FANUC Series 0*i*-PC

OPERATOR'S MANUAL

B-64154EN/01

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. Once machining has started, keep well clear of the machine. Some machines move their table at high speed, presenting a risk of injury to persons standing nearby.

7. 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.

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

WARNING

9. 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 this 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. 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.

4. 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.

WARNING

5. Special M codes

In principle, a block which includes any of the following M codes, which specify the execution of special functions, must not contain any other codes. When it is impossible to avoid specifying an M code together with another code in the same block, refer to the relevant description in the manual supplied by the machine–tool builder. Failure to follow the specified procedure may result in damage to the machine or injury to the user.

- Forming mode/forming mode cancel
- Workpiece clamp/unclamp
- Nibbling mode/nibbling mode cancel
- Switching between punch mode and laser mode

6. 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.

7. Absolute/incremental mode

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

8. 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.

9. 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.

10. Programmable mirror image

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

11. 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.

12. Auto–repositioning

If the amount of retraction or return for auto–repositioning is changed, and repositioning is repeated many times, grasping of the workpiece may fail, possibly causing damage to the machine. Be careful therefore, when changing the amount of retraction or return.

13. C–axis control

Before attempting to specify C–axis control, select a tool which supports the use of C–axis control. If C–axis control is applied while an incompatible tool is selected, C–axis rotation may cause damage to the metal die, magazine, and/or hitter.

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 this 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. Disabled override

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

6. Origin/preset operation

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

WARNING

7. Workpiece coordinate system shift

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

8. Software operator's panel and menu switches

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

9. Manual intervention

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

10. Feed hold, override, and single block

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

11. Dry run

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

12. Cutter and tool nose radius compensation in MDI mode

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

13. Program editing

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

14. Safety zone function

Setting an invalid safety zone may cause damage to the machine. Be careful when changing the safety zone.

5 WARNINGS RELATED TO DAILY MAINTENANCE

WARNING

1. Memory backup battery replacement

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 this manual for details of the battery replacement procedure.

WARNING

2. Absolute pulse coder battery replacement

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 this manual for details of the battery replacement procedure.

WARNING

3. Fuse replacement

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

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 procedure for replacing batteries.

APPENDIX

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

This manual does not describe parameters in detail. For details on parameters mentioned in this manual, refer to parameter manual (B–64160EN) of Series 0*i*–PC.

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 0*i*–PC	0*i*–PC	Series 0*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.

Related manuals

The table below lists manuals related to Series 0*i*–PC. In the table, this manual is marked with an asterisk (*).

Manual name	Specification number	
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C DESCRIPTIONS	B–64112EN	
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C CONNECTION MANUAL (HARDWARE)	B–64113EN	
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C CONNECTION MANUAL (FUNCTION)	B–64113EN–1	
FANUC Series 0*i*–PC CONNECTION MANUAL (FUNCTION)	B–64153EN	
FANUC Series 0*i*–PC OPERATOR'S MANUAL	B–64154EN	*
FANUC Series 0*i*–MODEL C/0*i* Mate–MODEL C MAINTENANCE MANUAL	B–64115EN	
FANUC Series 0*i*–PC PARAMETER MANUAL	B–64160EN	
Programming		
Macro Compiler/Macro Executor PROGRAMMING MANUAL	B–61803E–1	
FANUC MACRO COMPILER (For Personal Computer) PROGRAMMING MANUAL	B–66102E	
PMC		
PMC Ladder Language PROGRAMMING MANUAL	B–61863E	
Network		
PROFIBUS–DP Board OPERATOR'S MANUAL	B–62924EN	
Ethernet Board/DATA SERVER Board OPERATOR'S MANUAL	B–63354EN	
FAST Ethernet Board/FAST DATA SERVER OPERATOR'S MANUAL	B–63644EN	
DeviceNet Board OPERATOR'S MANUAL	B–63404EN	
OPEN CNC		
FANUC OPEN CNC OPERATOR'S MANUAL Basic Operation Package 1 (For Windows 95/NT)	B–62994EN	
FANUC OPEN CNC OPERATOR'S MANUAL (DNC Operation Management Package)	B–63214EN	

Related manuals of SERVO MOTOR α*is*/ α*i* series

The following table lists the manuals related to SERVO MOTOR α*is*/ α*i* series.

