ****    ****     ****    ****     ***     ***        *****
  908     ****    ****     ****    ****     ***     ***        *****
  909     ****    ****     ****    ****     ***     ***        *****
  910     ****    ****     ****    ****     ***     ***        *****
```

Modifying cutting condition data

1) Function for changing cutting condition data by screen operations
Select and display a screen, and press the [OPRT] soft key. Then, modification becomes possible. Move the cursor to the item to change, and modify data.
   a. When data with a currently active number is modified, the active data of the items except the "Cutter compensation" item is also modified immediately.
   b. The new "Cutter compensation" data after the modification becomes valid when the block at which the buffer content is rewritten has been executed.
2) Function for modifying cutting data by using an external signal
When modification data is transferred from the PMC to the CNC, data of a specified item of cutting data with the currently active set number can be increased and decreased.
Registered data of piercing data groups, power control data groups, and auxiliary data groups cannot be changed.
This function cannot be used when the option for additional cutting condition registration is specified.
   a. Specifying an R area
   Three bytes in the R area of the PMC are allocated to modification data. When the start address of the R area is set in parameter No. 15360, a 3–byte area starting from the start address is allocated.
   b. Modification data
   The three bytes in the R area which are allocated by parameter setting are assigned to the item to change, change amount, and change magnification as shown below. The CNC reads the three bytes of data at intervals of 16 ms, and modifies cutting data.
   The cutting data to change is the data of an item specified by the item to change in the cutting data with the set number selected by the E code. The data is increased or decreased by the value obtained by multiplying the change amount by the change magnification.
   The following data can be modified:
   – Feedrate
   – Peak power
   – Pulse frequency
   – Pulse duty cycle
   – Gap control reference displacement
   – Assist gas pressure

Start address	Item to change

[Data type] Byte

[Valid data range] 1 to 255

Item to change	Data
Feedrate	1
Peak power	2
Pulse frequency	3
Pulse duty cycle	4
Gap control reference displacement	5
Assist gas pressure	6

Start address + 1	Change amount

[Data type] Byte

[Valid data range] ±127

[Unit of data] See the following.

Item to change	Data
Feedrate	1 mm/min
Peak power	1 W
Pulse frequency	1 Hz
Pulse duty cycle	1 %
Gap control reference displacement	0.001 mm
Assist gas pressure	0.01 MPa

Start address + 2	Change magnification

[Data type] Byte

[Valid data range] 1 to 255

c. Transmission and reception of cutting data
The CNC reads the change amount data at intervals of 16 ms. If the read data is not 0, the CNC successively reads the item to change and the change magnification data, then writes 0 to the change amount to complete data reception. So, after checking that the change amount data is 0, the PMC sets data of the item to change and the change magnification (if the data is the same as the previous data, the setting can be omitted), then writes the change amount at the end.

Override

An override can be applied to the cutting condition data listed below by using an external signal.
For details of override, refer to “FS16i Connection Manual (Function)” (B–63523EN–1) and “FS16i–LB Connection Manual” (B–63663EN).

– Feedrate
– Peak power
– Pulse frequency
– Pulse duty cycle
– Assist gas pressure

Expansion of cutting condition data

The data area of the cutting condition setting function allows high–speed access but has a small capacity.
To use many types of workpiece materials, it is necessary to prepare a separate cutting condition data file and transfer necessary data from the file to the cutting condition data area. With the expansion nonvolatile memory controllable by the PMC and the enhanced CNC–PMC window function, application software that uses the expansion nonvolatile memory as a cutting condition data file can be developed by using C of the PMC.

- **Transmission and reception of cutting condition data**

1) Data transfer between the cutting condition data area and expansion nonvolatile memory
The PMC–CNC window function allows data transmission and reception between the CNC data area and the expansion nonvolatile memory.
This can be used for functions that use an M code for comment reading mentioned above to specify necessary data for cutting, transfer the data from the expansion nonvolatile memory, in which all cutting condition data has been registered in advance, to the data area, and perform operation while making high–speed access to the transferred data.
2) Data transfer between the cutting condition data area and macro executor
System variables are assigned to data items so that data can be read from and written in the cutting condition data area by using conversational macros and execution macros.
This function cannot be used when the option for additional cutting condition registration is specified.

a. Piercing data

Data item	E101	E102	E103
Peak power	#6500	#6515	#6530
Initial frequency	#6501	#6516	#6531
Initial duty	#6502	#6517	#6532
Step frequency	#6503	#6518	#6533
Step duty	#6504	#6519	#6534
Step time	#6505	#6520	#6535
Step number	#6506	#6521	#6536
Piercing time	#6507	#6522	#6537
Assist gas pressure	#6508	#6523	#6538
Assist gas type	#6509	#6524	#6539
Gas settling time	#6510	#6525	#6540
Reference displace	#6511	#6526	#6541

b. Cutting data

Data item	E1	E2	E3	E4	E5
Feedrate	#6300	#6315	#6330	#6345	#6360
Cutting peak power	#6301	#6316	#6331	#6346	#6361
Cutting frequency	#6302	#6317	#6332	#6347	#6362
Cutting duty	#6303	#6318	#6333	#6348	#6363
Assist gas pressure	#6304	#6319	#6334	#6349	#6364
Assist gas select	#6305	#6320	#6335	#6350	#6365
Gas settling time	#6306	#6321	#6336	#6351	#6366
Reference displace	#6307	#6322	#6337	#6352	#6367
Cutter compensation	#6308	#6323	#6338	#6353	#6368
Edge cutting select	#6309	#6324	#6339	#6354	#6369
Startup select	#6310	#6325	#6340	#6355	#6370
Power control	#6311	#6326	#6341	#6356	#6371

Data item	E6	E7	E8	E9	E10
Feedrate	#6375	#6390	#6405	#6420	#6435
Cutting peak power	#6376	#6391	#6406	#6421	#6436
Cutting frequency	#6377	#6392	#6407	#6422	#6437
Cutting duty	#6378	#6393	#6408	#6423	#6438
Assist gas pressure	#6379	#6394	#6409	#6424	#6439
Assist gas select	#6380	#6395	#6410	#6425	#6440
Gas settling time	#6381	#6396	#6411	#6426	#6441
Reference displace	#6382	#6397	#6412	#6427	#6442
Cutter compensation	#6383	#6398	#6413	#6428	#6443
Edge cutting select	#6384	#6399	#6414	#6429	#6444
Startup select	#6385	#6400	#6415	#6430	#6445
Power control	#6386	#6401	#6416	#6431	#6446

c. Edge data

Data item	E201	E202	E203	E204	E205
Judge angle for edge	#6700	#6715	#6730	#6745	#6760
Piercing power	#6701	#6716	#6731	#6746	#6761
Piercing frequency	#6702	#6717	#6732	#6747	#6762
Piercing duty	#6703	#6718	#6733	#6748	#6763
Piercing time	#6704	#6719	#6734	#6749	#6764
Piercing gas press	#6705	#6720	#6735	#6750	#6765
Piercing gas select	#6706	#6721	#6736	#6751	#6766
Recovery distance	#6707	#6722	#6737	#6752	#6767
Recovery feedrate	#6708	#6723	#6738	#6753	#6768
Recovery frequency	#6709	#6724	#6739	#6754	#6769
Recovery duty	#6710	#6725	#6740	#6755	#6770
Tracing displacement	#6711	#6726	#6741	#6756	#6771

d. Active E code

Data item	—
Piercing data set No.	#6030
Cutting data set No.	#6031

3) Power control data operation by custom macros
System variables are assigned to data items to allow custom macros to read and write power control data.
These system variables can be referenced by the macro executor. In this case, the number used for reference is obtained by adding 100000 to the corresponding system variable used by custom macros.
For example, data a custom macro references by using #6550 can be referenced by the macro executor by using #106550.

Data item	E901	E902	E903	E904	E905	E906	E907	E908	E909	E910
Minimum power	#6550	#6560	#6570	#6580	#6590	#6600	#6610	#6620	#6630	#6640
Power at feedrate of 0	#6551	#6561	#6571	#6581	#6591	#6601	#6611	#6621	#6631	#6641
Minimum frequency	#6552	#6562	#6572	#6582	#6592	#6602	#6612	#6622	#6632	#6642
Frequency at feedrate of 0	#6553	#6563	#6573	#6583	#6593	#6603	#6613	#6623	#6633	#6643
inimum duty cycle	#6554	#6564	#6574	#6584	#6594	#6604	#6614	#6624	#6634	#6644
Duty cycle at feedrate of 0	#6555	#6565	#6575	#6585	#6595	#6605	#6615	#6625	#6635	#6645
Allowable feedrate variation width	#6556	#6566	#6576	#6586	#6596	#6606	#6616	#6626	#6636	#6646

4 Data transfer using FOCAS1
With the FOCAS1 (FANUC Open CNC API Specifications version 1) window function, cutting condition data can be transferred between a personal computer, the CNC, and the PMC via Ethernet (TCP/IP) or HSSB (High Speed Serial Bus).
This allows a large amount of cutting condition data to be held in the personal computer and used.
This function cannot be used when the option for additional cutting condition registration is specified.
For details on how to use the function, refer to the FOCAS1 data window library specifications.

Functions

Data item	Without additional cutting condition registration	With additional cutting condition registration
Number of cutting data sets	10	30
Number of piercing data sets	3	10
Number of edge cutting data sets (auxiliary data)	5	20
Number of power control data sets (auxiliary data)	10	10
Function for modifying cutting data by using an external signal	○	×
Comment read	○	○
Data transfer between cutting condition data area and expansion nonvolatile memory	○	○
Data transfer between cutting condition data area and macro executor	○	×
Data transfer using FOCAS1	○	×

○ : Available × : Not available

15.10 LASER STATUS SCREEN

The current data of laser are displayed.

Procedure

1 Press OFFSET SETTING key, and the following soft keys are displayed.

POWER	SET	DATA		

2 Press next- menu key ▷.

3 Press **[STATUS]** key.

4 Laser status screen is displayed as follows.

```
MAT.  ***********                         O *****     N *****
E CODE CUTTING ***    PIERCING ***

F FEED              *****   mm/min        FA FEED       ***** mm/min
PC POWER             ****   W             PA POWER       **** W
FR FREQUENCY         ****   Hz
DU DUTY               ***   %             X -*****.***
GK GAS SELECT           *                 Y -*****.***
GP GAS PRESSURE      *.**   MPA           Z -*****.***
ER DEFLECTION     -*.***    mm            4 -*****.***
D OFFSET           *.***    mm

                                  STATUS
```

Explanations

- **Meaning of items**

E–code	: The current data set number for cutting and piercing of processing condition setting function is shown.
Feedrate (F)	: The commanded feedrate is shown. Unit mm/ min
Command power (P_C)	: The commanded laser peak power before applying override is shown. Unit Watt (W)
Pulse frequency (F_R)	: The commanded pulse frequency is shown. Unit Hertz (Hz)
Pulse duty (D_U)	: The commanded pulse duty is shown. Unit %
Assist gas select (Gk)	: The commanded assist gas select is shown.
Assist gas pressure (Gp)	: The actual assist gas pressure is shown. Unit MPa
Error of gap control (E)	: Error of gap control is shown. Unit mm
Offset value (D)	: The current tool offset value is shown. Unit mm
Program No. (O)	: The current program No. is shown.
Sequence No. (N)	: The current sequence No. in a part program is shown.
Actual feedrate (F_A)	: Actual feedrate is shown. Unit mm/ min
Actual power (P_A)	: Actual laser power is shown. Unit Watt (W)
Position	: The current position in the relative coordinate is shown.

15.11 THREE–VARIABLE TRANSFORM FUNCTION

When the target point and the basic point is specified, an arbitrary point in the coordinate system of a part program is converted into the part program by three dimensions. The mirror image conversion and three dimension movement conversion are prepared as three-variable transform function.

Procedure

1 When the function key OFFSET SETTING is pushed, the softkey is displayed as follows.

POWER	SET	DATA	3D.TRN	

2 Push the soft key [3D. TRN].

3 The mirror image source screen is displayed as follows.

```
 MIRROR IMAGE SOURCE                O0001 N0001

       SOURCE (P)      DESTINA. (Q)
      X-99999.999    X-99999.999
#0    Y-99999.999    Y-99999.999
      Z-99999.999    Z-99999.999

[ POWER ][  SET  ][ DATA ][ 3D.TRN ][        ]
```

```
 SHIFT FUNC.                        O0001 N0001

       SOURCE (P)      DESTINA. (Q)
      X-99999.999    X-99999.999
#1    Y-99999.999    Y-99999.999
      Z-99999.999    Z-99999.999

      X-99999.999    X-99999.999
#2    Y-99999.999    Y-99999.999
      Z-99999.999    Z-99999.999

#3    Y-99999.999    Y-99999.999
      Z-99999.999    Z-99999.999

[ POWER ][  SET  ][ DATA ][ 3D.TRN ][        ]
```

4 The screen consists of the mirror image source conversion and the three dimension movement conversion. It is 2 page, and change the page by the page key.

5 Move the cursor on the item where data must be set, and push the function key INPUT after inputting value.

Limitations

It is impossoble to set these data during operating a part program.

IV. MAINTENANCE

1 METHOD OF REPLACING BATTERY

This chapter describes how to replace the CNC backup battery and absolute pulse coder battery. This chapter consists of the following sections:

Battery for memory backup

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. Therefore, the above data is not lost even if the main battery fails. The backup battery is installed in the control unit prior to being shipped from the factory. This battery can provide backup for the memory contents for about a year.

When the battery voltage falls, alarm message “BAT” blinks on the LCD 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 one or two weeks of the alarm first being issued. This, however, depends on the system configuration.

If the battery voltage subsequently drops further, backup of memory can no longer be provided. Turning on the power to the control unit in this state causes system alarm 910 (SRAM parity alarm) to be issued because the contents of memory are lost. Replace the battery, clear the entire memory, then reenter the data.
Replace the memory backup battery within a few minutes while the control unit is brought off.
The following two kinds of batteries can be used.

- Lithium battery, incorporated into the CNC control unit.
- Two alkaline dry cells (size D) in an external battery case.

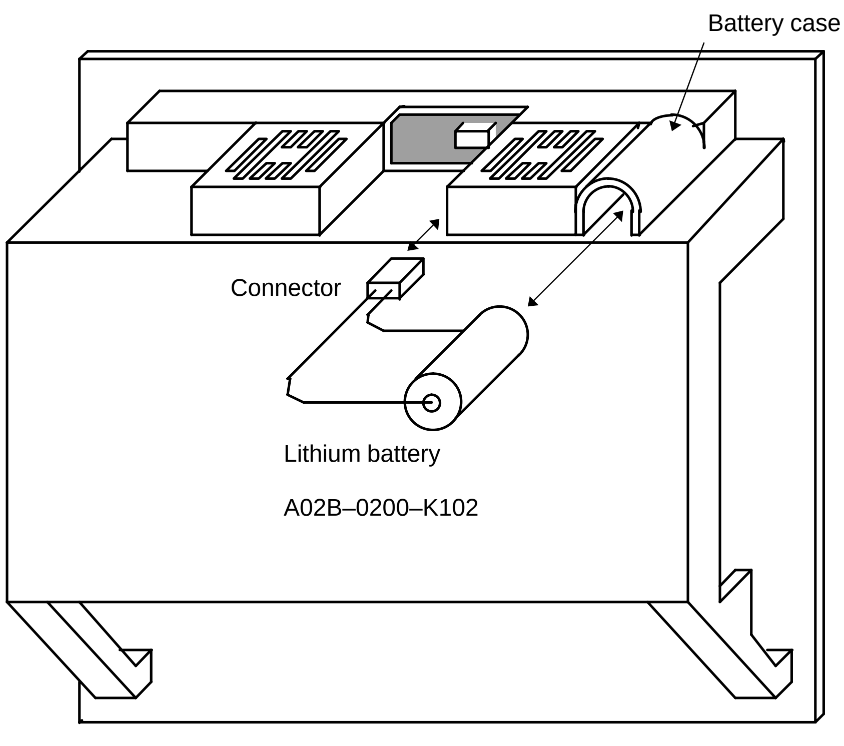
> **NOTE**
> A lithium battery is installed as standard at the factory.

1.1 REPLACING BATTERY FOR LCD–MOUNTED TYPE *i* SERIES

- **Replacement procedure**

When a lithium battery is used
Prepare a new lithium battery (ordering code: A02B–0200–K102 (FANUC specification: A98L–0031–0012)).

1) Turn on the power to the CNC. After about 30 seconds, turn off the power.
2) Remove the old battery from the top of the CNC control unit.
   First, unplug the battery connector, then take the battery out of its case.
   The battery case of a control unit without option slots is located at the top end of the unit as shown in the figure of the previous page. The battery case of a control unit with 2 slots or 4 slots is located in the central area of the top of the unit (between fans).
3) Insert a new battery and reconnect the connector.



WARNING
Using other than the recommended battery may result in the battery exploding. Replace the battery only with the specified battery (A02B–0200–K102).

CAUTION

Steps 1) to 3) should be completed within 30 minutes (or within 5 minutes for the 160*i*/180*i* with the PC function). Do not leave the control unit without a battery for any longer than the specified period. Otherwise, the contents of memory may be lost.

If steps 1) to 3) may not be completed within 30 minutes, save all contents of the CMOS memory to the memory card beforehand. Thus, if the contents of the CMOS memory are lost, the contents can be restored easily.

For the method of operation, refer to Maintenance manual (B–63005EN).

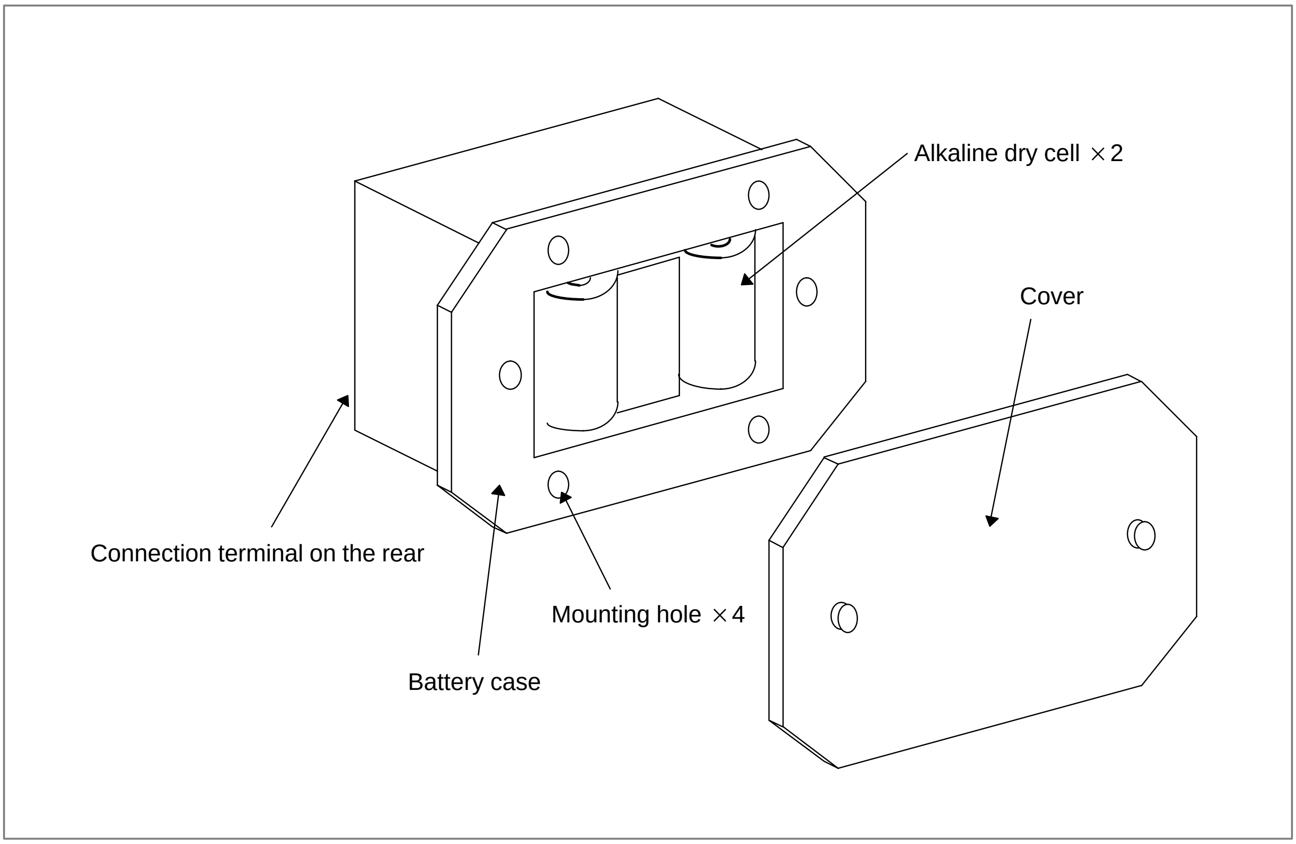
When discarding a battery, observe the applicable ordinances or other rules of your local government. Also, cover the terminals of the battery with vinyl tape or the like to prevent a short–circuit.

Replacing commercial alkaline dry cells (size D)

1) Prepare two alkaline dry cells (size D) commercially available.
2) Turn on the power to the Series 16*i*/18*i*/160*i*/180*i*.
3) Remove the battery case cover.
4) Replace the cells, paying careful attention to their orientation.
5) Reinstall the cover onto the battery case.

CAUTION

When replacing the alkaline dry cells while the power is off, use the same procedure as that for lithium battery replacement described above.



1.2 REPLACING THE BATTERY FOR STAND–ALONE TYPE *i* SERIES

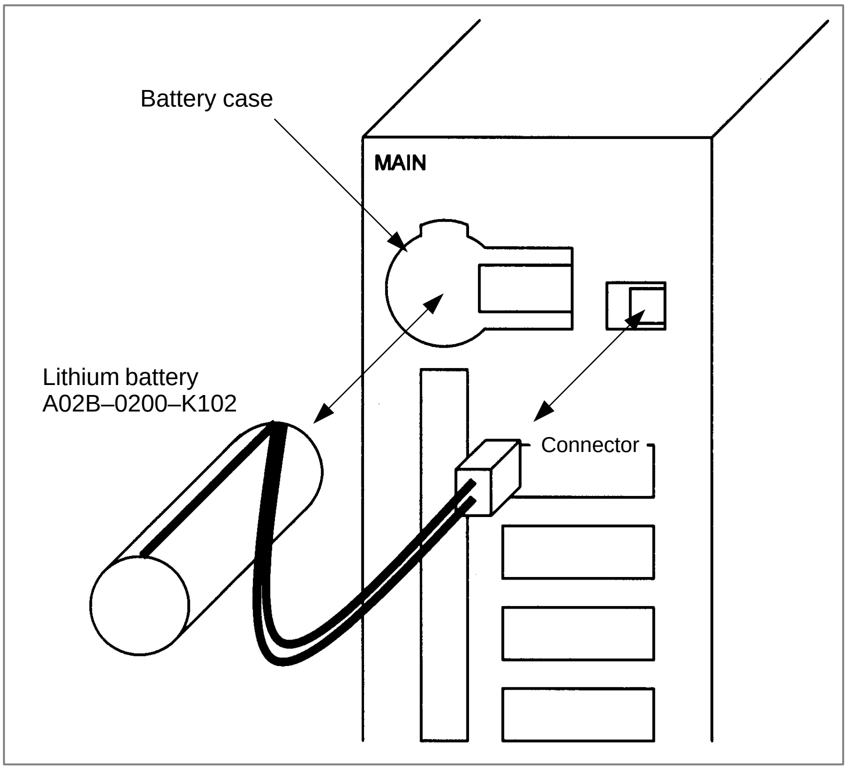
- **Replacing the battery**

If a lithium battery is used, have A02B–0200–K102 (FANUC internal code: A98L–0031–0012) handy.

(1) Turn the CNC on. About 30 seconds later, turn the CNC off.

(2) Remove the battery from the top area of the CNC unit.
Disconnect the connector first. Then, remove the battery from the battery case.
The battery case is provided in the top area of the face plate of the main CPU board.

(3) Replace the battery, then connect the connector.



WARNING
The incorrect mounting of the battery may cause an explosion. Avoid using any battery other than the one specified here (A02B–0200–K102).

> **NOTE**
> Complete steps (1) to (3) within 30 minutes.
> If the battery is left removed for a long time, the memory would lose the contents.
> If there is a danger that the replacement cannot be completed within 30 minutes, save the whole contents of the CMOS memory to a memory card. The contents of the memory can be easily restored with the memory card in case the memory loses the contents.

Discard the dead battery, observing appropriate municipal rules and regulations. When discarding the battery, insulate the terminal with a tape so that no short–circuit would occur.

When using commercial D–size alkaline dry cells

• Replacing the battery

(1) Have commercial D–size alkaline dry cells handy.

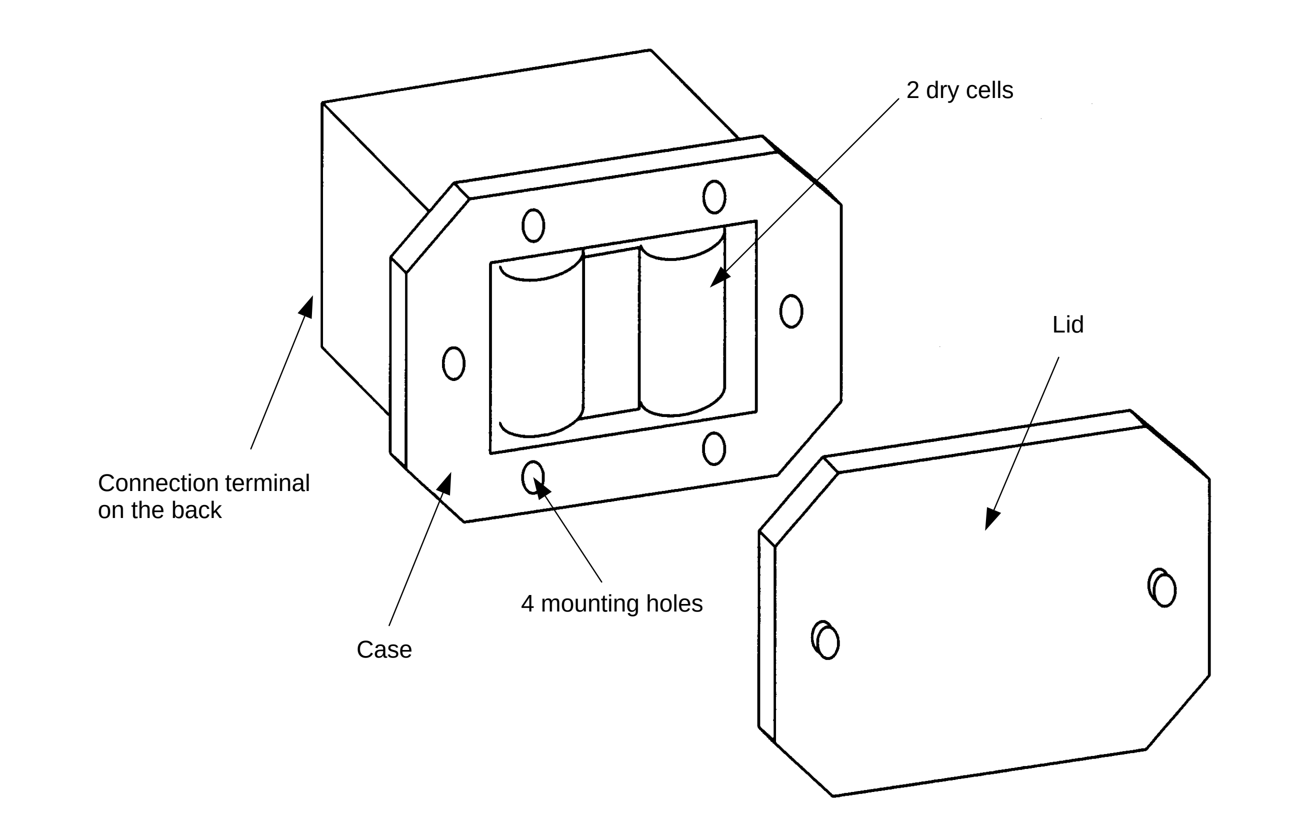
(2) Turn the CNC on.

(3) Remove the lid from the battery case.

(4) Replace the old dry cells with new ones. Mount the dry cells in a correct orientation.

(5) Replace the lid on the battery case.

> **NOTE**
> In the power–off state, the battery should be replaced as in the case of the lithium battery, which is descried above.


2 dry cells
Lid
Connection terminal
on the back
Case
4 mounting holes

1.3 BATTERY IN THE CNC DISPLAY UNIT WITH PC FUNCTIONS (3 VDC)

A lithium battery is used to back up BIOS data in the CNC display unit with PC functions. This battery is factory–set in the CNC display unit with PC functions. This battery has sufficient capacity to retain BIOS data for one year.

When the battery voltage becomes low, the LCD screen blinks. (The LCD screen also blinks if a fan alarm is issued.) If the screen blinks, replace the battery as soon as possible (within one week). FANUC recommends that the battery be replaced once per year regardless of whether a battery alarm is issued.

Replacing the battery

(1) To guard against the possible loss or destruction of BIOS parameters, write down the BIOS parameter values.

(2) Obtain a new lithium battery (A02B–0200–K102).

(3) After power has been supplied for at least five seconds, turn off the power to CNC display unit with PC functions. Remove the intelligent terminal from the panel so that replacement work can be done from the rear of the intelligent terminal.

(4) Detach the connector of the lithium battery, and remove the battery from the battery holder.

(5) Run the cable for the new lithium battery as shown in the Fig. 1.3.

(6) Attach the connector, and place the battery in the battery holder.

(7) Install CNC display unit with PC functions again.

(8) Turn on the power, and check that the BIOS parameters are maintained (BIOS setup is not activated forcibly).

Between removing an old battery and inserting new battery, no more than five minutes must be allowed to elapse.

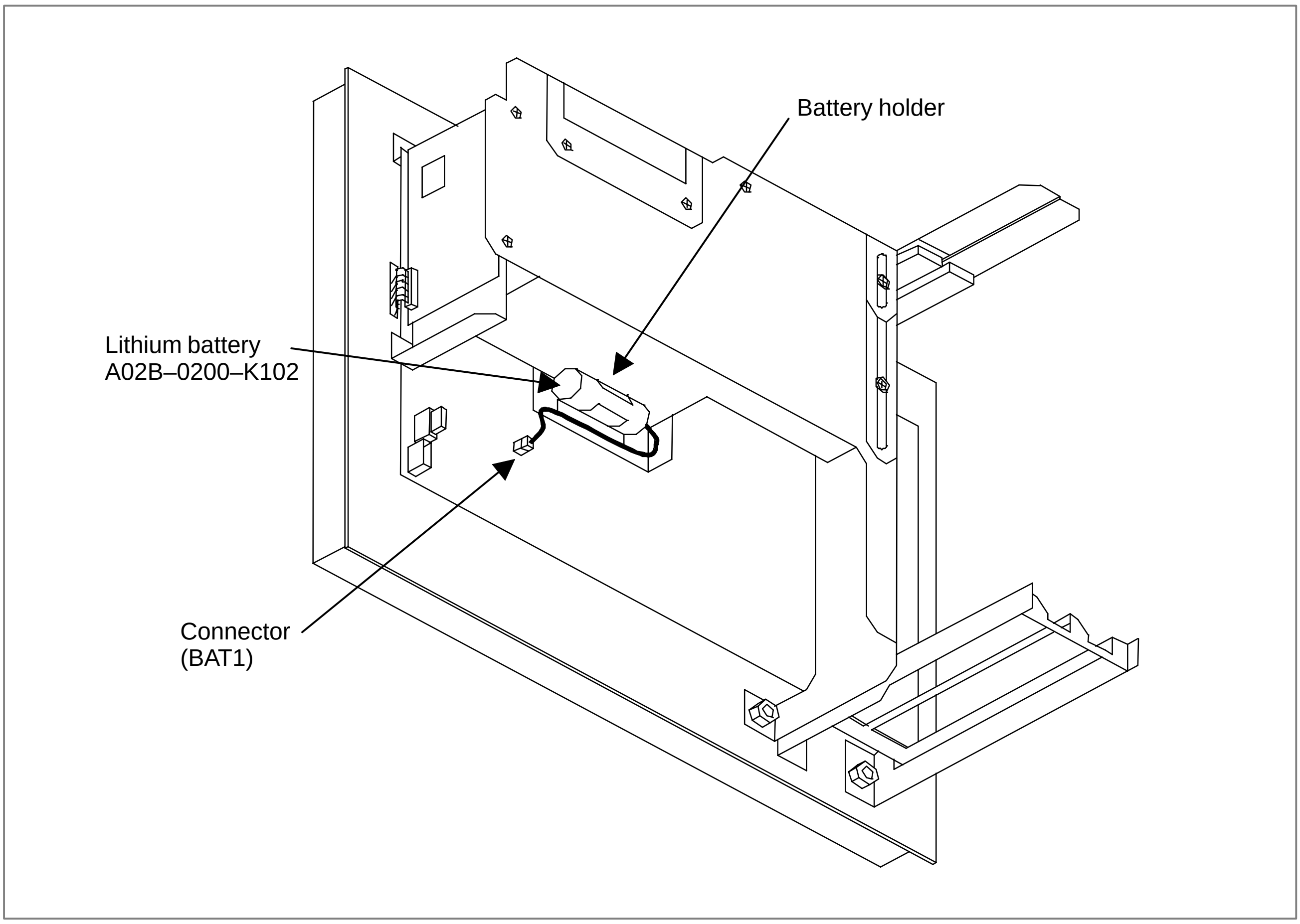


Fig. 1.3 Lithium battery connection for CNC display unit with PC functions

1.4 BATTERY FOR SEPARATE ABSOLUTE PULSE CODERS (6 VDC)

One battery unit can maintain current position data for six absolute pulse coders for a year.
When the voltage of the battery becomes low, APC alarms 306 to 308 (+ axis name) are displayed on the CRT display. When APC alarm 307 is displayed, replace the battery as soon as possible. In general, the battery should be replaced within one or two weeks, however, this depends on the number of pulse coders used.
If the voltage of the battery becomes any lower, the current positions for the pulse coders can no longer be maintained. Turning on the power to the control unit in this state causes APC alarm 300 (reference position return request alarm) to occur. Return the tool to the reference position after replacing the battery.
See Section 7.1.3 for details of connecting the battery to separate absolute pulse coders. The battery for the built–in absolute pulse coder is installed in the servo amplifier. For an explanation of the replacement procedure, refer to the FANUC SERVO MOTOR α Series Maintenance Manual.

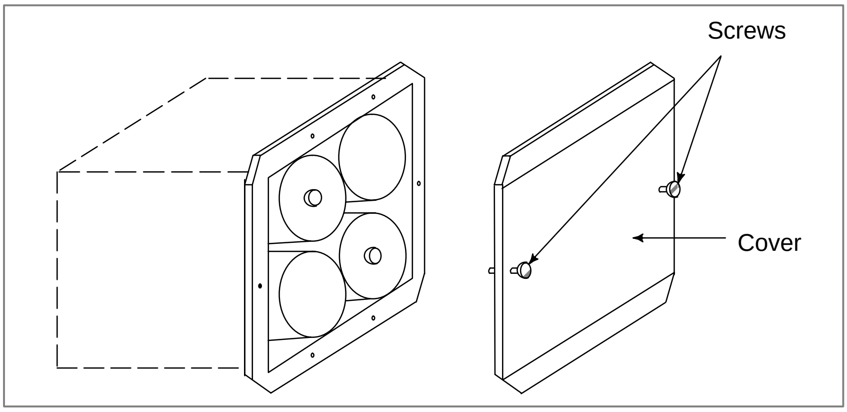
Replacing batteries

Obtain four commercially available alkaline batteries (size D).

(1) Turn on the power to the machine (*i* Series CNC).

(2) Loosen the screws on the battery case connected to the interface unit of the detector separately installed, and remove the cover.

(3) Replace the dry batteries in the case.
Note the polarity of the batteries as shown in the figure below (orient two batteries one way and the other two in the opposite direction).



(4) After installing the new batteries, replace the cover.

(5) Turn off the power to the machine (*i* Series CNC).

WARNING
If the batteries are installed incorrectly, an explosion may occur. Never use batteries other than the specified type (Size D alkaline batteries).

CAUTION
Replace batteries while the power to the *i* Series CNC is on. Note that, if batteries are replaced while no power is supplied to the CNC, the recorded absolute position is lost.

1.5 BATTERY FOR BUILT–IN ABSOLUTE PULSE CODERS (6 VDC)

When the battery voltage falls, APC alarms 306 to 308 are displayed on the screen. When APC alarm 307 is displayed, replace the battery as soon as possible. In general, the battery should be replaced within one or two weeks of the alarm first appearing. This, however, depends on the number of pulse coders being used.
If the battery voltage drops any further, the current positions for the pulse coders will be lost. Turning on the power to the control unit in this state.

(1) APC alarm 306 (battery zero alarm) and APC alarm 300 (reference position return request alarm) occur. Return the tool to the reference position after replacing the battery.

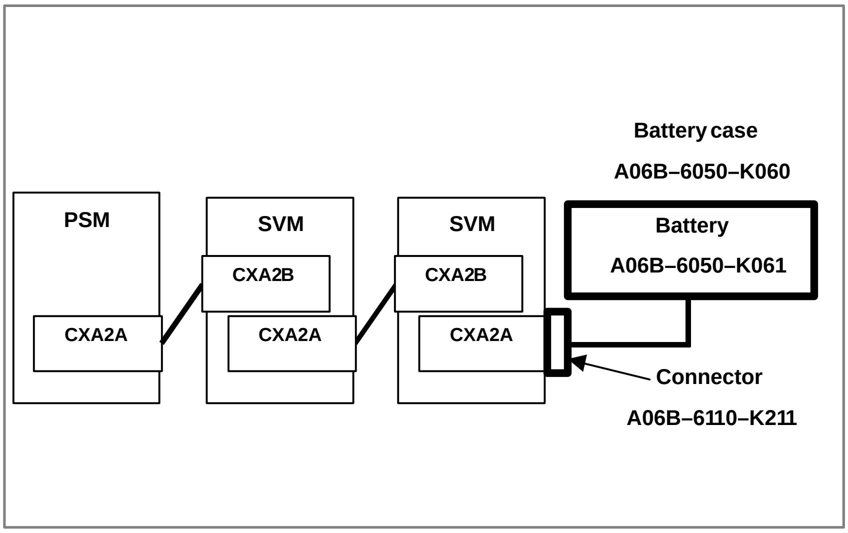
(2) The battery for the built–in absolute pulse coder is connected with the servo amplifier.

(3) Note that the attachment methods of the battery and ordering specifications of battery are different from SERVO AMPLIFIER α*i* series (SVM) to SERVO AMPLIFIER β series.

1.5.1 Method of Replacing Battery for Servo Amplifier α*i* series

The battery unit for the absolute pulse coder can be connected using [Connection scheme 1] and [Connection scheme 2] explained below. Refer to Subsection 9.3.2.6, “Battery” in “FANUC SERVO AMPLIFIER α*i* series Descriptions (B–63282EN)” for details.

[Connection scheme 1] Supplying power from one battery unit to more than one SVM



– If a low battery voltage or a battery voltage of 0 V is indicated by an APC (absolute pulse coder) alarm, replace the battery.
  If a battery voltage of 0 V is indicated, you need to make a zero point return.

– The absolute pulse coder of the ai series servo motor is incorporated with a backup capacitor as standard. This backup capacitor enables an absolute position detection to be continued for about 10 minutes. Therefore, no zero point return need be performed if the time during which servo amplifier power is kept off for battery replacement is within 10 minutes.

– When the replacing of the battery takes more than 10 minutes, replace the battery after see Item [Caution No.1 for battery replacement].

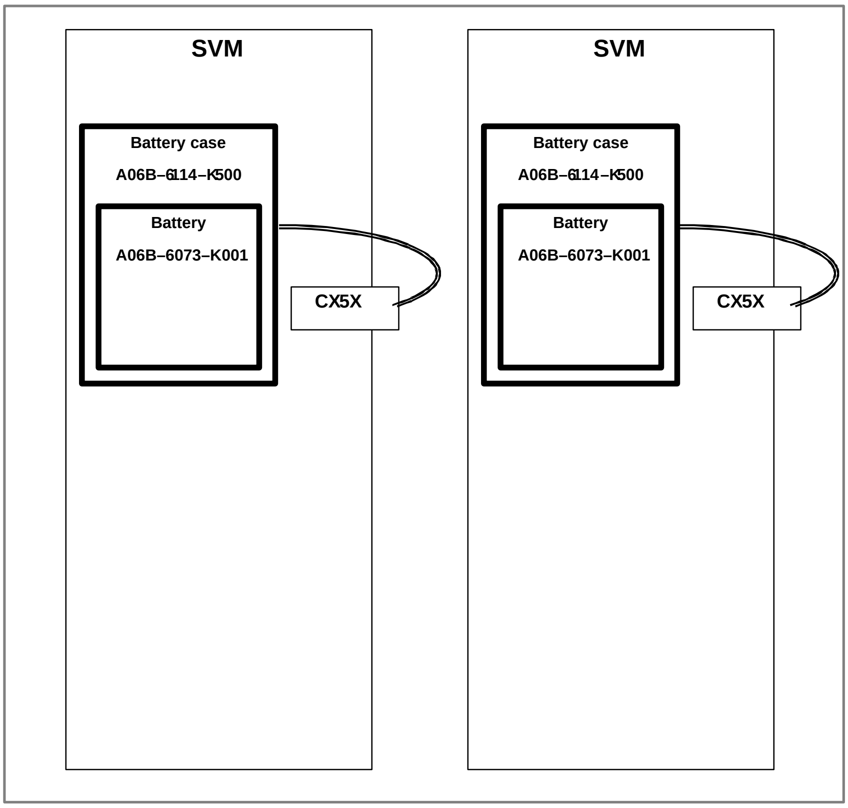
– The service life of the batteries is about two years if they are used in a six–axis configuration with α*i* series servo motors and one year if they are used in a six–axis configuration with α series servo motors. FANUC recommends that you replace the batteries periodically according to the battery service life.

– The battery unit consists of four R20 alkaline batteries. Commercial batteries can be used in the battery unit. The optional battery offered by FANUC is A06B–6050–K061.

WARNING

1 Do not connect more than one battery to the same BATL (B3) line. If the output voltage is different between the batteries, they may be short–circuited, resulting in the batteries becoming very hot.
2 Install the battery with correct polarity. If the battery is installed with incorrect polarity, it may overheat, blow out, or catch fire.

[Connection scheme 2] Incorporating each SVM with batteries



– If a low battery voltage or a battery voltage of 0 V is indicated by an APC (absolute pulse coder) alarm, replace the battery(A06B–6073–K001). If a battery voltage of 0 V is indicated, you need to make a zero point return.

– The absolute pulse coder of the αi series servo motor is incorporated with a backup capacitor as standard. This backup capacitor enables an absolute position detection to be continued for about 10 minutes. Therefore, no zero point return need be performed if the time during which servo amplifier power is kept off for battery replacement is within 10 minutes.
  On the contrary, the absolute pulse coder of the standard α series servo motor is not incorporated with a backup capacitor. Be careful when replacing the battery for this pulse coder. See [Caution No.1 for battery replacement] at the end of this section for details.
– The service life of the batteries is about two years with αi series servo motors and one year with α series servo motors.
  FANUC recommends that you replace the batteries periodically according to the battery service life.
– The built–in batteries are not commercially available. They must be purchased from FANUC. So, FANUC recommends that you keep spares.

WARNING

1 When using the built–in batteries (A06B–6073–K001), do not connect them to the BATL (B3) of connector CXA2A/CXA2B.
  The output voltages from different SVM batteries may be short–circuited, resulting in the batteries becoming very hot.
2 Do not connect more than one battery to the same BATL (B3) line. If the output voltage is different between the batteries, they may be short–circuited, resulting in the batteries becoming very hot.
3 Install the battery with correct polarity. If the battery is installed with incorrect polarity, it may overheat, blow out, or catch fire.