Manual name	Specification number
FANUC AC SERVO MOTOR α*is* series FANUC AC SERVO MOTOR α*i* series DESCRIPTIONS	B–65262EN
FANUC AC SERVO MOTOR α*is* series FANUC AC SERVO MOTOR α*i* series PARAMETER MANUAL	B–65270EN
FANUC AC SPINDLE MOTOR α*i* series DESCRIPTIONS	B–65272EN
FANUC AC SPINDLE MOTOR α*i* series PARAMETER MANUAL	B–65280EN
FANUC SERVO AMPLIFIER α*i* series DESCRIPTIONS	B–65282EN
FANUC AC SERVO MOTOR α*is* series FANUC AC SERVO MOTOR α*i* series FANUC AC SPINDLE MOTOR α*i* series MAINTENANCE MANUAL	B–65285EN

Related manuals of SERVO MOTOR α series

The following table lists the manuals related to 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 AC SPINDLE MOTOR α series DESCRIPTIONS	B–65152E
FANUC AC SPINDLE MOTOR α series PARAMETER MANUAL	B–65160E
FANUC SERVO AMPLIFIER α series DESCRIPTIONS	B–65162E
FANUC SERVO MOTOR α series MAINTENANCE MANUAL	B–65165E

Either of the following servo motors and the corresponding spindle can be connected to the CNC covered in this manual.

– FANUC SERVO MOTOR α*i* series

– FANUC SERVO MOTOR α series

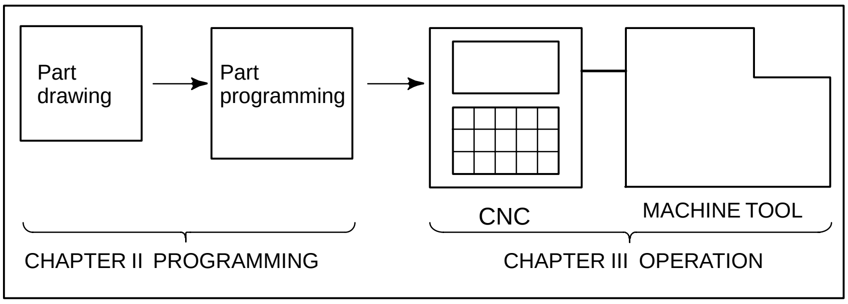
This manual mainly assumes that the FANUC SERVO MOTOR α*i* series of servo motor is used. For servo motor and spindle information, refer to the manuals for the servo motor and spindle that are actually connected.

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL

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

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

2) The program is to be read into the CNC system. Then, mount the workpieces and tools on the machine, and operate the tools according to the programming. Finally, execute the machining actually.
   How to operate the CNC system is described in the Chapter III. OPERATION.



Before the actual programming, make the machining plan for how to machine the part.
Machining plan
1. Determination of workpieces machining range
2. Method of mounting workpieces on the machine tool
3. Machining sequence in every cutting process
4. Cutting tools and cutting conditions
Decide the cutting method in every cutting process.