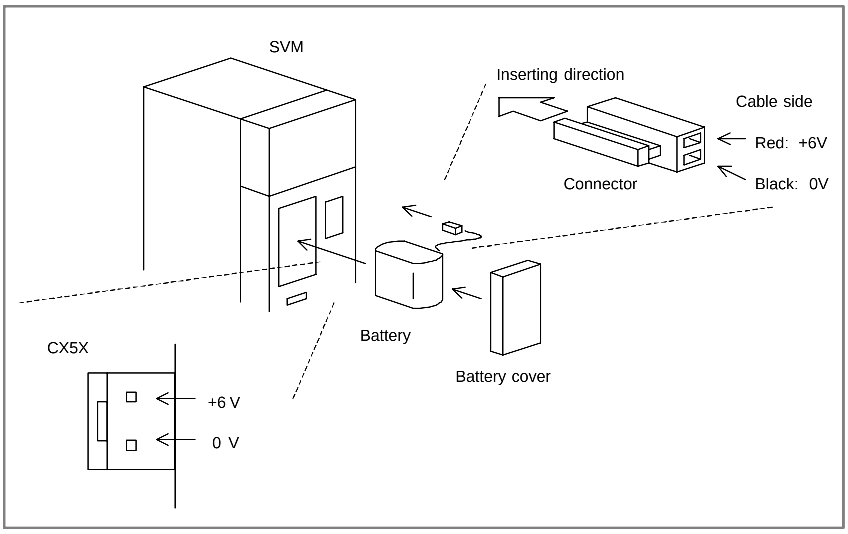
[Installation procedure for the battery]

(1) Remove the battery cover from the SVM.

(2) Install the battery in the SVM as shown in the figure below.

(3) Install the battery cover.

(4) Attach the battery connector to CX5X of the SVM.



CAUTION

1 When the battery is installed in the SVM from the side from which the cable is drawn, the cable may be stretched tight, which can lead to a poor contact condition. Therefore, install the battery so that the cable is not extended tightly.

2 Be careful when handling the connector. See [Caution No.2 for battery replacement] at the end of this section for details.

[Caution No.1 for battery replacement]

The pulse coder for the α series servo motor is not incorporated with a backup capacitor as standard. To keep the absolute position information in the absolute pulse coder, you need to keep the control power turned on during battery replacement. Follow the procedure explained below.

[Replacing procedure for the battery]

1. Make sure that the power to the SVM is on (the 7–segment LED on the front of the SVM is on).
2. Make sure that the emergency stop button of the system has been pressed.
3. Make sure that the motor is not activated.
4. Make sure that the DC link charge LED of the SVM is off.
5. Remove the old battery, and install a new battery.
6. This completes the replacement. You can turn off the power to the system.

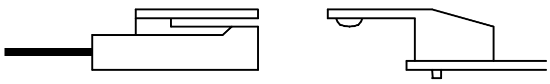
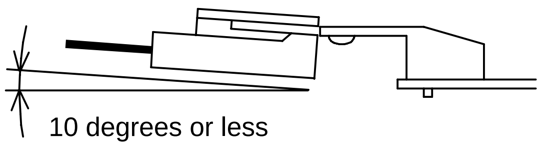
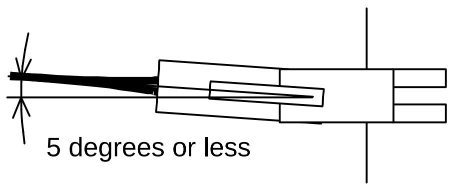
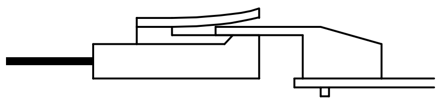
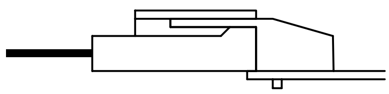
⚠ WARNING

1. When replacing the battery, be careful not to touch bare metal parts in the panel. In particular, be careful not to touch any high–voltage circuits due to the electric shock hazard.
2. Before replacing the battery, check that the DC link charge confirmation LED on the front of the servo amplifier is off. Neglecting this check creates an electric shock hazard.
3. Install the battery with correct polarity. If the battery is installed with incorrect polarity, it may overheat, blow out, or catch fire.
4. Avoid a short–circuit between the +6V and 0V lines of a battery or cable. A short–circuit may lead to a hot battery, an explosion, or fire.

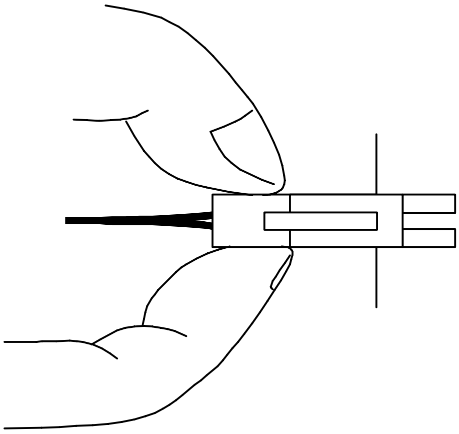
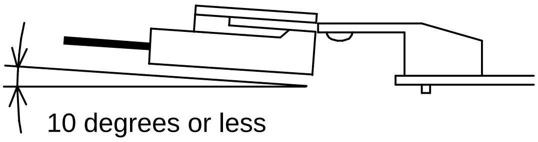
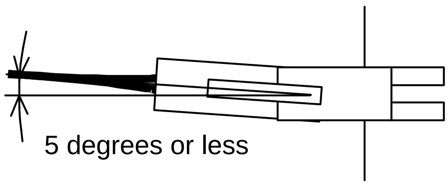
[Caution No.2 for battery replacement]

If an excessive strain is applied to a connector when it is inserted or removed, a poor contact may result. When inserting and removing the battery connector, therefore, be careful not to apply an excessive wrenching force to it; just follow the instructions given in the following table.

(1) Attaching connectors

<1>		Check the attachment position.
<2>		Plug the cable connector while raising it slightly.
<5>		Here, the angle of the cable connector to the horizontal must be 5 degrees or less.
<3>		After passing the lock pin, insert the connector straight.
<4>		The attachment of the connector is completed.

(2) Detaching the connector

<1>		Hold both the sides of the cable insulator and the cable, and pull them horizontally.
<2>		Pull out the cable side while raising it slightly.
<3>		Here, the angle of the cable to the horizontal must be 5 degrees or less.

1.5.2 Method of Replacing Battery for Servo Amplifier β series

The battery is connected in either of 2 ways as follows.

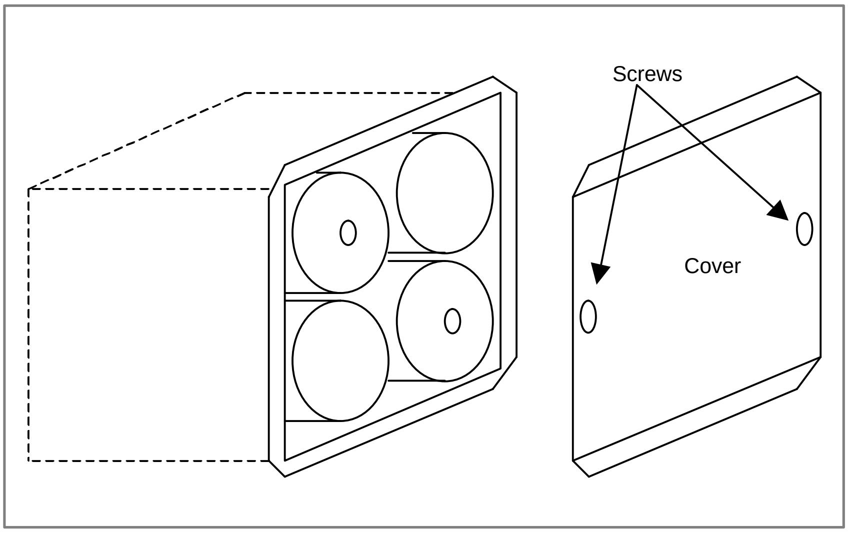
Method 1: Use the battery case (A06B–6050–K060).
Use the battery: A06B–6050–K061 or D–size alkaline battery.
Method 2: Attach the lithium battery to the SVM.
Use the battery: A06B–6093–K001.

Method	Item	Ordering specification
Method 1	Battery (4 pieces of D–size alkaline battery)	A06B–6050–K061
Method 2	Battery (Lithium battery)	A06B–6093–K001

- Replacement of batteries in the battery case. (Method 1)
  Replace four D–size alkaline batteries in the battery case installed in the machine.

  [Attachment procedure]

  (1) Turn the servo unit (machine) on.
  (2) Place the machine in the emergency stop state.
  (3) Confirm that the servo motors are not active.
  (4) Have four D–size alkaline batteries on hand.
  (5) Loosen the screws on the battery case. Remove the cover.
  (6) Replace the alkaline batteries in the case. Pay careful attention to the polarity of the alkaline batteries.
  (7) Attach the cover.



- Attach the lithium battery to the amplifier. (Method 2)
  Attach the lithium (A06B–6093–K001) battery to the amplifier.

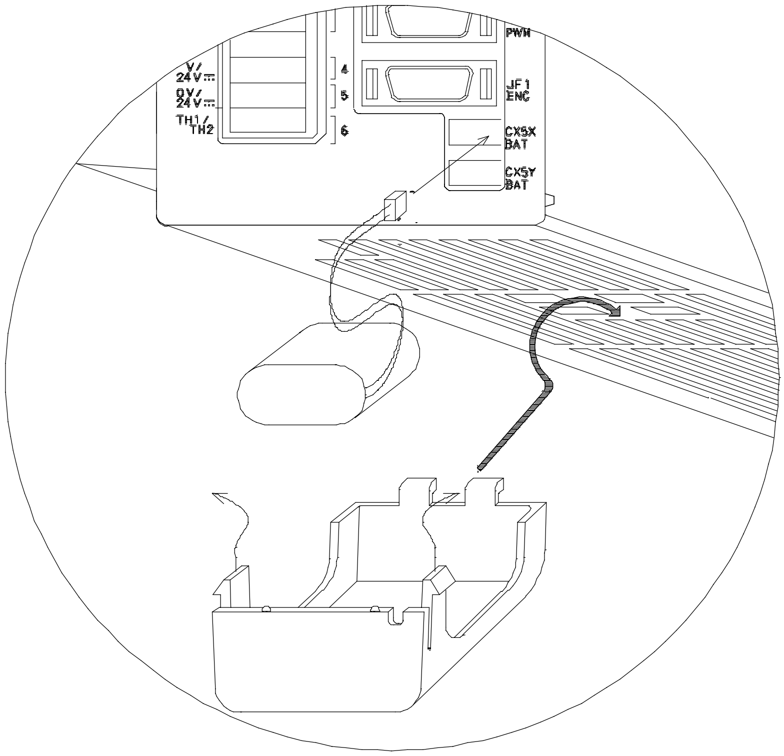
  [Attachment procedure]

  (1) Turn the servo unit (machine) on.
  (2) Place the machine in the emergency stop state.
  (3) Confirm that the servo motors are not active.
  (4) In case of SVU–12 or SVU–20, remove the battery cover under the servo unit grasping its left and right sides. In case of SVU–40 or SVU–80, remove the cover attached on right side of the servo unit grasping its upper and lower sides.

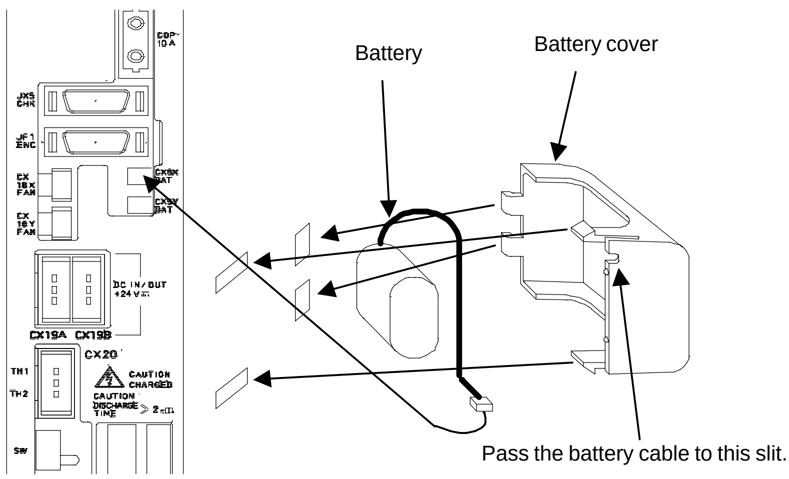
(5) Remove the battery from the servo unit.

(6) Replace the battery and connect the battery cable with the connector CX5X or CX5Y of the servo unit.

(7) Mount the battery cover.



SVU–12, SVU–20



SVU–40, SVU–80

CAUTION

The connector of the battery can be connected with either of CX5X and CX5Y.

Used batteries

Old batteries should be disposed as “INDUSTRIAL WASTES” according to the regulations of the country or autonomy where your machine has been installed.

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	With CUSTOM MACRO B
0			○	○		◦				0			○			◦				Number 0		
1	○		○	○		○			○	1						○			○	Number 1		
2	○		○	○		◦		○		2						◦		○		Number 2		
3			○	○		◦		○	○	3				○		◦		○	○	Number 3		
4	○		○	○		◦	○			4						◦	○			Number 4		
5			○	○		◦	○		○	5				○		◦	○		○	Number 5		
6			○	○		◦	○	○		6				○		◦	○	○		Number 6		
7	○		○	○		◦	○	○	○	7						◦	○	○	○	Number 7		
8	○		○	○	○	◦				8					○	◦				Number 8		
9			○	○	○	◦			○	9				○	○	◦			○	Number 9		
A		○				◦			○	a		○	○			◦			○	Address A		
B		○				◦		○		b		○	○			◦		○		Address B		
C	○	○				◦		○	○	c		○	○	○		◦		○	○	Address C		
D		○				◦	○			d		○	○			◦	○			Address D		
E	○	○				◦	○		○	e		○	○	○		◦	○		○	Address E		
F	○	○				◦	○	○		f		○	○	○		◦	○	○		Address F		
G		○				◦	○	○	○	g		○	○			◦	○	○	○	Address G		
H		○			○	◦				h		○	○		○	◦				Address H		
I	○	○			○	◦			○	i		○	○	○	○	◦			○	Address I		
J	○	○			○	◦		○		j		○		○		◦		○	○	Address J		
K		○			○	◦		○	○	k		○		○		◦		○		Address K		
L	○	○			○	◦	○			l		○				◦		○	○	Address L		
M		○			○	◦	○		○	m		○		○		◦	○			Address M		
N		○			○	◦	○	○		n		○				◦	○		○	Address N		
O	○	○			○	◦	○	○	○	o		○				◦	○	○		Address O		
P		○		○		◦				p		○		○		◦	○	○	○	Address P		
Q	○	○		○		◦			○	q		○		○	○	◦				Address Q		
R	○	○		○		◦		○		r		○			○	◦			○	Address R		
S		○		○		◦		○	○	s			○	○		◦		○		Address S		
T	○	○		○		◦	○			t			○			◦		○	○	Address T		
U		○		○		◦	○		○	u			○	○		◦	○			Address U		
V		○		○		◦	○	○		v			○			◦	○		○	Address V		
W	○	○		○		◦	○	○	○	w			○			◦	○	○		Address W		
X	○	○		○	○	◦				x			○	○		◦	○	○	○	Address X		
Y		○		○	○	◦			○	y			○	○	○	◦				Address Y		
Z		○		○	○	◦		○		z			○		○	◦			○	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	With CUSTOM MACRO B
DEL	○	○	○	○	○	∘	○	○	○	Del		○	○	○	○	∘	○	○	○	Delete (deleting a mispunch)	×	×
NUL						∘				Blank						∘				No punch. With EIA code, this code cannot be used in a significant information section.	×	×
BS	○				○	∘				BS			○		○	∘		○		Backspace	×	×
HT					○	∘			○	Tab			○	○	○	∘	○	○		Tabulator	×	×
LF or NL					○	∘		○		CR or EOB	○					∘				End of block		
CR	○				○	∘	○		○	____										Carriage return	×	×
SP	○		○			∘				SP				○		∘				Space	□	□
%	○		○			∘	○		○	ER					○	∘		○	○	Absolute rewind stop		
(			○		○	∘				(2–4–5)				○	○	∘		○		Control out (start of comment)		
)	○		○		○	∘			○	(2–4–7)		○			○	∘		○		Control in (end of comment)		
+			○		○	∘		○	○	+		○	○	○		∘				Plus sign	Δ	
–			○		○	∘	○		○	–		○				∘				Minus sign		
:			○	○	○	∘		○		____										Colon (address O)		
/	○		○		○	∘	○	○	○	/			○	○		∘			○	Optional block skip		
.			○		○	∘	○	○		.		○	○		○	∘		○	○	Period (decimal point)		
#	○		○			∘		○	○	Parameter (No. 6012)										Sharp		
$			○			∘	○			____										Dollar sign	Δ	○
&	○		○			∘	○	○		&					○	∘	○	○		Ampersand	Δ	○
’			○			∘	○	○	○	___										Apostrophe	Δ	○
*	○		○		○	∘		○		Parameter (No. 6010)										Asterisk	Δ	
,	○		○		○	∘	○			,			○	○	○	∘		○	○	Comma		
;	○		○	○	○	∘		○	○	____										Semicolon	Δ	Δ
<			○	○	○	∘	○			____										Left angle bracket	Δ	Δ
=	○		○	○	○	∘	○		○	Parameter (No. 6011)										Equal sign	Δ	
>	○		○	○	○	∘	○	○		____										Right angle bracket	Δ	Δ
?			○	○	○	∘	○	○	○	____										Question mark	Δ	○
@	○	○				∘				____										Commercial at mark	Δ	○
”			○					○		____										Quotation mark	Δ	Δ
[	○	○		○	○	∘		○	○	Parameter (No. 6013)										Left square bracket	Δ	
]	○	○		○	○	∘	○		○	Parameter (No. 6014)										Right square bracket	Δ	

NOTE

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. If 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.

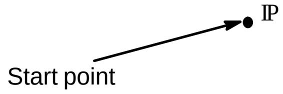
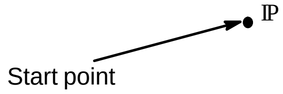
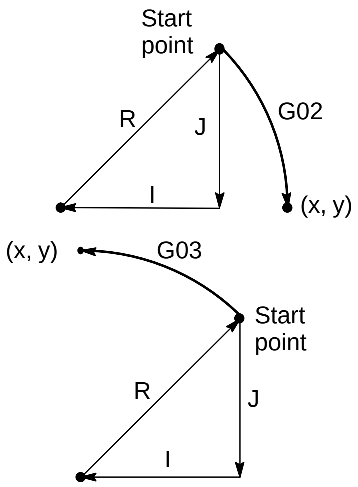
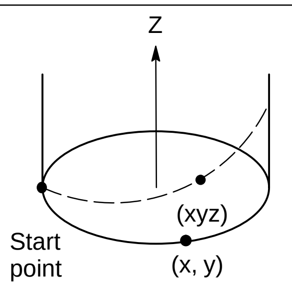
2 Codes not in this table are ignored if their parity is correct.

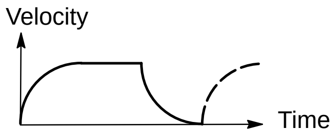
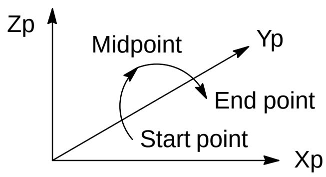
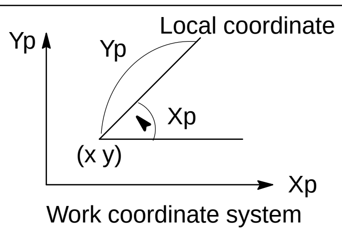
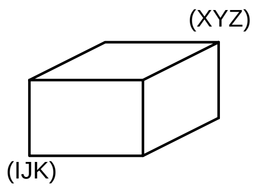
3 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.

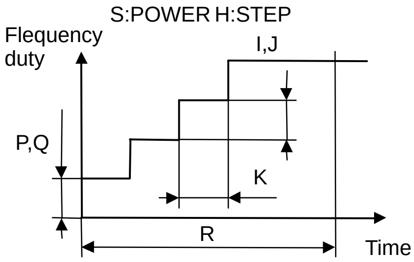
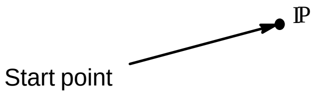
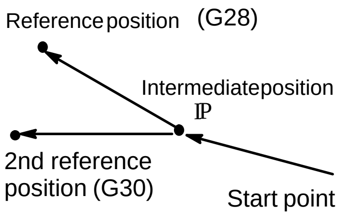
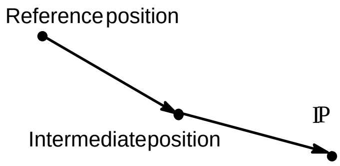
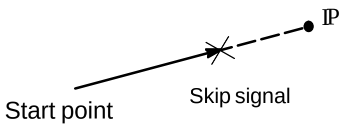
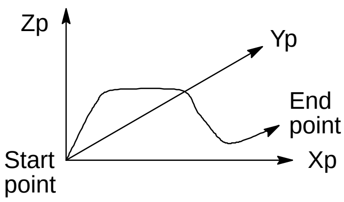
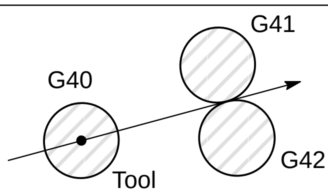
4 A character with all eight holes punched is ignored and does not generate TH alarm in EIA code.

B LIST OF FUNCTIONS AND TAPE FORMAT

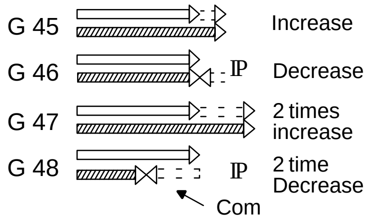
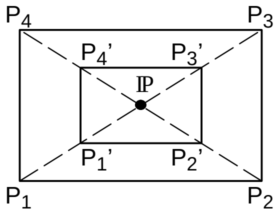
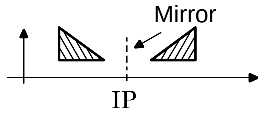
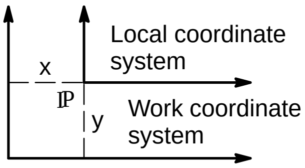
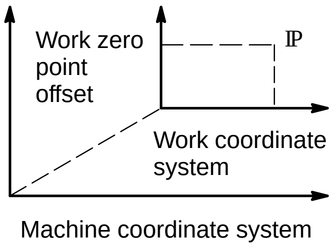
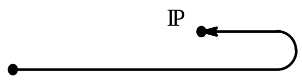
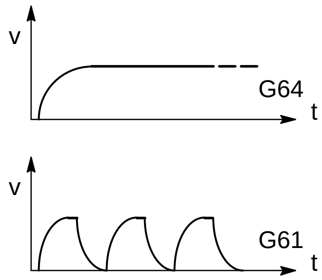
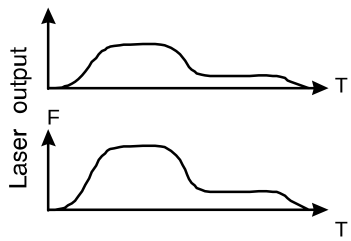
Some functions cannot be added as options depending on the model.
In the tables below, IP __:presents a combination of arbitrary axis addresses using X,Y,Z,A,B and C (such as X_Y_Z_A_).
x = 1st basic axis (X usually)
y = 2nd basic axis (Y usually)
z = 3rd basic axis (Z usually)

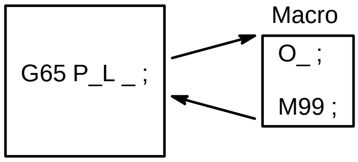
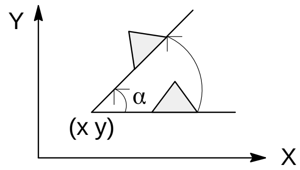
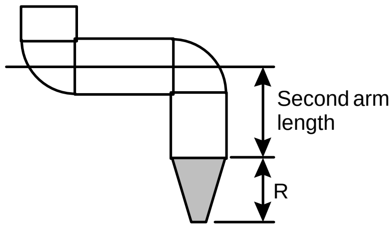
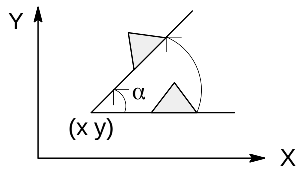
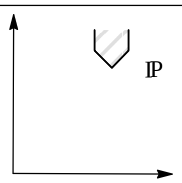
Functions	Illustration	Tape format
Positioning (G00)		G00 IP _ ;
Linear interpolation (G01)		G01 IP _ S_P_Q_F_;
Circular interpolation (G02, G03)		$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\ Y\_ \begin{Bmatrix} R\_ \\ I\_\ J\_ \end{Bmatrix} S\_\ P\_\ Q\_\ F\_\ ;$ $G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\ Z\_ \begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix} S\_\ P\_\ Q\_\ F\_\ ;$ $G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_\ Z\_ \begin{Bmatrix} R\_ \\ J\_\ K\_ \end{Bmatrix} S\_\ P\_\ Q\_\ F\_\ ;$
Helical interpolation (G02, G03)	 (In case of X–Y plane)	$G17 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\ Y\_ \begin{Bmatrix} R\_ \\ I\_\ J\_ \end{Bmatrix} \alpha\_\ F\_\ ;$ $G18 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} X\_\ Z\_ \begin{Bmatrix} R\_ \\ I\_\ K\_ \end{Bmatrix} \alpha\_\ F\_\ ;$ $G19 \begin{Bmatrix} G02 \\ G03 \end{Bmatrix} Y\_\ Z\_ \begin{Bmatrix} R\_ \\ J\_\ K\_ \end{Bmatrix} \alpha\_\ F\_\ ;$ α : Any axis other than circular interpolation axes.
Dwell (G04)		$G04 \begin{Bmatrix} X\_ \\ P\_ \end{Bmatrix} ;$

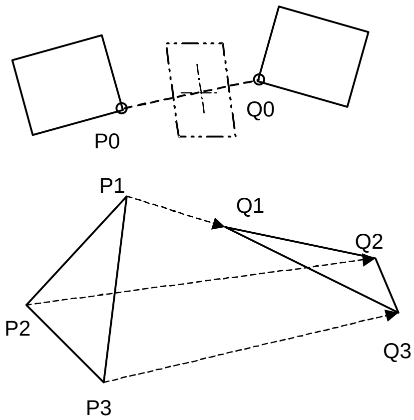
Functions	Illustration	Tape format
Cylindrical interpolation (G07.1)		G07IP_R_; Cylindrical interpolation R: Radius of cylinder G07IP 0; Cylindrical interpolation cancel
Look–ahead control (G08)		G08 P1: Look–ahead control mode on G08 P0: Look–ahead control mode off
Exact stop (G09)		$G09 \left\{ \begin{matrix} G01 \\ G02 \\ G03 \end{matrix} \right\}$ IP_;
Change of offset value by program (G10)		Tool offset memory A G10 L11 P_R_; Tool offset memory B G10 L10 P_R_; (Wear offset value) G11 L11 P_R_; (Geometry offset value) Tool offset memory C G10 L10 P_R_; (Wear offset value/H) G10 L11 P_R_; (Geometry offset value/H) G10 L12 P_R_; (Wear offset value/D) G10 L13 P_R_; (Geometry offset value/D)
Spatial circular interpolation (G12)		G12IP _S _ P_ Q_ F ; Midpoint G12IP _ ; End point
Gap control (G13, G14)		G13 P_L_ ; Gap mode G14 ; Gap mode cancellation
Polar coordinate (G15, G16)		G17 G16 Xp_ Yp_ . . . ; G18 G16 Zp_ Xp_ . . . ; G19 G16 Yp_ Zp_ . . . ; G15 ; Cancel
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;

<table>
<tr><th>Functions</th><th>Illustration</th><th>Tape format</th></tr>
<tr><td>Piercing</td><td>
</td><td>G24 S_P_Q_I_J_K_H_R_</td></tr>
<tr><td>Reference position return check (G27)</td><td>
</td><td>G27 IP_ ;</td></tr>
<tr><td>Reference position return (G28)
2nd, reference position re-turn (G30)</td><td>
</td><td>G28 IP_ ;
G30 IP_ ;</td></tr>
<tr><td>Return from reference position to start point (G29)</td><td>
</td><td>G29 IP_ ;</td></tr>
<tr><td>Skip function (G31)</td><td>
</td><td>G31 IP_ F_;</td></tr>
<tr><td>Assist gas control</td><td></td><td>G32 { P_Q_
P_T_R_ }</td></tr>
<tr><td>Spatial corner R insertion (G33, G34)</td><td>
</td><td>G33 G01 ; Spatical corner R insertion mode on
X_ Y_ Z_ A_ B_ R_ F_ ;
X_ Y_ Z_ A_ B_ ;
X_ Y_ Z_ A_ B_ ;
R_ ;
X_ Y_ Z_ A_ B_ ;
G34 ; Mode corner</td></tr>
<tr><td>Cutter compensation C (G40 – G42)</td><td>
</td><td>{ G17
G18
G19 } { G41
G42 } D_ ;

D : Tool offset
G40 : Cancel</td></tr>
<tr><td>Normal direction control (G40.1, G41.1, G42.1)
(G150, G151, G152)</td><td></td><td>G41.1 (G151) Normal direction control left
G42.1 (G152) Normal direction control right
G40.1 (G150) Normal direction control cansel</td></tr>
</table>

Functions	Illustration	Tape format
Tool offset (G45 to G48)		{G45 / G46 / G47 / G48} IP _ D_; D : Tool offset number
Scaling (G50, G51)		G51 IP_ P_; P, I, J, K : Scaling magnification X, Y, Z : Center coordinate value of scaling G50 ; Cancel
Programmable mirror image		G51.1 IP _ ; G50.1 ; ...Cancel
Setting of local coordinate system (G52)		G52 IP_ ;
Command in machine coordinate system (G53)		G53 IP_ ;
Selection of work coordinate system (G54 – G59)		{G54 / : / G59} IP_ ;
Single direction positioning (G60)		G60 IP_ ;
Exact stop mode, Automatic corner override		G61_ ; Exact stop mode G62_ ; Automatic corner override
Power control (G63)		G63 P1 ; Power control mode G63 P0 ; Power control mode cancellation

Functions	Illustration	Tape format
Custom macro (G65, G66, G67)		One–shot call G65 P_ L_ <Argument assignment> ; P : Program No. L : Number of repeatition Modal call G66 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
Three–dimensional coordi-nate conversion (G68, G69)		G68 X_ Y_ Z_ A_ B_ I_ J_ K_ ; G01 IP_ F_; G02 IP_ ; G03 IP_ ; . . . G69 ;
Processing head A axis length compensation (G71)		G71 R_;
Coordinate system rotation (G84, G85)	 (In case of X–Y plane)	G84 { G17 X_ Y_ / G18 Z_ X_ / G19 Y_ Z_ } R α ;
Absolute/incremental programming (G90/G91)		G90_ ; Absolute command G91_ ; Incremental command G90_ G91_ ; Combined use
Change of workpiece coordinate system (G92)		G92 IP_ ;

Functions	Illustration	Tape format
Workpiece coordinate system preset (G92.1)		G92.1 IP 0 ;
Three–dimensional transform (G98, G99)		Mirror image transform G98 P0 X_Y_Z_; Base point G98 Q0 X_Y_Z_; Target point Movement transform G98 P1 X_Y_Z_; Base point G98 P2 X_Y_Z_; Base point G98 P3 X_Y_Z_; Base point G98 Q1 X_Y_Z_; Target point G98 Q2 X_Y_Z_; Target point G98 Q3 X_Y_Z_; Target point

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 **Note**	240000 mm/min
Feedrate range **Note**	1 to 240000 mm/min
Incremental feed	0.001, 0.01, 0.1, 1 mm/step
Cutter 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 **Note**	240000 mm/min
Feedrate range **Note**	0.01 to 9600 inch/min
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step
Cutter 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 **Note**	9600 inch/min
Feedrate range **Note**	0.01 to 9600 inch/min
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step
Cutter 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 **Note**	9600 inch/min
Feedrate range **Note**	1 to 240000 mm/min
Incremental feed	0.001, 0.01, 0.1, 1 mm/step
Cutter compensation	0 to ±999.999 mm
Dwell time	0 to 99999.999 sec

Rotation axis

	Increment system
	IS–B
Least input increment	0.001 deg
Least command increment	0.001 deg
Max. programmable dimension	±99999.999 deg
Max. rapid traverse **Note**	240000 deg/min
Feedrate range **Note**	1 to 240000 deg/min
Incremental feed	0.001, 0.01, 0.1, 1 deg/step

NOTE
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.

Laser command

Peak power	0 to 7000W
Pulse frequency	5 to 2000Hz
Pulse duty	0 to 100%
Displacement from the Gap control base point	0 to ±32,767mm

D NOMOGRAPHS

D.1 BEAM 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 beam path (nozzle center path) and the programmed path as shown in Fig.D.1 (a).
Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

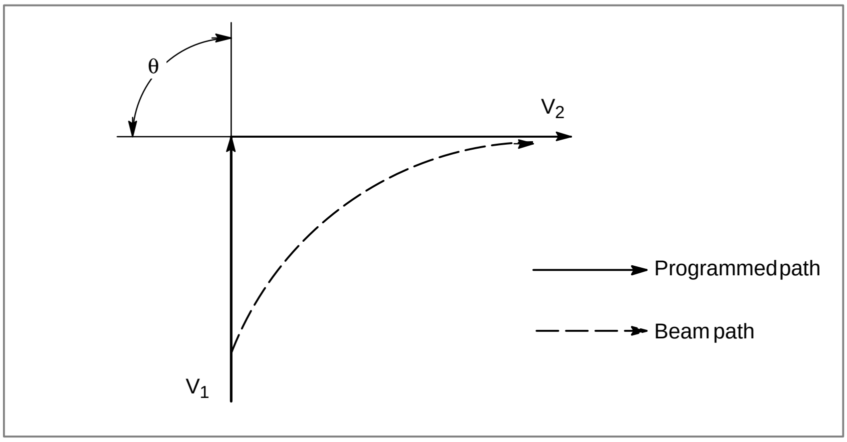


Fig.D.1 (a) Slight deviation between the beam path and the programmed path

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 beam path and above beam 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 beam path shown in Fig.D.1 (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.)

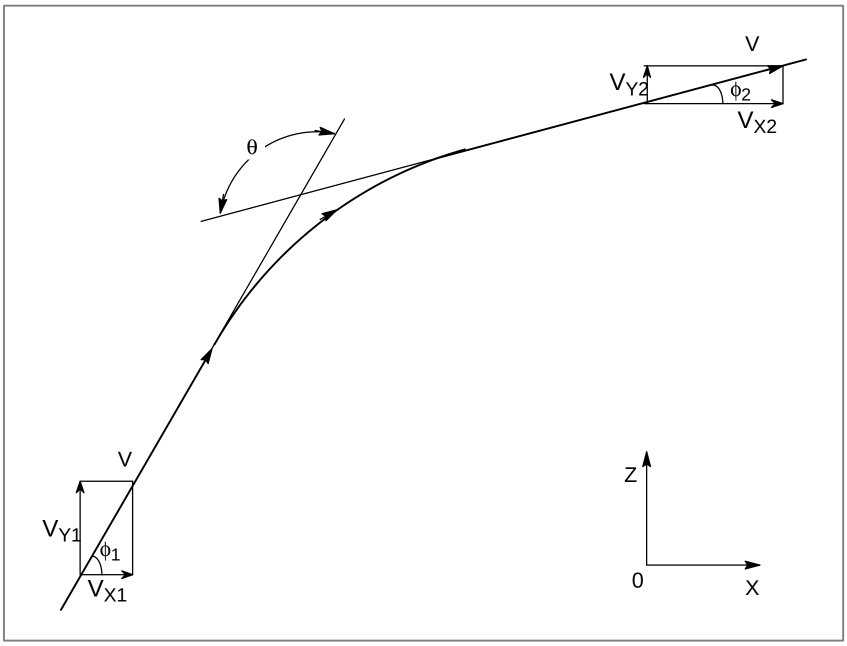


Fig.D.1 (b) Example of tool path

- **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

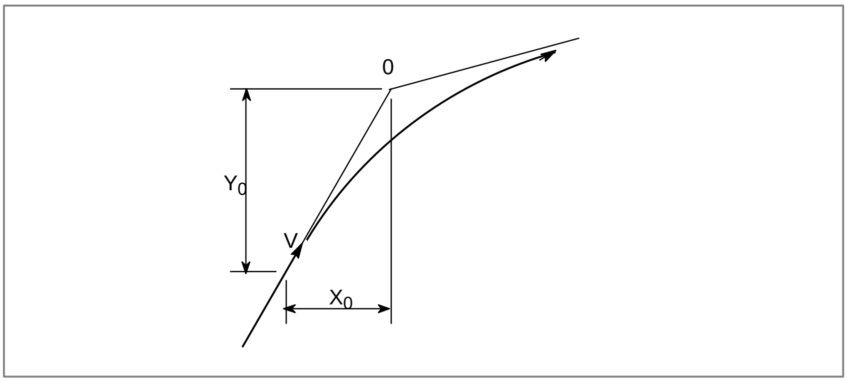


Fig.D.1 (c) Initial value

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:

$$X(t) = \int_0^t V_X(t)dt-X_0$$

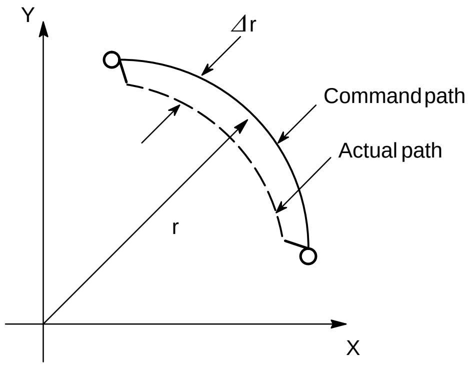
$$= \frac{V_{X2}-V_{X1}}{T_1-T_2}\{T_1{}^2\exp(-\frac{t}{T_1})-T_2{}^2\exp(-\frac{t}{T_2})\}-V_{X2}(T_1 + T_2-t)$$

$$Y(t) = \int_0^t V_Y(t)dt-Y_0$$

$$= \frac{V_{Y2}-V_{Y1}}{T_1-T_2}\{T_1{}^2\exp(-\frac{t}{T_1})-T_2{}^2\exp(-\frac{t}{T_2})\}-V_{Y2}(T_1 + T_2-t)$$

D.2 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 beam 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:



$$\Delta r = \frac{1}{2}(T_1{}^2 + T_2{}^2)\frac{V^2}{r} \quad .... (1)$$

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 bell–shaped acceleration/deceleration and 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 bell–shaped acceleration/deceleration and 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 (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 :
○:The status is not changed or the movement is continued.
×:The status is cancelled or the movement is interrupted.

Item		When turning power on	Cleared	Reset
Setting data	Offset value	○	○	○
	Data set by the MDI setting operation	○	○	○
	Parameter	○	○	○
Various data	Programs in memory	○	○	○
	Contents in the buffer storage	×	×	○ : MDI mode × : Other mode
	Display of sequence number	○	○ (Note 1)	○ (Note 1)
	One shot G code	×	×	×
	Modal G code	Initial G codes. (The G20 and G21 codes return to the same state they were in when the power was last turned off.)	Initial G codes. (G20/G21 are not changed.)	○
	F	Zero	Zero	○
	T, M	×	○	○
	K (Number of repeats)	×	×	×
Workpiece coordinate value		Zero	○	○

Item		When turning power on	Cleared	Reset
Action in operation	Movement	×	×	×
	Dwell	×	×	×
	Issuance of M, T codes	×	×	×
	Length compensation	×	Depending on parameter LVK(No.5003#6)	○ : MDI mode Other modes depend on parameter LVK(No.5003#6).
	Cutter compensation	×	×	○ : MDI mode × : Other modes
	Storing called subprogram number	×	× (Note 2)	○ : MDI mode × : Other modes (Note 2)
Output signals	CNC alarm signal AL	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm
	Reference position return completion LED	×	○ (× : Emergency stop)	○ (× : Emergency stop)
	T and B codes	×	○	○
	M code	×	×	×
	M, T strobe signals	×	×	×
	CNC ready signal MA	ON	○	○
	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)	×	×	×

NOTE

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 head of main program.
Execution cannot be started from the middle of the subprogram.

F CHARACTER–TO–CODES CORRESPONDENCE TABLE

Char-acter	Code	Comment	Char-acter	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.
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.
022	NO CIRCULAR RADIUS	When circular interpolation is specified, neither R (specifying an arc radius), nor I, J, and K (specifying the distance from a start point to the center) is specified.
025	CANNOT COMMAND F0 IN G02/G03	F0 (fast feed) was instructed by F1 –digit column feed in circular interpolation. 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.
030	ILLEGAL OFFSET NUMBER	The offset number specified by D/H code for cutter compensation is too large. Alternatively, the number of the additional workpiece coordinate system specified by a P code 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.

Number	Message	Contents
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.
042	G45/G48 NOT ALLOWED IN CRC	Tool offset (G45 to G48) is commanded in cutter compensation. Modify the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2, P3 and P4 are commanded for 2nd, 3rd and 4th reference position return command.
051	MISSING MOVE AFTER CHF/CNR	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
052	CODE IS NOT G01 AFTER CHF/CNR	The block next to the chamfering or corner R block is not G01,G02,or G03. Modify the program.
053	TOO MANY ADDRESS COMMANDS	For systems without the arbitary angle chamfering or corner R cutting, a comma was specified. For systems with this feature, a comma was followed by something other than R or C Correct the program.
055	MISSING MOVE VALUE IN CHF/CNR	In the arbitrary angle chamfering or corner R block, the move distance is less than chamfer or corner R amount. Alternatively, an arc end–point command has been omitted.
058	END POINT NOT FOUND	In a arbitrary angle chamfering or corner R cutting block, a specified axis is not in the selected plane. Correct the program.
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 125 (basic), 200 (option), 400 (option), or 1000 (option). 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.

Number	Message	Contents
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.
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.
088	LAN FILE TRANS ERROR (CHANNEL–1)	File data transfer over the OSI–Ethernet was terminated as a result of a transfer error.
089	LAN FILE TRANS ERROR (CHANNEL–2)	File data transfer over the OSI–Ethernet was terminated as a result of a transfer error.
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.
091	REFERENCE RETURN INCOMPLETE	In the automatic operation halt state, manual reference position return cannot be performed.
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.
109	FORMAT ERROR IN G08	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
110	DATA OVERFLOW	The absolute value of fixed decimal point display data exceeds the allowable range. Modify the program.

Number	Message	Contents
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.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high–speed cycle cutting. The header contents are improper in a high–speed cycle cutting. This alarm is given in the following cases: 1. The header corresponding to the specified machining cycle number called is not found. 2. The cycle connection data value is out of the allowable range (0 – 999). 3. The number of data in the header is out of the allowable range (0 – 32767). 4. The start data variable number of executable format data is out of the allowable range (#20000 – #85535). 5. The storing data variable number of executable format data is out of the allowable range (#85535). 6. The storing start data variable number of executable format datais overlapped with the variable number used in the header. Modify the program.
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	QUADRUPLICATE MACRO MODAL–CALL	A total of four macro calls and macro modal calls are nested. Correct 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, $1 \leqq n \leqq 3$ 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.
138	SUPERIMPOSED DATA OVERFLOW	In PMC–based axis control, the increment for pulse distribution on the CNC and PMC side are too large when the superimposed control extended function is used.
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.

Number	Message	Contents
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.
145	ILLEGAL CONDITIONS IN POLAR COORDINATE INTERPOLATION	The conditions are incorrect when the polar coordinate interpolation starts or it is canceled. 1) In modes other than G40, G12.1/G13.1 was specified. 2) An error is found in the plane selection. Parameters No. 5460 and No. 5461 are incorrectly specified. Modify the value of program or parameter.
146	IMPROPER G CODE	G codes which cannot be specified in the polar coordinate interpolation mode was specified. See section polar coordinate interpolation and modify the program.
148	ILLEGAL SETTING DATA	Automatic corner override deceleration rate is out of the settable range of judgement angle. Modify the parameters (No.1710 to No.1714)
160	G72.1 NESTING ERROR	Code G72.1 was specified in a sub–program after the same code had already been specified for copying with rotation.
161	G72.2 NESTING ERROR	Code G72.2 was specified in a sub–program after the same code had already been specified for parallel copying.
175	ILLEGAL G107 COMMAND	Conditions when performing cylindrical interpolation start or cancel not correct. To change the mode to the cylindrical interpolation mode, specify the command in a format of "G07.1 rotation–axis name radius of cylinder."
176	IMPROPER G–CODE IN G107	Any of the following G codes which cannot be specified in the cylindrical interpolation mode was specified. 1) G codes for positioning, such as G28 including the codes specifying the rapid traverse cycle 2) G codes for setting a coordinate system: G52,G92, 3) G code for selecting coordinate system: G53 G54–G59 Modify the program.
177	CHECK SUM ERROR (G05 MODE)	Check sum error Modify the program.
178	G05 COMMANDED IN G41/G42 MODE	G05 was commanded in the G41/G42 mode. Correct the program.
179	PARAM. (PRM NO. 7510) SETTING ERROR	The number of controlled axes set by the parameter 7510 exceeds the maximum number. Modify the parameter setting value.
180	COMMUNICATION ERROR (REMOTE BUF)	Remote buffer connection alarm has generated. Confirm the number of cables, parameters and I/O device.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.
210	CAN NOT COMAND M198/M99	1) M198 and M99 are executed in the schedule operation. Or M198 is executed in the DNC operation. 2) In a multiple repetitive pocketing canned cycle, an interrupt macro was specified, and M99 was executed.
212	ILLEGAL PLANE SELECT	The arbitrary angle chamfering or a corner R is commanded or the plane including an additional axis. Correct the program.
213	ILLEGAL COMMAND IN SYNCHRO–MODE	Any of the following alarms occurred in the operation with the simple synchronization control. 1) The program issued the move command to the slave axis. 2) The program issued the manual continuous feed/manual handle feed/incremental feed command to the slave axis. 3) The program issued the automatic reference position return command without executing the manual reference position return after the power was turned on. 4) The difference between the position error amount of the master and slave axes exceeded the value specified in parameter No. 8313.
214	ILLEGAL COMMAND IN SYNCHRO–MODE	Coordinate system is set or cutter compensation of the shift type is executed in the synchronous control. Correct the program.

Number	Message	Contents
222	DNC OP. NOT ALLOWED IN BG.–EDIT	Input and output are executed at a time in the background edition. Execute a correct operation.
224	RETURN TO REFERENCE POINT	Reference position return has not been performed before the automatic operation starts. Perform reference position return only when parameter ZRN_X (No.1005#0) 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.
232	TOO MANY HELICAL AXIS COMMANDS	Three or more axes (in the normal direction control mode two or more axes) were specified as helical axes in the helical interpolation mode.
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.
4000	P/S ALARM	Although the end of a record is reached, the near point is not found during near point search. ”END OF RECORD” was read during program restart (laser specification).
4001	P/S ALARM	The Z–axis is specified as the gap control axis during near point search or program restart (laser specification).
4002	G CODE UNAVAILABLE DURING SEARCH	G53 is specified in independent axis mode (bit 3 of PRM. 15600, MIA = 1) during near point search or program restart (laser specification).
4004	THREE–DIMENSIONAL COORDINATE CONVERSION ALARM	One or two of the I, J, and K commands are not specified or a command for the X, Y, Z, 4, or 5 axis is not specified.
4005	THREE–DIMENSIONAL COORDINATE CONVERSION ALARM	G27, G28, or G29 was issued during three–dimensional coordinate conversion.
4006	THREE–DIMENSIONAL COORDINATE CONVERSION ALARM	G68 was issued again during three–dimensional coordinate conversion.
4007	THREE–DIMENSIONAL COORDINATE CONVERSION ALARM	G12 (spatial circular interpolation) was issued during three–dimensional coordinate conversion.
4008	THREE–DIMENSIONAL COORDINATE CONVERSION ALARM	Scaling or coordinate rotation was specified during three–dimensional coordinate conversion.
4010	THREE–DIMENSIONAL TRANSFORM FORMAT ERROR	The G98 command format is invalid.
4011	MATRIX TRANSFORM ERROR	No matrix can be created during G98 processing.
4012	COORDINATE CONVERSION ERROR	Coordinate conversion cannot be performed for G98.
4016	P/S ALARM	The value of arc radius R specified in G33 mode is too large.
4017	P/S ALARM	A G code in group 01 other than G01 was issued in G33 mode. Cutter compensation was specified in G33 mode.
4021	P/S ALARM	A negative value is specified as the length of the nozzle. The nozzle length falls outside the specification range. The total of the second arm length and nozzle length must be a maximum of 500 mm (19.685 inches).
4025	P/S ALARM	The spatial arc or spatial corner rounding command specifies axis movement for an axis other than the first to fifth axes.

Number	Message	Contents
4026	COMMUNICATION ERROR	An overrun, parity, or framing error occurred during read via the interface of the teaching box. The number of the input data bits is incorrect, or the set baud rate or I/O device specification number is illegal.
4027	DR SIGNAL OFF	The I/O device operation ready signal (DR) is off when data is input or output via the interface of the teaching box. The power to the I/O device may be off, or there may be a cable disconnection or a printed circuit board failure.
4028	BUFFER OVERFLOW	During data read via the interface of the teaching box, although a read operation is stopped, input continues even after ten characters have been input. The I/O device or printed circuit board is faulty.
4031	MIRROR AXIS PRM. SETTING ERROR	For the mirror axis under constant optical path control, the following parameters are not set: Bit 0 of parameter No. 7052 = 1, bit 1 of parameter No. 2005 = 0, bit 7 of parameter No. 1819 = 1, bit 0 of parameter No. 6131 = 0, parameter No. 7510 = 2
4032	APPROACH TIME ERROR	Because the value set in parameter No. 15782 is too large, the approach operation feedrate cannot be controlled at the time of beam output. Set a smaller value than the piercing time (parameter No. 15087) in parameter No. 15872.
4033	Z AXIS PRM. ERROR	For the Z–axis, the following settings are not made: Bit 5 of parameter No. 1806 = 1, bit 1 of parameter No. 2224 = 1
4700	PROGRAM ERROR (OT +)	The value specified in the X-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4701	PROGRAM ERROR (OT –)	The value specified in the X-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4702	PROGRAM ERROR (OT +)	The value specified in the Y-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4703	PROGRAM ERROR (OT –)	The value specified in the Y-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
4704	PROGRAM ERROR (OT +)	The value specified in the Z-axis move command exceeded the positive value of stored stroke limit 1. (Advance check)
4705	PROGRAM ERROR (OT –)	The value specified in the Z-axis move command exceeded the negative value of stored stroke limit 1. (Advance check)
5000	ILLEGAL COMMAND CODE	The specified code was incorrect in the high–precision contour control (HPCC) mode.
5003	ILLEGAL PARAMETER (HPCC)	The parameter setting is incorrect.
5004	HPCC NOT READY	High–precision contour control is not ready.
5006	TOO MANY WORD IN ONE BLOCK	The number of words specified in a block exceeded 26 in the HPCC mode.
5007	TOO LARGE DISTANCE	In the HPCC mode, the machine moved beyond the limit.
5009	PARAMETER ZERO (DRY RUN)	The maximum feedrate (parameter No. 1422) or the feedrate in dry run (parameter No. 1410) is 0 in the HPCC model.
5010	END OF RECORD	The end of record (%) was specified.
5011	PARAMETER ZERO(CUT MAX)	The maximum cutting feedrate (parameter No. 1422) is 0.
5012	G05 P10000 ILLEGAL START UP (HPCC)	G05 P10000 has been specified in a mode from which HPCC mode cannot be entered.
5013	HPCC:CRC OFS REMAIN AT CANCEL	G05P0 has been specified in G41/G42 mode or before cancellation axis is not found.
5014	TRACE DATA NOT FOUND	Trace data is not available, preventing transfer from being performed.
5016	ILLEGAL COMBINATION OF M CODE	M codes which belonged to the same group were specified in a block. Alternatively,an M code which must be specified without other M codes in the block was specified in a block with other M codes.
5020	PARAMETER OF RESTART ERROR	The parameter for specifying program restart is not set correctly.

Number	Message	Contents
5046	ILLEGAL PARAMETER (ST.COMP)	An illegal parameter has been specified for straightness compensation. Possible reasons are as follows: 1 There is no axis corresponding to the axis number specified in the move axis or compensation axis parameter. 2 More than 128 pitch error compensation points are not sequentially numbered. 3 The straightness compensation points are not sequentially numbered. 4 A specified straightness compensation point is outside the range between the pitch error compensation points having the maximum positive and negative coordinates. 5 The compensation value specified for each compensation point is too large or too small.
5051	M–NET CODE ERROR	When the chopping function is used, a move command was specified for a chopping axis in chopping mode (during reciprocation between a top dead point and bottom dead point).
5052	M–NET ETX ERROR	“ETX” code is abnormal.
5053	M–NET CONNECT ERROR	Connection time supervision error (parameter No.175)
5054	M–NET RECEIVE ERROR	Boring time supervision error (parameter No.176)
5055	M–NET PRT/FRM ERROR	Vertical parity or framing error detection
5057	M–NET BOARD SYSTEM DOWN	Transmit time–out error (parameter No. 177) ROM parity error CPU interruption detection of not listed above
5059	RADIUS VALUE OUTSIDE ALLOWABLE RANGE	When the center of the arc was specified by I, J, and K for circular interpolation, the length of the radius value exceeded nine digits.
5063	IS NOT PRESET AFTER REF.	This message is output when the position counter has not been preset before the start of plate thickness measurement. This alarm is issued in one of the cases below. 1) When an attempt was made to perform measurement before a reference position had been established. 2) When, after manual reference position return, an attempt was made to start measurement without first setting the position counter.
5064	DIFFERRENT AXIS UNIT (IS–B, IS–C)	Circular interpolation was specified for a plane formed by axes using different increment systems.
5065	DIFFERRENT AXIS UNIT (PMC AXIS)	In PMC–based axis control, axes using different increment systems are specified for the same DI/DO group. Modify parameter No. 8010.
5068	G31 P90 FORMAT ERROR	No axis is specified for movement. Two or more axes were specified for movement.
5073	NO DECIMAL POINT	A decimal point is not specified for a command for which a decimal point must be specified.
5074	ADDRESS DUPLICATION ERROR	The same address appears more than once in a block. Alternatively, a block contains two or more G codes belonging to the same group.
5082	DATA SERVER ERROR	Details are displayed on the data server message screen.
5110	IMPROPER G–CODE (G05.1 G1 MODE)	An illegal G code was specified in AI contour control mode.
5111	IMPROPER MODAL G–CODE (G05.1 G1)	An illegal G code is left modal when AI contour control mode was specified.
5112	G08 CAN NOT BE COMMANDED (G05.1 G1)	Look–ahead control (G08) was specified in AI contour control mode.
5113	CAN NOT ERROR IN MDI MODE (G05.1)	AI contour control (G05.1) was specified in MDI mode.
5114	NOT STOP POSITION (G05.1 Q1)	At the time of restart after manual intervention, the coordinates at which the manual intervention occurred have not been restored.
5134	FSSB : OPEN READY TIME OUT	Initialization did not place FSSB in the open ready state.
5135	FSSB : ERROR MODE	FSSB has entered error mode.

Number	Message	Contents
5136	FSSB : NUMBER OF AMPS IS SMALL	In comparison with the number of controlled axes, the number of amplifiers recognized by FSSB is not enough.
5137	FSSB : CONFIGURATION ERROR	FSSB detected a configuration error.
5138	FSSB : AXIS SETTING NOT COMPLETE	In automatic setting mode, axis setting has not been made yet. Perform axis setting on the FSSB setting screen.
5139	FSSB : ERROR	Servo initialization did not terminate normally. The optical cable may be defective, or there may be an error in connection to the amplifier or another module. Check the optical cable and the connection status.
5156	ILLEGAL AXIS OPERATION (SHPCC)	In simple high–precision contour control (SHPCC) mode, the controlled axis selection signal (PMC axis control) changes. In SHPCC mode, the simple synchronous axis selection signal changes.
5157	PARAMETER ZERO (AICC)	Zero is set in the parameter for the maximum cutting feedrate (parameter No. 1422 or 1432). Zero is set in the parameter for the acceleration/deceleration before interpolation (parameter No. 1770 or 1771). Set the parameter correctly.
5196	ILLEGAL OPERATION (HPCC)	Detach operation was performed in HPCC mode. (If detach operation is performed in HPCC mode, this alarm is issued after the currently executed block terminates.)
5197	FSSB : OPEN TIME OUT	The CNC permitted FSSB to open, but FSSB was not opened.
5198	FSSB : ID DATA NOT READ	Temporary assignment failed, so amplifier initial ID information could not be read.
5199	FINE TORQUE SENSING PARAMETER	A parameter related to the fine torque sensing function is illegal. · The storage interval is invalid. · An invalid axis number is set as the target axis. Correct the parameter.
5212	SCREEN COPY : PARAMETER ERROR	There is a parameter setting error. Check that 4 is set as the I/O channel.
5213	SCREEN COPY : COMMUNICATION ERROR	The memory card cannot be used. Check the memory card. (Check whether the memory card is write–protected or defective.)
5214	SCREEN COPY : DATA TRANSFER ERROR	Data transfer to the memory card failed. Check whether the memory card space is insufficient and whether the memory card was removed during data transfer.
5218	ILLEGAL PARAMETER (INCL. COMP)	There is an inclination compensation parameter setting error. Cause: 1. The number of pitch error compensation points between the negative (–) end and positive (+) end exceeds 128. 2. The relationship in magnitude among the inclination compensation point numbers is incorrect. 3. An inclination compensation point is not located between the negative (–) end and positive (+) end of the pitch error compensation points. 4. The amount of compensation per compensation point is too large or too small. Correct the parameter.
5220	REFERENCE POINT ADJUSTMENT MODE	A parameter for automatically set a reference position is set. (Bit 2 of parameter No. 1819 = 1) Perform automatic setting. (Position the machine at the reference position manually, then perform manual reference position return.) Supplementary: Automatic setting sets bit 2 of parameter No. 1819 to 0.

Number	Message	Contents
5222	SRAM CORRECTABLE ERROR	The SRAM correctable error cannot be corrected. Cause: A memory problem occurred during memory initialization. Action: Replace the master printed circuit board (SRAM module).
5227	FILE NOT FOUND	A specified file is not found during communication with the built–in Handy File.
5228	SAME NAME USED	There are duplicate file names in the built–in Handy File.
5229	WRITE PROTECTED	A floppy disk in the built–in Handy File is write protected.
5231	TOO MANY FILES	The number of files exceeds the limit during communication with the built–in Handy File.
5232	DATA OVER–FLOW	There is not enough floppy disk space in the built–in Handy File.
5235	COMMUNICATION ERROR	A communication error occurred during communication with the built–in Handy File.
5237	READ ERROR	A floppy disk in the built–in Handy File cannot be read from. The floppy disk may be defective, or the head may be dirty. Alternatively, the Handy File is defective.
5238	WRITE ERROR	A floppy disk in the built–in Handy File cannot be written to. The floppy disk may be defective, or the head may be dirty. Alternatively, the Handy File is defective.
5242	ILLEGAL AXIS NUMBER (M series)	The axis number of the synchronous master axis or slave axis is incorrect. (This alarm is issued when flexible synchronization is turned on.) Alternatively, the axis number of the slave axis is smaller than that of the master axis.
5257	G41/G42 NOT ALLOWED IN MDI MODE	G41/G42 (cutter compensation C) was specified in MDI mode. (Depending on the setting of bit 4 of parameter No. 5008)
5302	ILLEGAL COMMAND IN G68 MODE	A command to set the coordinate system is specified in the coordinate system rotation mode.
5303	TOUCH PANEL ERROR	A touch panel error occurred. Cause: 1. The touch panel is kept pressed. 2. The touch panel was pressed when power was turned on. Remove the above causes, and turn on the power again.
5306	MODE CHANGE ERROR	In a one–touch macro call, mode switching at the time of activation is not performed correctly.
5307	INTERNAL DATA OVER FLOW	In the following function, internal data exceeds the allowable range. 1) Improvement of the rotation axis feedrate
5311	FSSB:ILLEGAL CONNECTION	A connection related to FSSB is illegal. This alarm is issued when either of the following is found: 1. Two axes having adjacent servo axis numbers (parameter No. 1023), odd number and even number, are assigned to amplifiers to which different FSSB systems are connected. 2. The system does not satisfy the requirements for performing HRV control, and use of two pulse modules connected to different FSSB systems having different FSSB current control cycles is specified.

*AICC = AI contour control

2) Background edit alarm

Number	Message	Contents
???	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit. (P/S alarm No. 070, 071, 072, 073, 074, 085 to 087) Modify the program.
140	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 to 8).
301	APC alarm: nth–axis communication	nth–axis (n=1 to 8) 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 to 8) 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 to 8) 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 to 8) 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 to 8) APC pulse error alarm. APC alarm.APC or cable may be faulty.
306	APC alarm: nth–axis battery voltage 0	nth–axis (n=1 to 8) 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 to 8) 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 to 8) 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 : n AXIS ZRN IMPOSSIBL	An attempt was made to perform reference position return without rotating the motor through one or more turns. Rotate the motor through one or more turns, turn off the power then on again, then perform reference position return.

4) Industosyn alarms

No.	Message	Contents
330	INDUCTOSYN:DATA ALARM	The absolute–position data (offset data) from Inductosyn cannot be detected.
331	INDUCTOSYN:ILLEGAL PRM	Parameter No. 1874, 1875, or 1876 is set to 0.

5) 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	Description
360	n AXIS : ABNORMAL CHECKSUM (INT)	A checksum error occurred in the built–in pulse coder.
361	n AXIS : ABNORMAL PHASE DATA (INT)	A phase data error occurred in the built–in pulse coder.
362	n AXIS : ABNORMAL REV.DATA (INT)	A rotation speed count error occurred in the built–in pulse coder.
363	n AXIS : ABNORMAL CLOCK (INT)	A clock error occurred in the built–in pulse coder.
364	n AXIS : SOFT PHASE ALARM (INT)	The digital servo software detected invalid data in the built–in pulse coder.
365	n AXIS : BROKEN LED (INT)	An LED error occurred in the built–in pulse coder.
366	n AXIS : PULSE MISS (INT)	A pulse error occurred in the built–in pulse coder.
367	n AXIS : COUNT MISS (INT)	A count error occurred in the built–in pulse coder.
368	n AXIS : SERIAL DATA ERROR (INT)	Communication data from the built–in pulse coder cannot be received.
369	n AXIS : DATA TRANS. ERROR (INT)	A CRC or stop bit error occurred in the communication data being received from the built–in pulse coder.
380	n AXIS : BROKEN LED (EXT)	The separate detector is erroneous.
381	n AXIS : ABNORMAL PHASE (EXT LIN)	A phase data error occurred in the separate linear scale.
382	n AXIS : COUNT MISS (EXT)	A pulse error occurred in the separate detector.
383	n AXIS : PULSE MISS (EXT)	A count error occurred in the separate detector.
384	n AXIS : SOFT PHASE ALARM (EXT)	The digital servo software detected invalid data in the separate detector.
385	n AXIS : SERIAL DATA ERROR (EXT)	Communication data from the separate detector cannot be received.
386	n AXIS : DATA TRANS. ERROR (EXT)	A CRC or stop bit error occurred in the communication data being received from the separate detector.
387	n AXIS : ABNORMAL ENCODER (EXT)	An error occurs in the separate detector. For details, contact the manufacturer of the scale.

- **The details of serial pulse coder alarm No.350**

The details of serial pulse coder alarm No. 350 (pulse coder alarm) are displayed in the diagnosis display (No. 202) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
202		CSA	BLA	PHA	PCA	BZA	CKA	SPH

SPH : The serial pulse coder or feedback cable is defective. Replace the serial pulse coder or cable.

CKA: The serial pulse coder is defective. Replace it.

BZA : The pulse coder was supplied with power for the first time. Make sure that the batteries are connected.
Turn the power off, then turn it on again and perform a reference position return. This alarm has nothing to do with alarm No. 350 (serial pulse coder alarm).

PCA : The serial pulse coder is defective. Replace it.

PHA : The serial pulse coder or feedback cable is defective. replace the serial pulse coder or cable.

BLA : The battery voltage is low. Replace the batteries. This alarm has nothing to do with alarm No. 350 (serial pulse coder alarm).

CSA : The serial pulse coder is defective. Replace it.

- **The details of serial pulse coder alarm No.351**

The details of serial pulse coder alarm No. 351 (communication alarm) are displayed in the diagnosis display (No. 203) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
203	DTE	CRC	STB	PRM				

PRM: An invalid parameter was found. Alarm No. 417 (invalid servo parameter) is also issued.

STB : the serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective.
Replace the pulse coder, feedback cable, or NC–axis board.

CRC : The serial pulse coder encountered a communication error. The pulse coder, feedback cable, or feedback receiver circuit is defective. Replace the pulse coder, feedback cable, or NC–axis board.

DTE : The serial pulse coder encountered a communication error. The pulse coder, feedbak cable, or feedback receiver circuit is defective. Replace the pulse coder, feedback cable, or NC–axis board

6) Servo alarms

Number	Message	Contents
401	SERVO ALARM: n–TH AXIS VRDY OFF	The n–th axis (axis 1 to 8) servo amplifier READY signal (DRDY) went off. Refer to procedure of trouble shooting.
402	SERVO ALARM: SV CARD NOT EXIST	The axis control card is not provided.
403	SERVO ALARM: CARD/SOFT MISMATCH	The combination of the axis control card and servo software is illegal. The possible causes are as follows: · A correct axis control card is not provided. · Correct servo software is not installed on flash memory.
404	SERVO ALARM: n–TH AXIS VRDY ON	Even though the n–th axis (axis 1 to 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.
407	SERVO ALARM: EXCESS ERROR	The following failure occurred during simplified synchronization control. 1) The difference in synchronous axis position deviation exceeded the value set in parameter No. 8314.
409	SERVO ALARM: n AXIS TORQUE ALM	Abnormal servo motor load has been detected.
410	SERVO ALARM: n–TH AXIS – EXCESS ERROR	Either of the following failures occurred. 1) The position deviation when the n axis stops exceeded the value set in parameter No. 1829. 2) The amount of compensation for synchronization exceeded the value set in parameter No. 8325 during simplified synchronization control. This alarm can be generated only for the slave axis.
411	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1 to 8) moves is larger than the set value.
413	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis (axis 1 to 8) exceeded $\pm 2^{31}$ power. This error usually occurs as the result of an improperly set parameters.
415	SERVO ALARM: n–TH AXIS – EXCESS SHIFT	A speed higher than 524288000 units/s was attempted to be set in the n–th axis (axis 1 to 8). This error occurs as the result of improperly set CMR.

Number	Message	Contents
417	SERVO ALARM: n–TH AXIS – PARAMETER INCORRECT	This alarm occurs when the n–th axis (axis 1 to 8) 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) contains a value out of the range from 1 to the number of axes, or an isolated value (for example, 4 not prceded by 3).was set in parameter No. 1023 (servo axisnumber). 7) A parameter is set incorrectly for PMC axis torque control. (The torque constant parameter is set to 0.)
420	SERVO ALARM: n AXIS SYNC TORQUE (M series)	During simple synchronous control, the difference between the torque commands for the master and slave axes exceeded the value set in parameter No. 2031.
421	SERVO ALARM: n AXIS EXCESS ER (D)	The difference between the errors in the semi–closed loop and closed loop has become excessive during dual position feedback. Check the values of the dual position conversion coefficients in parameters No. 2078 and 2079.
422	SERVO ALARM: n AXIS	In torque control of PMC axis control, a specified allowable speed has been exceeded.
423	SERVO ALARM: n AXIS	In torque control of PMC axis control, the parameter–set allowable cumulative travel distance has been exceeded.
430	n AXIS : SV. MOTOR OVERHEAT	A servo motor overheat occurred.
431	n AXIS : CNV. OVERLOAD	1) αiPS: Overheat occurred. 2) β series SVU: Overheat occurred.
432	n AXIS : CNV. LOWVOLT CON./ POWFAULT	1) αiPS: Phase missing occurred in the input voltage. 2) αiPSR: The control power supply voltage has dropped. 3) α series SVU: The control power supply voltage has dropped.
433	n AXIS : CNV. LOW VOLT DC LINK	1) αiPS: The DC link voltage has dropped. 2) αiPSR: The DC link voltage has dropped. 3) α series SVU: The DC link voltage has dropped. 4) β series SVU: The DC link voltage has dropped.
434	n AXIS : INV. LOW VOLT CONTROL	αiSV: The control power supply voltage has dropped.
435	n AXIS : INV. LOW VOLT DC LINK	αiSV: The DC link voltage has dropped.
436	n AXIS : SOFTTHERMAL (OVC)	The digital servo software detected the soft thermal state (OVC).
437	n AXIS : CNV. OVERCURRENT POWER	αiPS: Overcurrent flowed into the input circuit.
438	n AXIS : INV. ABNORMAL CURRENT	1) αiSV: The motor current is too high. 2) α series SVU: The motor current is too high. 3) β series SVU: The motor current is too high.
439	n AXIS : CNV. OVERVOLT POWER	1) αiPS: The DC link voltage is too high. 2) αiPSR: The DC link voltage is too high. 3) α series SVU: The C link voltage is too high. 4) β series SVU: The link voltage is too high.
440	n AXIS : CNV. EX DECELERATION POW.	1) αiPSR: The regenerative discharge amount is too large. 2) α series SVU: The regenerative discharge amount is too large. Alternatively, the regenerative discharge circuit is abnormal.

Number	Message	Contents
441	n AXIS : ABNORMAL CURRENT OFFSET	The digital servo software detected an abnormality in the motor current detection circuit.
442	n AXIS : CNV. CHARGE FAULT/INV. DB	1) α*i*PS: The spare discharge circuit of the DC link is abnormal. 2) α*i*PSR: The spare discharge circuit of the DC link is abnormal. 3) α series SVU: The dynamic brake circuit is abnormal.
443	n AXIS : CNV. COOLING FAN FAILURE	1) α*i*PS: The internal stirring fan failed. 2) α*i*PSR: The internal stirring fan failed. 3) β series SVU: The internal stirring fan failed.
444	n AXIS : INV. COOLING FAN FAILURE	α*i*SV: The internal stirring fan failed.
445	n AXIS : SOFT DISCONNECT ALARM	The digital servo software detected a broken wire in the pulse coder.
446	n AXIS : HARD DISCONNECT ALARM	A broken wire in the built–in pulse coder was detected by hardware.
447	n AXIS : HARD DISCONNECT (EXT)	A broken wire in the separate detector was detected by hardware.
448	n AXIS : UNMATCHED FEEDBACK ALARM	The sign of feedback data from the built–in pulse coder differs from that of feedback data from the separate detector.
449	n AXIS : INV. IPM ALARM	1) α*i*SV: IPM (intelligent power module) detected an alarm. 2) α series SVU: IPM (intelligent power module) detected an alarm.
453	n AXIS : SPC SOFT DISCONNECT ALARM	Software disconnection alarm of the α pulse coder. Turn off the power to the CNC, then remove and insert the pulse coder cable. If this alarm is issued again, replace the pulse coder.
456	ILLEGAL CURRENT LOOP	The current control cycle settings (parameter No. 2004, bit 0 of parameter No. 2003, and bit 0 of parameter No. 2013) are incorrect. Possible problems are as follows. – For the two axes whose servo axis numbers (settings of parameter No. 1023) are an odd number followed by an even number (a pair of axes 1 and 2 or axes 5 and 6, for example), a different current control cycle is set for each of the axes. – The requirements for slaves needed for the set current control cycle, including the number, type, and connection method of them, are not satisfied.
457	ILLEGAL HI HRV (250US)	Use of high–speed HRV is specified although the current control cycle is 200 μs.
458	CURRENT LOOP ERROR	The current control cycle setting does not match the actual current control cycle.
459	HI HRV SETTING ERROR	For the two axes whose servo axis numbers (settings of parameter No. 1023) are an odd number followed by an even number (a pair of axes 1 and 2 or axes 5 and 6, for example), the SVM for one of the axes supports high–speed HRV control but the SVM for the other does not. Refer to the SVM specification.
460	n AXIS : FSSB DISCONNECT	FSSB communication was disconnected suddenly. The possible causes are as follows: 1) The FSSB communication cable was disconnected or broken. 2) The power to the amplifier was turned off suddenly. 3) A low–voltage alarm was issued by the amplifier.
461	n AXIS : ILLEGAL AMP INTERFACE	The axes of the 2–axis amplifier were assigned to the fast type interface.
462	n AXIS : SEND CNC DATA FAILED	Because of an FSSB communication error, a slave could not receive correct data.
463	n AXIS : SEND SLAVE DATA FAILED	Because of an FSSB communication error, the servo system could not receive correct data.
464	n AXIS : WRITE ID DATA FAILED	An attempt was made to write maintenance information on the amplifier maintenance screen, but it failed.

Number	Message	Contents
465	n AXIS : READ ID DATA FAILED	At power–up, amplifier initial ID information could not be read.
466	n AXIS : MOTOR/AMP COMBINA-TION	The maximum current rating for the amplifier does not match that for the motor.
467	n AXIS : ILLEGAL SETTING OF AXIS	The servo function for the following has not been enabled when an axis occupying a single DSP (corresponding to two ordinary axes) is specified on the axis setting screen. 1. Learning control (bit 5 of parameter No. 2008 = 1) 2. High–speed current loop (bit 0 of parameter No. 2004 = 1) 3. High–speed interface axis (bit 4 of parameter No. 2005 = 1)
468	HI HRV SETTING ERROR (AMP)	Use of high–speed HRV is specified for a controlled axis of an amplifier which does not support high–speed HRV.

- **Details of servo alarm No.414**

The details of servo alarm No. 414 are displayed in the diagnosis display (No. 200 and No.204) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
200	OVL	LV	OVC	HCA	HVA	DCA	FBA	OFA

OFA : An overflow alarm is being generated inside of digital servo.
FBA : A disconnection alarm is being generated.
(This bit causes servo alarm No.416.The details are indicated in diagnostic data No. 201)
DCA : A regenerative discharge circuit alarm is being generated in servo amp.
Check LED.
HVA : An overvoltage alarm is being generated in servo amp.
Check LED.
HCA : An abnormal current alarm is being generated in servo amp.
Check LED.
OVC : A overcurrent alarm is being generated inside of digital servo.
LV : A low voltage alarm is being generated in servo amp.
Check LED.
OVL : An overload alarm is being generated.
(This bit causes servo alarm No. 400. The details are indicated in diagnostic data No.201).

	#7	#6	#5	#4	#3	#2	#1	#0
204		OFS	MCC	LDA	PMS			

PMS : A feedback pulse error has occured because the feedback cable is defective.
LDA : The LED indicates that serial pulse coder C is defective
MCC : A magnetic contactor contact in the servo amplifier has welded.
OFS : A current conversion error has occured in the digital servo.

- **Details of servo alarms No. 400 and No.416**

The details of servo alarms No. 400 and No. 416 are displayed in the diagnosis display (No. 201) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
201	ALD			EXP				

When OVL equal 1 in diagnostic data No.200
(servo alarm No. 400 is being generated):

ALD 0 : Motor overheating
1 : Amplifier overheating

When FBA equal 1 in diagnostic data No.200
(servo alarm No. 416 is being generated):

ALD	EXP	Alarm details
1	0	Built–in pulse coder disconnection (hardware)
1	1	Separately installed pulse coder disconnection (hardware)
0	0	Pulse coder is not connected due to software.

7) Over travel alarms

Number	Message	Contents
500	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1 to 8) + side stored stroke limit I. (Parameter No.1320 or 1326 **NOTE**)
501	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1 to 8) – side stored stroke limit I. (Parameter No.1321 or 1327 **NOTE**)
502	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1 to 8) + side stored stroke limit II. (Parameter No.1322)
503	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1 to 8) – side stored stroke limit II. (Parameter No.1323)
504	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1 to 8) – side stored stroke limit III. (Parameter No.1324)
505	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1 to 8) – side stored stroke limit III. (Parameter No.1325)
506	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1 to 8) + side hardware OT.
507	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1 to 8) – side hardware OT.
510	OVER TRAVEL : +n	A stroke limit check, made before starting movement, found that the end point of a block falls within the plus (+) side inhibited area along the n–axis defined by a stroke limit. Correct the program.
511	OVER TRAVEL : –n	A stroke limit check, made before starting movement, found that the end point of a block falls within the minus (–) side inhibited area along the N–axis defined by a stroke limit. Correct the program.

NOTE
Parameters 1326 and 1327 are effective when EXLM (stroke limit switch signal) is on.

8) 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.

9) Laser alarms

Number	Message	Contents
4050	CONTOUR DATA ERROR	The value set as contour data is beyond the allowable range.
4051	ASSIST GAS DATA ERR.	The value specified for assist gas is beyond the allowable range.
4052	TRACE DATA ERROR	The value specified for gap control is beyond the allowable range.
4053	PIERCING DATA ERROR	The piercing data value is beyond the allowable range.

Number	Message	Contents
4054	E CODE DATA ERROR	The specified E code value is beyond the allowable range.
4055	E CODE NOT COMMANDED	Because no E code is specified, a laser output calculation cannot be performed.
4056	OPT. PATH NOT SET	In constant optical path control, the mirror block position cannot be calculated with the current positions of the axes. Perform reference position return.
4060	TRACE ERROR EXCESS	In gap control, the nozzle is brought too close to the workpiece.
4061	A/D CONVERTER–1	A/D converter 1 (for reading power and gas pressure) on the laser interface board does not perform conversion normally.
4062	A/D CONVERTER–2	A/D converter 2 (for reading voltage and current) on the laser interface board does not perform conversion normally.
4063	RF POWER SUPPLY	The R/F power supply in the laser oscillator becomes abnormal.
4065	SHUTTER ACTION	The mechanical shutter in the laser oscillator does not operate normally.
4066	DISCHARGING	At the start of discharging of the oscillator, discharging could not be performed normally.
4067	LASER CABINET OH	The temperature in the oscillator has exceeded a specified level.
4068	BEAM REFLECTION	The beam reflection from the workpiece has exceeded a specified level.
4069	LASER IF PCB	The laser interface board is not connected. Alternatively, the laser interface board is abnormal.
4070	CHILLER NOT READY	No ready signal has been received from the chiller.
4071	ASSIST GAS NOT READY	No ready signal has been received from the assist gas supply unit.
4072	CHILL FLOW	The flow rate of water supplied to the oscillator is insufficient.
4073	LASER GAS PRES.	The laser gas pressure supplied to the oscillator is too low.
4074	OXIGEN DENSITY	The oxygen concentration in the oscillator exceeds a specified level.
4075	CHILL TEMP.	There is dew condensation in the oscillator. The water temperature is too low.
4076	LASER POWER DOWN	The laser power lowered from the average value of commands by a specified value or more.
4077	ABSORBER TEMP.	The temperature of the absorber has exceeded a specified value.
4078	LASER TUBE PRES.	The laser gas pressure has exceeded the range of (100 (1 = 13Pa) of the set value.
4079	PUSH RESET KEY	The emergency stop button was pressed. Release the emergency stop state, then press the reset key. This message is issued when an emergency stop state is entered in the stages from LRDY to LSTR.
4080	LASER TUBE EXHAUST	At RUN–ON, a specified gas pressure cannot be obtained by exhaustion within 10 minutes.
4081	GAS PRES. CONTROL	The laser gas pressure is not within the range of (20 (1 = 13 (P) of the set value in 45 seconds after the start of gas pressure control.
4082	TUBE PRES. SENSOR	The value from the negative pressure sensor is 0. The negative sensor may be faulty, or there may be a cable disconnection.
4083	SHUTTER NOT OPEN	Although the shutter was not open, a laser output command was issued.
4085	MIRROR CLEANING	Because the output compensation coefficient has exceeded a specified value, clean the mirrors.
4087	SHUTTER OH	The shutter temperature has exceeded a specified level.
4088	LASER VOLTAGE DOWN	The discharging tube voltage has lowered below a specified level.
4089	ASSIST GAS NO SELECT	Although assist gas was not active, a laser output command was issued.
4090	LASER NOT GENERATE	Although the oscillator did not generate oscillation, a laser output command was issued.
4091	INVERTER	The inverter for the blower is abnormal.
4093	BLOWER GREASE	Because vibration in the turbo blower has increased, change grease.

Number	Message	Contents
4094	VANE PUMP	The vacuum pump is abnormal. A thermal trip occurred in the magnetic contactor because of overload on the vacuum pump.
4095	A/D CONVERTER–3	A/D converter 3 (for reading gap displacement) does not operate normally. Check the analog input board in the CNC.
4096	UNDER TRACING	With the gap control axis selection function, an attempt was made to switch between axis selection signals during gap control.
4097	TRACE AXIS SELECT	With the gap control axis selection function, no gap control axis selection signal is selected, or more than one gap control axis selection signal is selected.
4098	TRACE AXIS & PLANE	With the gap control axis selection function, plane selection is specified for the gap controlled axis.
4099	GAS PRES. NOT REACH	When the base discharge state is entered, the laser gas pressure has not reached a specified pressure level.
4100	INVERTER 1	Blower inverter 1 becomes abnormal.
4101	OUT OF FREQUENCY 1	The inverter frequency has not reached a specified level within 120 seconds after turbo blower 1 was started.
4102	OUT OF VIBRATION1	Vibration in turbo blower 1 was detected.
4103	T.B NOT STOPPING 1	The rotation speed of turbo blower 1 has not lowered to a specified level within 120 seconds after RUN–OFF.
4104	TURBO OVER CURRENT 1	An overcurrent alarm was issued in turbo blower 1.
4105	TURBO TEMP. 1	The temperature of turbo blower 1 has exceeded a specified level.
4106	BLOWER OIL SHORTAGE	Lubricant in turbo blower is insufficient.
4107	MIRROR NOT INSTALLED	The folding mirror is not mounted.
4110	INVERTER 2	Blower inverter 2 becomes abnormal.
4111	OUT OF FREQUENCY 2	The inverter frequency has not reached a specified level within 120 seconds after turbo blower 2 was started.
4112	OUT OF VIBRATION 2	Vibration in turbo blower 2 was detected.
4113	T. B NOT STOPPING 2	The rotation speed of turbo blower 2 has not lowered to a specified level within 120 seconds after RUN–OFF.
4114	TURBO OVER CURRENT 2	An overcurrent alarm was issued in turbo blower 2.
4115	TURBO TEMP. 2	The temperature of turbo blower 2 has exceeded a specified level.
4116	BLOWER OIL TIM LIMIT	Turbo blower oil has been used for a longer time than the replacement period. Because there is a possibility that the turbo blower is damaged, change oil as soon as possible.
4124	OVER COOL	Even when warm–up operation is performed, the temperature of the oscillator cannot increase to a specified level or higher.
4129	SHTR OPEN IN DISCHRG	The shutter was opened during discharging in the oscillator.
4130	STEP TIME NOT SET	The step time for high–speed piercing is not set.
4131	INCORRECT STEP DISTANCE SETTING	In step control, the rest of travel distance becomes shorter than the ramp–down distance before the ramp–up distance is completed. Alternatively, the ramp–up distance cannot be completed within one block.
4132	PARAMETER WAS CHANGED.	The setting of parameter No. 15223, 15242, or 15243 was changed.
4133	TRACE DEC. POSITION	The gap control displacement observed at the start of gap control is smaller than the deceleration start deviation (parameter No. 15848). Sufficient deceleration for the gap controlled axis may fail, which can cause a collision with the workpiece. This alarm is not issued if bit 2 of parameter No. 15840 is set to 1.
4134	PRG. ERROR FOR TRACE	A move command for the gap controlled axis was issued from the machining program during gap control.
4135	PULSE PFB DATA ERROR	Because the pulse power feedback table data read through a memory card or reader/punch interface was incorrect, the read operation was stopped halfway.
4136	INTERNAL PRESSURE TOO HIGH	This alarm is issued when the pressure inside the laser gas mixer becomes too high (1MPa).

Number	Message	Contents
4137	MIXER POWER FAILURE	This alarm is issued when a power failure occurs on the relay printed circuit board in the laser gas mixer.
4138	INSUFFICIENT DEGREE OF VACUUM	This alarm is issued when a specified degree of vacuum cannot be achieved within a specified time during vacuum operation of the laser gas mixer.
4139	MIXER PRESSURE SWITCH ERROR	This alarm is issued when the laser gas supply pressure in the oscillator is lowered, but the tank supply pressure of the laser gas mixer is not lowered.

10) System alarms

(These alarms cannot be reset with reset key.)

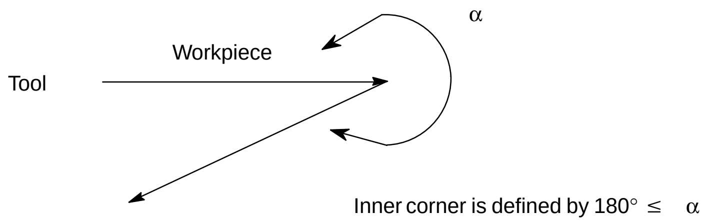
Number	Message	Contents
900	ROM PARITY	A parity error occurred in the CNC, macro, or servo ROM. Correct the contents of the flash ROM having the displayed number.
910	SRAM PARITY : (BYTE 0)	A RAM parity error occurred in the part program storage RAM. Clear the RAM, or replace the SRAM module or motherboard. Subsequently, re–set the parameters and all other data.
911	SRAM PARITY : (BYTE 1)	
912	DRAM PARITY : (BYTE 0)	A RAM parity error occurred in the DRAM module. Replace the DRAM module.
913	DRAM PARITY : (BYTE 1)	
914	DRAM PARITY : (BYTE 2)	
915	DRAM PARITY : (BYTE 3)	
916	DRAM PARITY : (BYTE 4)	
917	DRAM PARITY : (BYTE 5)	
918	DRAM PARITY : (BYTE 6)	
919	DRAM PARITY : (BYTE 7)	
920	SERVO ALARM (1–4 AXIS)	Servo alarm (first to fourth axis). A watchdog alarm condition occurred, or a RAM parity error occurred in the axis control card. Replace the axis control card.
921	SERVO ALARM (5–8 AXIS)	Servo alarm (fifth to eighth axis). A watchdog alarm condition occurred, or a RAM parity error occurred in the axis control card. Replace the axis control card.
926	FSSB ALARM	FSSB alarm. Replace the axis control card.
930	CPU INTERRUPT	CPU error (abnormal interrupt). The motherboard or CPU card may be faulty.
935	SRAM ECC ERROR	An error occurred in RAM for part program storage. Action: Replace the master printed circuit board (SRAM module), perform all–clear operation, and set all parameter and other data again.
950	PMC SYSTEM ALARM	An error occurred in the PMC. The PMC control circuit on the motherboard may be faulty.
951	PMC WATCH DOG ALARM	An error occurred in the PMC. (Watchdog alarm) The motherboard may be faulty.
972	NMI OCCURRED IN OTHER MODULE	An NMI occurred on a board other than the motherboard. The option board may be faulty.
973	NON MASK INTERRUPT	An NMI occurred as a result of an unknown cause.
974	F–BUS ERROR	A bus error occurred on the FANUC bus. The motherboard or option board may be faulty.
975	BUS ERROR	A bus error occurred on the motherboard. The motherboard may be faulty.
976	L–BUS ERROR	A bus error occurred on the local bus. The motherboard may be faulty.

H GLOSSARY

Term	Description

[A]

Term	Description
Absolute linear scale	Detector for an absolute position on a straight line.
Absolute position detector	Detector that indicates the absolute coordinates of a machine element, relative to a selected origin.
Absolute programming	Method of programming the coordinates of a tool movement end point.
Absolute pulse coder	Rotary absolute position detector.
Absolute value	Distance or angle from the origin of a coordinate system.
Actual cutting feedrate display	Display of a tool feedrate per minute.
Address	Alphabetic character that defines the use to which the number that follows it is applied (ie x axis command).
Alarm	Error detected by the CNC, such as a program error, operator error, or hardware failure.
Alarm history display	Storing alarms detected by the CNC and displaying them on the screen.
Angular axis control	Controlling the movement of each of two controlled axes mounted at an angle other than a right angle. Programming is done in rectangular coordinates.
Approach forbidden area	That part of a safety zone which a tool is forbidden to enter.
Argument specification	Assigning an actual value to a variable used in a custom macro program to be called.
ASCII code	Information exchange code complying with the applicable ANSI standard. Used in numerical control.
Assist gas	Gas provided to the workpiece during laser processing. Oxygen, inert gas, or other gas is used depending on the workpiece material and the processing method.
Assist gas control	Series of control relating to assist gas output, including switching of the gas type and control of the gas pressure. The optimum gas can be selected according to the material of the workpiece and the method of processing.
Assist gas pressure override	Manual control in which the operator can temporarily change the assist gas pressure during operation. This function is used to create the same figure with different materials or stock thicknesses.
Attitude control A	Three–dimensional control applied to the attitude of a processing head for which the nozzle tip position is fixed (zero offset type).
Attitude control B	Three–dimensional control applied to the attitude of an offset–type processing head.
Automatic acceleration/ deceleration	Applying acceleration/deceleration automatically when the tool starts or stops moving, in order to reduce mechanical stress imposed on the machine.
Automatic corner deceleration	Changing the cutting feedrate for machining a corner according to a difference in the corner angle between machining blocks or a difference in the cutting feedrate between axes.
Automatic corner override	Overriding the cutting feedrate for an inner corner and changing the cutting feedrate for an inner arc.
Automatic insertion of sequence number	Automatically inserting a sequence number into each block in EDIT mode during program creation based on manual data input.
Automatic operation	Operation based on a program.

Term	Description
Automatic override for inner corner	Automatically overriding a cutting feedrate at each end of an inner corner, produced based on a tool path that has been subjected to cutter compensation. 
Automatic reference position return	Automatically feeding a specified axis to a reference position using a program command.
Automatic tool length measurement	Issuing an automatic measurement command to the CNC to move the tool to the measurement position, thereby allowing the CNC to automatically calculate the tool offset value.
Automatic tool offset	Giving an automatic measurement command to the CNC and moving the tool to a measurement position, thereby letting the CNC automatically measure tool offset values.
Auxiliary function lock	Disabling a specified M, S, or T function.
Axis control function	Generic name for control functions for controlled axes.
Axis interchange	Switching the correspondence between a specified axis movement command and the machine axis that actually moves.
Axis name	Name given to an axis controlled by the CNC or PMC.
Axis number	Number used to associate an axis name used in programming with the recognition number (controlled axis number) of the CNC control section and the recognition number (servo axis number) of a machine drive motor.
Axis recomposition (Two–path control function)	Two–path synchronous, composite, and superposition control.

[B]

B code	Coded number, following the B address, that specifies the second auxiliary function or index table indexing.
B–axis control function	Drilling or boring performed using an axis (B–axis) that operates independently of the two basic axes (X and Z) of a lathe.
Background editing	Editing a program during the execution of another program.
Background graphic display	Drawing a machining path specified by a program during the execution of another program.
Backlash compensation	Compensation for mechanical backlash.
Balace cut	Machining a thin workpiece by cutting it from both sides in order to prevent the workpiece from deforming, thereby achieving a high level precision.
Base point	Start point of a pattern function program, such as the position of the tool when a pattern function is specified or the position that is identified with coordinates specified using a base point command.
Base point command	Specifying the coordinates of a base point for the pattern function.
Basic controlled axes	Controlled axis having a name fixed by the CNC.
Cutter compensation C	Compensating the figure by displacing the beam from the programmed beam path by the radius of the actual beam, in the direction perpendicular to the beam path.
Cutter compensation memory	Number of cutter compensation values which can be stored in CNC memory.
Bending compensation	Compensating the position of a hole for displacement due to the workpiece being bent.
Block	One of the command units constituting a program.

Term	Description
Block restart	Resuming automatic operation from the start, or an intermediate point, of a block if automatic operation has been interrupted in that block due, for example, to tool breakage.
Buffering	Standby state set up before a command is executed.

[C]

C–axis control	Controlling a tool angle using a C command.
C–axis synchronous control	Using two motors to synchronously control the punch and die of a tool under C–axis control.
Calling a subprogram stored in external memory	Calling and executing a subprogram from an external input/output device such as a floppy cassette or program File Mate during memory operation.
Canned cycle	Set of predefined sequences prepared for boring, drilling, and/or tapping.
Chamfering	Obliquely cutting an inner or outer corner of a workpiece.
Changing workpiece coordinate system	Relocating the origin of a workpiece coordinate system.
Chopping function	Grinding the side surface of a workpiece by executing a contour program for an axis other than the grinding axis while constantly moving the grinding axis back and forth.
Chuck and tail stock barrier	Checking for interference between the chuck, tail stock, and tool to prevent damage to the machine.
Circle cutting function	Simplified machining method for producing a true circle by moving a tool along the circumference of a target circle from the center of the circle.
Circular interpolation	Obtaining a path necessary to move the tool along an arc in a specified plane.
Circular threading	Combination of two–axis circular interpolation and linear interpolation for up to two axes, one of which is the major axis in circular interpolation while the other is any axis. Circular threading can be used for constant–pitch threading, grooving, and tool grinding on a barrel–shaped surface.
Clamp of maximum spindle speed	Specifying a rotation speed per minute as the maximum spindle speed during constant surface speed control.
Cleared state	Initially specified device state.
Command format	Array of program component enabling direct manipulation of the CNC.
Comment section	Information such as comments and directions output to the operator.
Common variable	Macro variable that can be used by two or more different custom macro programs.
Compensating backlash along C–axis for each tool group	Compensating the position of tools that can be controlled using the C–axis for C–axis backlash.
Compensating position of C–axis	Correcting mechanical error when a tool that can be controlled using the C–axis is mounted.
Compensation function	Generic term applied to tool path, backlash, and pitch error compensation.
Composite control (Two– path control function)	Two–path control in which a move command for an axis in one of the paths is exchanged with a move command for an axis in the other path.
Conical interpolation	Obtaining a conical path by adding a one–axis movement command to a spiral interpolation command to specify an increment/decrement per spiral turn for the added axis.
Constant surface speed control	Controlling the spindle speed during turning so that the cutting feedrate remains constant despite changes in the target radius.
Continuous threading	Threading in which threading command blocks are specified continuously so that spindle synchronization is not lost between blocks. This method is useful for producing special threads such as one for which the lead changes midway.
Controlled axis	Axis controlled by the CNC or PMC.
Conversational automatic programming function	Programming by entering data in response to figures and guidance displayed on the screen,

Term	Description
Conversational programming with graphic function	Interactively programming blocks, one at a time, based on a G code menu displayed on the screen.
Coordinate system	Right–hand orthogonal coordinate system in which three linear axes, X, Y, and Z, are normal to one another. This is used to define coordinates for informing the CNC of the position to which the tool is to be fed.
Coordinate system rotation	Rotating a figure, specified in a program, around a specified point.
Corner circular interpolation	Circular interpolation performed at a corner of the tool path by using the tool offset values as the corner radius in offset mode for tool nose radius compensation.
Corner offset circular interpolation	Circular interpolation performed at a corner between blocks by using the tool offset value as the corner radius during cutter compensation.
Corner R	Grinding the outer or inner corner of a workpiece to make it round.
CRT/MDI	Panel which incorporates both a cathode ray tube (CRT) and a manual data input (MDI) keyboard. Used to display and set program and data in the CNC.
Cs axis control switching function	Program–controlled switching between Cs axes (spindles subjected to contour control) controlled by each tool part.
Current position display	Displaying the current tool position using coordinates.
Custom macro	A program or sub–program which, in addition to commanding motion and giving commands to the machine, can also communicate with the PMC, do calculations, and do conditional executions, branches and loops.
Custom macro Interrupt signal	Interrupt signal used to execute an interruption type custom macro.
Cutter compensation	Shifting a tool path programmed for a tool by the offset value (radius) of the tool in a direction normal to the tool path.
Cutting feed	Feeding a tool at a speed (cutting feedrate) specified in a program.
Cutting feedrate clamp	Clamping the cutting feedrate to the upper limit specified with the CNC if a command specifies a value greater than the upper limit.
Cutting feedrate override	Manual control in which the operator can change the cutting feedrate.
Cutting mode	Operation mode in which the tool moves to the next block without being decelerated at the end of the current block.
Cutting speed	Tool feedrate relative to the feedrate for the workpiece being cut.
Cycle start	Starting an automatic operation.
Cycle time	Duration of one automatic operation session (excluding stop and pause).
Cylindrical interpolation	Converting the rotation angle of a rotary axis into a displacement on a linear axis on the circumference of a circle in the CNC, performing linear interpolation or circular interpolation between the linear axis and another axis, then converting the interpolated distance to an angle. Cylindrical interpolation is used to simplify programming for grooving in a cylindrical cam.

[D]

D code	Coded number, following the D address, that specifies a tool offset number.
Data protection key	Key provided to protect programs, offset values, parameters, and setting data from being inadvertently registered, altered, or deleted.
Decimal point programming	Entering numeric data using a decimal point.
Diameter programming	Programming for turning in which the amount of movement along the X–axis (or coordinates) is represented using diameters.
Dimension word	Word that represents an amount related to axis movement. It can be an axis movement destination or an arc radius.
Direct drawing dimension programming	Operating the CNC based on a program that uses line angles, chamfer values, and corner R values on machining drawings.
Direct input or tool offset value measured B	Automatically setting a tool offset value or workpiece coordinate system shift value in a CNC for a lathe by manually operating the tool.
Directory	List of files.

Term	Description
Distance to go	The remaining amount of movement specified in a block.
Distribution amount	Number of pulses to be distributed during pulse distribution.
DNC operation	Automatic operation based on a program being loaded into the CNC via an interface. In this operation, the program to be loaded can be specified, and the CNC can be operated based on the specified execution sequence and a specified execution count.
Drilling mode	Mode in which a hole can be machined.
Dry run	Operation for program testing with no workpiece attached. A feedrate can be selected manually to override a programmed feedrate.
Dwell	Deferring the execution of the next block by a specified period.
Dynamic graphic display	Drawing a programmed tool path and a target figure on the CNC screen.

[E]

Each–block calling	Unconditionally calling a specified custom macro program for each individual CNC command block.
EDIT mode	CNC state in which programs can be edited.
Editing unit	Minimum unit in which program editing, such as deletion, can be performed. Usually the editing unit begins at one address and ends jut before the next address.
EIA code	Information exchange code complying with EIA standard EIA–244–B (abolished in July, 1992). Used in numerical control.
Emergency stop	Entering an emergency stop signal to the CNC to cancel all commands, thereby bringing the machine to an immediate stop.
End of block code (EOB)	Code (character) that signifies the end of a block.
End of program	Miscellaneous function indicating the end of a main program. (M02,M30)
End of record (EOR)	Code that signifies the end of a program. This code is displayed as a percent symbol (%) on the CNC program screen.
End of subprogram	Miscellaneous function indicating the end of a subprogram. (M99)
Error code	Number assigned to a classified alarm
Exact stop mode	Operation mode in which the tool is decelerated at the end of a block. The next block is not started until after it has been confirmed that the tool is in an in–position state.
Exponential interpolation	Changing the rotation of a workpiece exponentially as the rotary axis moves, and performing linear interpolation between that axis and another. Used for tapered grooving with a constant spiral angle.
Extended part program editing	CNC program editing, such as copying or moving an individual CNC program, or connecting it to another CNC program.
External I/O device	Device connected to the CNC to transfer programs and tool offset data with the CNC.
External motion function	Outputting a signal (external operation function signal) from the CNC each time a block in a program finishes positioning, causing the machine to perform a specific operation.
External workpiece origin offset value	Offset value from the machine zero point used to offset the workpiece coordinate system origin. There is only one external workpiece origin offset value, common to all workpiece coordinate systems, while the workpiece origin offset value is provided for each individual workpiece coordinate system.

[F]

F code	Coded number, following the F address, that specifies a feedrate or an amount of feed relative to a workpiece.
Feed function	Controlling the tool feedrate.
Feed hold	Temporarily stopping feed during program execution.

Term	Description
Feed per minute	Cutting feed in which the distance the tool is to advance is specified per minute.
Feed per revolution	Cutting feed in which the distance the tool is to advance is specified per spindle rotation.
Feedrate clamp based on arc radius	Automatically clamping a circular cutting feedrate so that an arc radius error due to post–interpolation acceleration/deceleration and servo delay does not exceed an allowable error.
Figure copy	Repetitive machining performed by rotating a figure specified by a subprogram, or shifting it in parallel.
File	Named set that is stored or processed as a single unit.
File heading	Specifying a file that is to be manipulated (for example, to be input to the CNC).
Floating reference position return	Returning a tool to a floating reference position. The floating reference position serves as a reference position for a specified mechanical operation. It is not necessarily a fixed position. It may be relocated.
Forbidden area	Area where a tool is forbidden to enter.
Format guidance	Displaying program creation guidance in a specified format on the screen.
Function for switching between diameter and radius programming	Preparatory function for switching between diameter and radius specifications.
Function key	Key on the MDI keyboard used to select a screen to be displayed on the CNC display unit. Function keys are classified by function.

[G]

G code	Code that represents a preparatory instruction. A number that allows the G address.
G function	Command that determines a machine and/or CNC function mode, such as interpolation type , canned cycle, threading, and coordinate system selection.
Geometric offset value	That part of a tool offset value which compensates for the geometry of the tool.
Graphic function	Drawing the trajectory of a tool, being driven by the current machining program, on the CNC screen.
Group number	Common number assigned to G codes having similar functions. For example, group number 00 is assigned to one–shot G codes such as G04, G05 and G45.

[H]

H code	Coded number, following the H address, that specifies a tool offset number in a machining center.
Helical interpolation	Obtaining a path for moving the tool along a spiral by feeding another axis in synchronization with circular interpolation.
Help function	Displaying detail information relating to alarm detected by the CNC, or operating instructions for the CNC, on the screen.
High precision contour control	The following functions, executed at high speed to eliminate machining error due to post–interpolation acceleration/deceleration. (1) Pre–interpolation error– free acceleration/deceleration based on multiple blocks read in advance. (2) Smooth acceleration/deceleration in which changes in shape and speed are accommodated and the allowable acceleration of the machine is observed by reading multiple blocks in advance.
High–speed cycle cutting	Converting a figure to be machined to a group of data items that can be distributed as high–speed pulses, saving the conversion results to memory, and executing the CNC command to call the data group in a machining cycle.
High–speed machining function	Executing preprocessing for a machining program before actual machining, saving the preprocessing results into memory, and performing actual machining based on the preprocessing results retrieved from memory.
High–speed remote buffer	Enabling the supply of a large amount of data to the CNC both at high speed and continuously via a serial interface.
High–speed remote buffer A	High–speed remote buffer for supplying movement data in binary.

Term	Description
High–speed remote buffer B	High–speed remote buffer for supplying movement data as source code created in an automatic programming unit.
HPCC mode	Mode in which high–precision contour control (HPCC) is performed.
Hypothetical axis interpolation	Distributing pulses by using one of circular interpolation axes as a hypothetical axis, thereby varying the feedrate of the controlled axis sinusoidally.

[I]

I/O channel	Channel used to transfer data between input/output devices and the CNC.
Imaginary tool note	Point defined on an imaginary axial straight line extending from the tip of a tool. This makes it easier to set the tool to its start position or reference position.
In–position	State in which a servo motor is positioned to a point within a specified range (previously set in the CNC).
Inch threading	Precise threading in which the number of thread crests per inch is specified.
Inch/metric switching	Selecting whether data is to be input in inch or metric units.
Increment system	Generic term for least input increment and least command increment.
Incremental feed	Feeding a controlled axis by a preset amount each time the corresponding button is pressed.
Incremental programming	Method by which an amount of tool movement (relative to the previous tool position) is programmed.
Incremental value	Distance or angle relative to the previous position.
Index table indexing function	Indexing on the index table of a machining center.
Initial position	Level in a hole axial direction to which positioning is performed for the first time during a canned hole machining cycle. Succeeding drills return to the R Plane.
Input buffer	Area into which input data is temporarily saved.
Interference check	Preventing a tool from overcutting the workpiece or from striking the tool on the other tool post.
Interlock	Preventing the movement of a controlled axis. This function is enabled by supplying an interlock signal to the CNC.
Internal circular cutting feedrate change	Controlling circular cutting so that the feedrate for a programmed path matches the specified cutting feedrate when the actual tool path is offset within the programmed tool path.
Interpolation functions	Obtaining a tool path according to a function (such as a linear or arc function) corresponding to a specified preparatory function.
Interruption type custom macro	Calling a program to be executed by entering an interrupt signal to the CNC during the execution of another program.
Inverse time feed	Cutting feed for which the reciprocal of the time required to feed the tool is specified.
Involute interpolation	Determining the path necessary to move the tool along an involute curve in a specified plane.
ISO code	Information exchange code complying with the applicable ISO standard. Used in numerical control.

[J]

Jog feed	Manually feeding a specified controlled axis at a specified feedrate.
Jog feedrate override	Manual control in which the operator can change the jog feedrate.

[L]

Lag of servo system	State in which the feed axis of a machine lags behind the corresponding feed command.

Term	Description
Laser beam on/off control	Turning on and off a laser beam. A laser beam can be turned on and off either manually or automatically.
Laser sequence control	Control applied to the series of operations from activation to stop of the laser oscillator. Control of a carbonic gas laser oscillator and that of a YAG laser oscillator are supported.
LCD/MDI	Panel which incorporates both a liquid crystal display (LCD) panel and a manual data input (MDI) keyboard. Used to display and set programs and data in the CNC.
Leader section	Program component used as a program file header.
Leading edge compensation	Offsetting a tool path by the tool radius so that the tool edge coincides with a programmed path if the tool (tool axis) is oriented in an arbitrary direction in three–dimensional space.
Least command increment	The smallest unit of controlled axis movement that can be specified by the CNC or PMC.
Least input increment	The smallest unit of data that can be input to a program.
Linear acceleration/ deceleration after cutting feed interpolation	Linear acceleration/deceleration applied to a specified cutting feedrate, in which the post–interpolation cutting feedrate is proportional to the elapsed time.
Linear acceleration/ deceleration before cutting feed interpolation	Linear acceleration/deceleration applied to a specified cutting feedrate, in which the pre–interpolation cutting feedrate is proportional to the elapsed time.
Linear axis	Axis along which a machine element moves linearly with the X–, Y–, or Z–axis of the machine coordinate system, or axis parallel to that axis.
Linear copy	Repetitive machining performed by moving a subprogram–specified figure in parallel.
Linear interpolation	Obtaining a path necessary to move the tool along a straight line.
Linear interpolation type positioning	Positioning in which the tool path coincides with a path obtained by linear interpolation.
Load meter display	Representing as a bar graph, the load ratio of a servo motor or spindle motor relative to its rated load as 100%.
Local coordinate system	Coordinate system defined in a workpiece coordinate system in order to facilitate programming based on the workpiece coordinate system.
Local variable	Macro variable that can be independently used in individual custom macro program.
Look–ahead control	Enabling high–speed, high–precision machining by suppressing acceleration/ deceleration delays and servo delays that would otherwise become larger with increases in the feedrate.

[M]

M code	Coded number, following the M address, that specifies a miscellaneous function.
M code group check function	Checking that the combination of M codes specified in a block is valid.
M code group function	Displaying M codes by group and checking that the combination of M codes specified in a block is valid.
M codes for tool post synchronization	M code that causes a tool post to wait for another during machining.
M function	Specifying machine operations such as start and stop of the spindle and the end of a program.
Machine coordinate system	Coordinate system whose origin is defined as being the machine zero point, a machine–specific point which acts as a reference point for the machine
Machine lock	Changing position displays, without moving the controlled axes, for program checking.
Machining time stamp function	Measuring the time required to execute a program, on a memory operation basis, and displaying the measured time on the CNC screen. The measured time is written as a comment in the program.
Macro call	Calling a custom macro program for execution, passing parameters.

Term	Description
Macro compiler/macro executer	Programs used to convert a custom macro source to an executable form (macro compiler), save the conversion results into ROM, and execute them (macro executer).
Macro statement	Block containing a calculation command, control command, or macro call command.
Macro variable	Variable used in a custom macro program.
Main program	Set of instructions that form the main part of a program. This term is used in contrast to the term subprogram.
Manual absolute on and off	Manual intervention for selecting whether to add the amount of movement caused by manual operation to the coordinates (current position in a workpiece coordinate system) handled by the CNC.
Manual feed in specified direction	Feeding a controlled axis manually in any specified direction.
Manual handle feed	Feeding a specified controlled axis by rotating the handle to generate command pulses.
Manual handle interruption	Manual handle feed performed during automatic operation, in such a way that the manual–feed amount is added to the automatic–feed amount.
Manual intervention	Pausing automatic operation and starting manual operation.
Manual numeric command	Feeding a controlled axis in jog mode by executing the data specified in program form.
Manual operation	Feeding a controlled axis manually.
Manual per revolution feed	Jog feed in which the feedrate is obtained by multiplying the feedrate per rotation, set in the CNC, by the spindle speed. Used to override the jog feedrate.
Manual pressing	Manual control in which the operator presses a button on the machine operator's panel to start punching.
Manual pulse generator	Unit that converts rotation to pulse train when its handle is rotated manually. Used for manual handle speed.
Manual rapid traverse	Feeding a controlled axis at the rapid traverse rate in jog mode.
Manual reference position return	Manual feeding a specified controlled axis to a reference position.
Master axis	Axis for which move commands can be specified during synchronous operation.
Maximum Stroke	Maximum range of movement that can be controlled by the CNC or PMC.
MDI mode	Mode in which MDI operation is possible.
MDI operation	Automatic operation based on a program input to the CNC from the MDI keyboard. The program is erased upon the completion of MDI operation.
Memory mode	Mode in which memory operation can be performed.
Memory operation	Automatic operation based on a program previously stored into CNC memory.
Menu switch	Use of the CRT/MDI, LCD/MDI or PDP/MDI panel to emulate some switches on the machine operator's panel.
Mirror images	Inverting an incremental value for a programmed dimension word along a specified coordinate axis from positive to negative, or vice versa, with respect to a specified reference point.
Miscellaneous function Auxiliary function	Specifying start or stop of the spindle, or the end of a program. Spindle and tool functions may sometimes be included, in which case the term auxiliary function is used.
Modal call	Calling a custom macro program (once an instruction to call it is specified) each time a block having an axis movement command is executed. The calling is repeated until the call instruction is canceled.
Modal G code	G code which, once issued, remains valid until another G code in the same group is issued.
Mode	Holding a specified function in the CNC. For example, once a cutter compensation preparatory function is issued, the CNC stays in the state in which cutter compensation is possible until a cutter compensation cancel preparatory function is issued (cutter compensation mode).

Term	Description
Mode selection	Selecting an operation mode.
Move command calling	Calling a specific custom program from a block containing a move command, after the move command has been executed.
Multi–edit function	Displaying two programs side–by–side so that they can be edited simultaneously.
Multi–piece machining function	Using simplified commands to punch out two or more products of the same shape from a workpiece.
Multibuffer	Preventing interpolation from being stopped between blocks by buffering multiple blocks.
Multiple M commands in a single block	Enabling the issue of more than one M code in a single block.
Multiple repetitive cycle	Canned cycle that is repeated until a program–specified target figure is attained. By means of this method, the specification of only a final figure, for example, enables the automatic determination of intermediate tool paths.
Multiple subscreens	Displaying information about the current position and a program being executed on a subscreen (window placed on the main screen).
Multiple tool control	Automatically indexing tools in a multiple–tool unit, which consists of two or more different tools in a single tool holder.

[N]

NC statement	Non–macro statement block that directly controls the CNC.
Nibbling	Punching performed by running the press continuously and repetitively.
Nonlinear interpolation type positioning	Positioning individual axes independently.
Normal direction control	Controlling a rotary axis so that the tool is oriented in a direction normal to that of its forward motion.
Normal operation	Operation in which the movements of the master and slave axes are specified using separate axis addresses. This is equivalent to normal CNC operation and is used to machine workpieces on different tables independently.
Number of registerable programs	Number of programs that can be saved to CNC memory.

[O]

Offset	Deviation from a true tool path or coordinate system origin to compensate for tool size. Synonymous with “compensation”.
Offset memory	CNC memory used for storing tool offset values, workpiece origin offset values, and external workpiece origin offset values.
Offset mode	CNC state in which tool path offset is allowed.
Offset plane	Plane in which tool path offset is active.
Offset space	Space in which tool path offset is allowed.
Offset vector	Vector whose direction and size are the same as those of a specified tool offset. As the tool advances, the vector direction is rewritten for individual blocks according to calculations within the CNC so that it is always held normal to the tool path.
One–digit F code feed	Cutting feed in which the tool is fed at the feedrate set in the CNC, and which corresponds to the digit (from 1 to 9) immediately after the F address.
One–shot G code	G code that remains valid only within the block in which it is listed (such as G31).
Operating monitor display	Display of the servo axis load meter, spindle load meter, and speed meter.
Operation in the tape mode	Automatic operation based on a program loaded into the CNC via an interface. In this operation, the program to be loaded can be specified, and the CNC can be operated based on the specified execution sequence and specified execution count.
Operation mode	Mode in which automatic or manual operation is possible.

Term	Description
Operator message display	Screen used to inform the operator of the current machine status, and to display prompts to the operator.
Optional block skip	Adding a “/”, followed by a number, to the beginning of a block so that that block can be selectively skipped.
Optional stop	Miscellaneous functions for causing a program to pause when the “Optional Stop” switch on the machine operator’s panel is set to the ON position.
Output ahead of T–code function	Searching fhrough a machining program for T commands in the execution sequence, starting from the beginning, and outputting the detected T commands before executing the program. This function enables the machine to prepare for tool exchange.
Overall position display	Simultaneous display of the current position and remaining distance in the workpiece coordinate system, relative coordinate system, and machine coordinate system.
Override cancel	Clamping a feedrate override value to 100%.
Override playback	Storing a cutting feedrate override value and spindle speed override value during the execution of a program. Restoring and using the override values when the program is next executed.
Overtravel	Decelerating and stopping a tool if it goes beyond a machine stroke end, and displaying an alarm.

[P]

P/S alarm	Alarm related to programs and manipulation.
Parallel axis	Controlled axis (such as the U–, V–, or W–axis) parallel to the X–, Y–, or Z–axis, respectively.
Parallel operation	Operation mode in which a move command for a certain program axis is used to simultaneously feed two or more controlled axes (parallel axes) having the same name. A parallel axis is represented using a combination of the same axis name address as for the corresponding basic axis and a number (such as X1, X2, and so on).
Parameter	Data (such as feedrate, coordinate system, spindle, and tool parameters) set in the CNC to define its specifications.
Part program	Sequence of instructions created using a language and format that support the direct manipulation of the CNC. Alternatively, a sequence of instructions prepared as input data to be processed during automatic programming.
Part program storage length	Size of a program that can be stored in CNC memory as an equivalent paper tape length (number of characters x 2.54 mm).
Password function	Disabling the editing of specific programs (such as those identified by program number 09000 to 09999).
Pattern data input	Simplified programming in which menus are used to set numeric data (pattern date), based on drawings, in the CNC.
Pattern function	Punching at two or more positions arranged in a known layout, using a single block.
Pattern storage and recall	Pattern function in which codes A1 to A5 are assigned to patterns of the same figure, storing them, and restoring them using codes B1 to B5 when necessary.
PDP/MDI	Panel which incorporates both a plasma display panel (PDP) and a manual data input (MDI) keyboard. Used to display and set programs and data in the CNC.
Pitch error compensation	Compensating for pitch errors in a mechanical feed section.
Plane conversion function	Machining in which a machining program created on a G17 plane is converted so that the resulting figure looks the same when viewed from another plane in an orthogonal coordinate system.
Plane selection	Selecting a plane for circular interpolation, a plane for cutter compensation, a plane for coordinate system rotation, or a plane for hole machining, using a preparatory function.
Playback function	Programming in which a command assumes that a position to which the tool is moved manually is that command’s target tool position.

Term	Description
PMC	Sequence controller configured in the CNC and used to execute ladder program. The term PMC stands for programmable machine controller. The PMC is placed between the CNC and machine to control the input/output of signals between them.
Pocket calculator type decimal point programming	Decimal number input in which the values are input in units of mm, inches, or degrees.
Polar coordinate command	Program command that specifies the end point of tool movement in a polar coordinate system (using a radius and angle).
Polar coordinate interpolation	Interpolation performed by converting a command programmed in an orthogonal coordinate system into a combination of a linear axis movement (tool movement) and rotary axis movement (workpiece rotation). This is used, for example, when grinding a cam shaft.
Polygon turning	Machining a polygon by changing the rotation ratio between the workpiece and tool, and the number of cutters used.
Position coder	Device, connected to the spindle by means of a belt, that detects and outputs the rotation angle of the spindle as a pulse train. It is used to detect the tool exchange position and to perform threading.
Positioning	Feeding a tool to the target position at a traverse feedrate previously specified in the CNC.
Power output command	Command for specifying the peak power when outputting a laser. The peak power is specified with the value immediately after the S address in the block which contains G01 (G02, G03, or G24).
Preparatory function	Command that determines a machine and/or CNC function mode, such as interpolation type, canned cycle, threading, and coordinate selection.
Press start lock	Preventing a press from starting. The press is prevented from starting by inputting a press start lock signal to the CNC.
Press start waiting	Deferring the start of a press according to the machine conditions. The press is prevented from starting until a press start waiting signal applied from the machine is released.
Pressing (Punch)	Using a punch press to punch out a product from a workpiece or mold a product.
Program	In the CNC operator's manual, a sequence of instructions created using a language and format enabling direct manipulation of the CNC. In many cases, other types of programs are identified using qualifiers, as in "conversational programs."
Program encryption	Protecting programmed information by mean of encryption.
Program end	Miscellaneous function indicating the end of a main program.
Program number	Number following the O address that is added to the beginning of a program to discriminate it from others.
Program number search	Searching through programs for one identified by a specified number, and calling that program once located.
Program restart	Resuming automatic operation from an intermediate block of the program.
Program section	The part of a program between a program number and an end–of–program code.
Program start	Symbol signifying the start of a program.
Program stop	Miscellaneous function for temporarily stopping program execution.
Programmable mirror image	The ability, in the part program, to command mirror image of axes(is).
Programmable parameter input	Enabling a program to change parameter values. This function is used to set pitch error compensation data, or change the maximum cutting feedrate or cutting constants according to the machining condition.
Programmable rapid traverse override	Overriding a rapid traverse rate during automatic operation by specifying the F address followed by a number from 1 to 4 that corresponds to the override ratio.

Term	Description
Programmed path	Tool path drawn using a specific point on a cutting tool when compensation has not been applied for that tool. In a program, a programmed tool path and compensation (such as tool length compensation or cutter compensation) are specified independently. The CNC determines the actual tool path by correcting the programmed path according to a compensation command.
Pulse distribution	Converting the amount of movement specified for each axis to a number of pulses, according to a command issued for a tool path, and distributing the pulses to each controlled axis.
Pulse frequency override	Manual control in which the operator can temporarily change the laser pulse frequency during operation, to vary the laser output power. This function is used to create the same figure with different materials or stock thicknesses.
Punch forbidden area	Disabling punch commands (if any) in a safety zone.

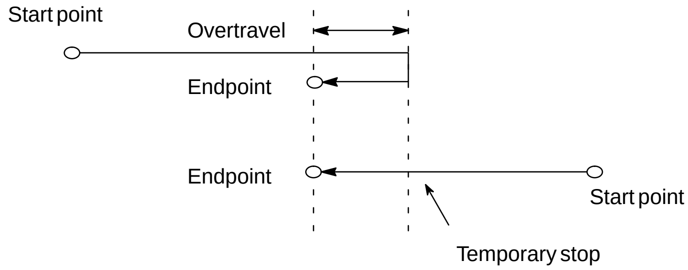
[R]

R Plane	Return position level in a hole axial direction, set up in the immediate vicinity of a workpiece, in order to quicken hole machining operations when a canned cycle is used repeatedly to machine holes.
Radius programming	Programming for turning in which the amount of movement along the X–axis (or coordinates) is represented using radiuses.
Rapid traverse	Feeding the tool at a speed (rapid traverse rate) specified in the CNC when a positioning command is issued.
Rapid traverse override	Manual control in which the operator can change the rapid traverse rate during machining.
Reader/puncher interface	Interface between an input/output device and the CNC.
Reference position	Specific position on the machine along an axis, relative to the origin of a machine coordinate system.
Reference position return	Moving a specified axis to the reference position.
Reference position return check	Checking that the tool has been successfully returned to the reference position. This check is used by a program that is designed to return a tool to the reference position.
Relative coordinate system	Coordinate system established in reference to the coordinates set by the operator using the CNC.
Repeat search	Searching for an address or word again by using the repeat key.
Reset state	Initial state defined for a device.
Retrace function	Causing a tool to move back along a path which it previously traversed (reverse), then retracing the same path again (re–forward).
Retract	Automatic operation in which the tool is retracted by a programmed amount.
Return point level R Plane	Level to which a tool is retracted from the bottom of a hole being created during the execution of a canned cycle. This is either the point R level or initial level.
Rewinding a program	Locating the beginning of a program.
Rigid tapping	High–precision tapping achieved by controlling spindle rotation and drill axis feed as two–axis linear interpolation so that no tapping pitch error occurs at the bottom of the hole during acceleration/deceleration.
Rotary axis	Axis (such as A, B, or C) that rotates about a linear axis in a machine.
Rotary axis roll–over function	Rounding off a rotary axis coordinate to within 360.
Rotary table dynamic fixture offset	Automatically calculating an offset from a rotation center when the rotary table rotates, thereby defining a workpiece coordinate system.
Rotational copy	Repetitive machining performed by rotating a subprogram–specified shape.
Rotational handle feed around tool tip	Manually feeding a tool using a handle in such a way that, when the tool direction is changed, the tool tip is held in the same position.
RS–232–C	EIA standard specifying a binary serial data interface for input/output devices.
RS–422	EIA standard specifying a binary serial data interface for input/output devices.

Term	Description

[S]

S code	Coded number, following the S address, that specifies the rotational speed of the spindle.
S function	Controlling the rotational speed of the spindle by specifying a number after the S address.
Scaling	Reducing or enlarging a programmed figure, using a specified point as the center.
Scheduling function	Selecting a file on an external input/output floppy device, so that automatic operation is performed based on the specified execution sequence and specified execution count.
Second auxiliary function	Auxiliary function for specifying a function such as indexing table positioning.
Selecting a workpiece coordinate system	Selecting a workpiece coordinate system from those set in the CNC.
Self–diagnosis function	Failure diagnostic function provided for the CNC. This function identifies mechanical, electrical, and human errors.
Sequence number	A number preceded by the N address and placed at the beginning of a program block to identify a specific block.(need not be sequential)
Sequence number comparison and stop	Searching for a block identified by a specified sequence number during program execution, executing the target block (if found), and then stopping automatic operation.
Sequence number search	Searching a program for the block identified by a specified sequence number and selecting that block.
Serial pulse coder	Rotary detector that encodes a detected position as serial data prior to transmission.
Servo off	Shutting down the power supply for a servo motor. This function is enabled by inputting a signal to the CNC. It can be used to clamp a controlled axis mechanically and to prevent a servo motor from being overloaded.
Setting a workpiece coordinate system	Defining a workpiece coordinate system in the CNC.
Setting data	Data that is selected and set by the user in the CNC to determine the CNC specifications, such as output data code setting, command format setting, and input/output device selection setting.
Shutter control	Opening and closing the mechanical shutter of the laser oscillator. The shutter can be opened and closed either manually or automatically.
Significant information section	The part of a program which begins at the program number and ends at the end of the program, and from which all comments have been executed.
Simple call	Custom macro program calling in which a call instruction is issued each time the program is to be executed.
Simple conversational programming	Creating a program according to a menu displayed on a screen.
Simple synchronous control	Controlling two axes with one command, in some CNCs ignoring any difference in lag between the axes. The axes can be synchronized or separately controlled based on machine input in some CNCs.
Simultaneous automatic and manual operation	Simultaneously executing automatic and manual operations.
Simultaneously controlled axes	Axis that can be controlled simultaneously with another.
Single block	Automatic operation in which one program block is executed each time CYCLE START is initiated.

Term	Description
Single direction positioning	Final positioning performed in a single direction to accurately position a tool or workpiece by excluding play, or lost motion, in the mechanical section. 
Skip function	Linear interpolation (G31) in which the commanded motion and remaining distance–to–go is discarded when a signal (skip signal) is received from outside the CNC. The position is saved in a system variable.
Skip signal	Input signal received from outside the CNC, informing the CNC of the movement end point during the execution of a skip motion command.
Slave axis	Axis whose movement is synchronized with the master axis during synchronous operation.
Smooth interpolation	Interpolation in which a figure requiring a high degree of accuracy, such as a corner, is machined based on programmed commands, and in which a figure having a large radius requiring a smooth finish is machined by generating a curve from a sequence of specified points and subsequently interpolating it.
Soft key	Key displayed on the CNC display unit. Used to select a menu or command.
Software operator’s panel	Software–implemented operator’s panel that enables the CRT/MDI panel to take the place of the indicators and switches of the machine operator’s panel.
Spindle control switch function	Program–controlled switching between the spindles controlled by each tool post on a two–spindle, two–tool post machine.
Spindle orientation	Stopping the spindle at a preset position.
Spindle positioning	Orienting a workpiece, attached to the spindle, to a certain angle.
Spindle speed fluctuation detection function	Issuing an alarm when the actual spindle speed becomes a value higher or lower than that specified because of a condition existing in the machine.
Spindle speed function	Controlling the rotation speed of the spindle by specifying a number after the S address.
Spiral interpolation	Determining a spiral path by specifying an increment or decrement in the number of rotations or a radius per rotation, as well as a circular interpolation command.
Spline interpolation	Determining a path for a spline curve that passes through a series of specified points.
Start–up	Tool movement when cutter compensation is started in offset cancel mode.
Status display	Displaying the status of the CNC operation.
Storage of macro	Registering a macro by placing the U address, followed by a two–digit number, before two or more block commands to be stored, and by closing the commands with a V address followed by the same two–digit number used with the U address.
Stored stroke check	Setting a forbidden area in the CNC for a tool, decelerating the tool to a stop, and issuing an alarm if the tool is about to enter the forbidden area.
Stored stroke limit	(See “Stored stroke check.”)
Stroke limit check before move	Stroke limit check performed before the movement specified in a block is started.
Subprogram	Program that can be called repeatedly by the control section of another program.
Superposition control (Two–path control function)	Two–path control in which a move command for an axis in one of the paths is superimposed on an axis in the other path.
Synchronization control (Two–path control function)	Two–path control in which a move command for an axis in one of the paths is used to control an axis in the other path so that they are synchronized.

Term	Description
Synchronous operation	Operation in which an axis is controlled using a move command for another axis so that both axes are synchronized. This is used to machine extremely large workpieces that extend over two tables.
System variable	Macro variable used to read or write CNC data, such as a tool offset value and current position.

[T]

T code	Coded number, following the T address, that specifies a tool function.
T command neglect	Ignoring T commands. This function is enabled by supplying an ignore T command signal to the CNC.
T function	Specifying a tool or data related to the specified tool.
T–axis control	In a turret punch press machine, causing the CNC to calculate the required amount of movement relative to the current turret position and the turret position corresponding to a T command, thereby indexing the turret.
Tandem control	Control in which two motors are used to drive a single axis. This is used to drive, for example, a table that would be too large for a single motor to supply sufficient torque.
Tangential speed constant control	Maintaining a constant feedrate tangential to the tool path.
Tape code	Information interchange code used for numerical control.
Tape end	Symbol indicating the end of a program file.
Tape start	Symbol indicating the beginning of a program file.
Tapping mode ?? mixture of continuous threading and ???	Operation mode in which the tool moves to the next block without being decelerated at the end of the current block. Cutting feedrate override and feed hold are disable in this operation mode.
TEACH IN HANDLE mode	TEACH–IN mode where the manual operation is manual handle feed.
TEACH IN JOG mode	TEACH–IN mode where the manual operation is jog feed.
TEACH IN mode	Mode used to store information about the position of a controlled axis, obtained by manual operation, into the CNC memory for program creation.
Test operation	Confirming that a program operates as intended.
TH check	Checking whether the total number of 1 bits in a character is even or odd.
Thread cutting	Threading performed by feeding the tool at the cutting feedrate, per minute, determined from spindle speeds that are read at constant intervals.
Three–dimensional coordinate conversion	Three–dimensional coordinate conversion around a rotation center axis performed by specifying the center of rotation, the direction of the rotation center axis, and rotation angle.
Three–dimensional cutter compensation	Offsetting, by the tool radius, a tool (tool axis) that is oriented in an arbitrary direction in three–dimensional space by using a plane normal to the tool axis as a compensation plane. Instances of three–dimensional cutter compensation include tool side compensation and leading edge compensation.
Three–dimensional handle feed	Operation performed on a tool tilted around a rotary axis by using the manual handle. Instances of three–dimensional handle feed include tool direction handle feed, tool normal direction handle feed, and rotational handle feed around the tool tip.
Three–dimensional rigid tapping	Rigid tapping performed using a tool (tool axis) that is oriented in an arbitrary direction in three–dimensional space along the tool axis.
Three–dimensional tool compensation	Offsetting a tool path by the tool offset value in a three–dimensional direction specified in a program.
Tool direction handle feed	Manually feeding a tool tilted by the rotation of a rotary axis in a direction parallel to the tool axis of that tool.
Tool function	Specifying a tool or data related to the specific tool.
Tool length compensation	Compensating for the difference in length between the tool assumed during programming and the tool to be used for actual machining.

Term	Description
Tool length compensation along the tool axis	Tool length compensation for a tool (tool axis) that is oriented in an arbitrary direction in three–dimensional space.
Tool length measurement	Manual operation in which a reference tool and the tool to be measured are pressed against a fixed point on the machine, one after the other, and the difference in length between the tools is set as a tool length offset value in the CNC.
Tool length/workpiece origin measurement B	Measuring and setting the tool length/workpiece origin offset value and setting it.
Tool life management function	Managing the life (number of uses or age) of tools in a group and automatically selecting a new tool from the same group once the life of the current tool expires.
Tool normal direction handle feed	Manually feeding a tool tilted by the rotation of a rotary axis in a direction normal to the tool axis of that tool.
Tool nose radius compensation	Compensation for any difference between a programmed tool position and the actual nose contour of a tool having a circular nose. This compensation is performed in a direction normal to the tool path.(lathes)
Tool offset	Shifting a specified tool along the controlled axis.
Tool offset memory	CNC memory used to store tool offset values.
Tool offset number	Number preceded by the H or D address to specify a tool offset value.
Tool offset value	Offset value used by the tool length compensation, cutter compensation, and tool offset functions.
Tool path	Tool path drawn using a specific point on a cutting tool.
Tool post interference check	Operation in which the CNC detects a command that may cause the two tool posts of the CNC lathe to interfere with each other and stops the tool posts before they can collide.
Tool retract and recover	Retracting a tool from the workpiece, allowing the tool to be exchanged during machining (if broken) or the state of machining to be checked, and subsequently repositioning the tool to restart machining.
Tool selection function	Number that follows the T address, used to select a tool on the machine.
Tool side compensation	Offsetting, by the tool radius, a tool (tool axis) that is oriented in an arbitrary direction in three–dimensional space, so that the side of the tool coincides with the programmed tool path.
Trace interlock function	Stopping tracing control temporarily. This function is used to prevent the laser nozzle from dropping into a processed hole when it passes that hole during tracing control.
Tracing control	Control applied to maintain a constant distance between the workpiece surface and nozzle tip. This function is used to enable stable processing by ensuring that the focal position in the direction of the workpiece thickness remains constant.
Traverse inhibit limit function	Stopping an axis and continuing automatic operation if an absolute value related to that axis exceeds a preset value.
TV check	Checking whether the total number of characters in a block (starting immediately after an end–of–block code and ending at the next end–of–block code) is even or odd.
Twin table control	Switching between synchronous, independent, and normal operation for two or more specified axes.
Two–path control function	Controlling the two tool posts on the CNC lathe simultaneously and independently.

[W]

Warning message	Message displayed on the screen to indicate when incorrect data has been entered or an invalid operation has been performed from the CRT/MDI panel.
Waveform diagnosis function	Displaying data relating to servo and spindle motor movement graphically.
Wear offset value	The part of a tool offset value used to compensate for tool wear.
Word	Set consisting of an address followed by a multiple–digit number. A word is a component of a block.

Term	Description
Workpiece coordiate system shift	Shifting a workpiece coordinate system set in the CNC as required so that it matches a workpiece coordinate system assumed during programming.
Workpiece coordinate system	Coordinate system that is fixed for a workpiece and is used to machine that workpiece.
Workpiece coordinate system preset	Returning a workpiece coordinate system to its initial position if it has been shifted manually.
Workpiece origin offset value	Offset of the origin of a workpiece coordinate system from the machine zero point. If an external workpiece origin offset value is given, an offset from the machine zero point is defined by combining the external workpiece origin offset and the workpiece origin offset.
Workpiece zero point manual setting function	Specifying the workpiece origin offset on the workpiece origin offset screen so that the current position matches the specified origin.

Index

[Numbers]

[A]

[B]

[C]

[D]

[E]

[F]

[G]

[H]

[I]

[J]

[K]

[L]

[M]

[N]

[O]

[P]

[R]

[S]

[T]

[U]

[V]

[W]

Revision Record

FANUC Series 16*i*/160*i*–LB OPERATOR’S MANUAL (B–63664EN)

Edition	Date	Contents	Edition	Date	Contents
02	May, 2005	• Addition of following items: High–speed skip signal (G31), AI nano contour control, Machining condition setting function (E code), Gap control function, Laser high–speed control function, Improvement in nozzle length compensation, Torch turning control function, Data input/output by embedded Ethernet, and Automatic aging function setting screen • Addition of following system variables: Cutting condition setting function, Gap controlled axis automatic save/restore function, and Function for beam output during approach • Addition of following laser functions: Specification with E codes, Gap controlled axis automatic save/restore function, and Function for beam output during approach • Addition of following “method of replacing battery”: Battery in the CNC display unit with PC functions (3 VDC) • Addition of alarms • Correction of errors			
01	Jun., 2002	————————			