1.2 CAUTIONS 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 CAUTIONS ON VARIOUS KINDS OF DATA

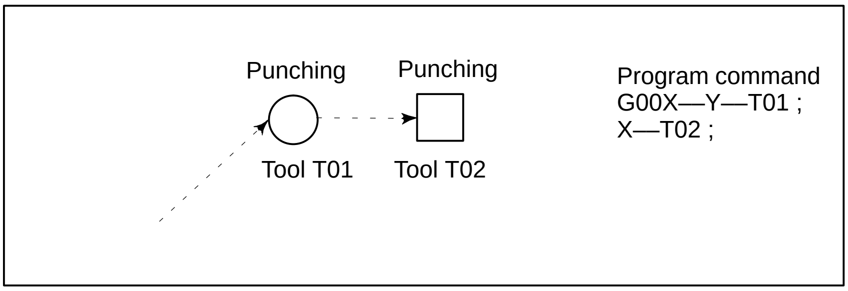
CAUTION

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.

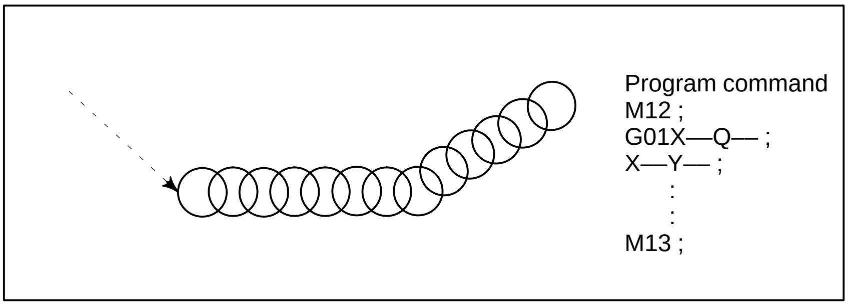
II. PROGRAMMING

1 GENERAL

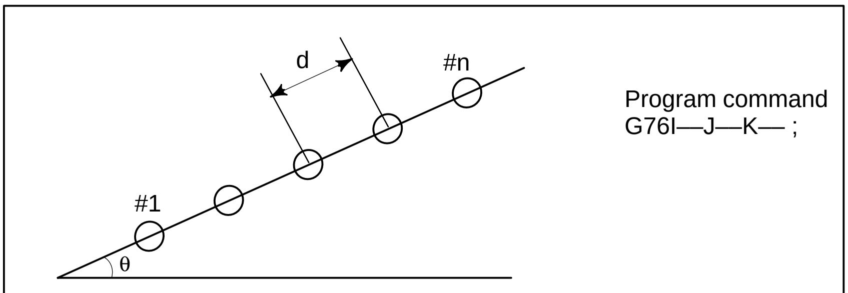
1) Punching is performered after positioning.
.... Punching function



2) Continuous, repetitive punching can be performed without halting the pressing process after positioning
.... Nibbling function



3) By giving commands for block, it is possible to perform at multiple positions in a given profile.
.... Pattern function



In case of line at angle (G76)

This CNC supports the eight different patterns that will be used most frequently.

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE– INTERPOLATION

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

Explanations

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

- **Tool movement along a straight line**

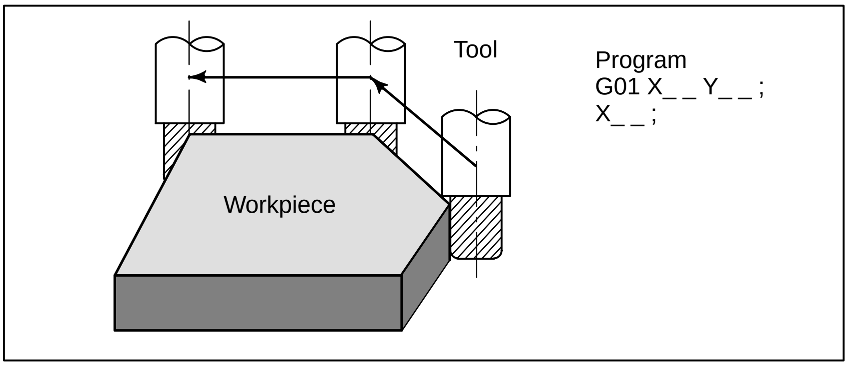


Fig. 1.1 (a) Tool movement along a straight line

- **Tool movement along an arc**

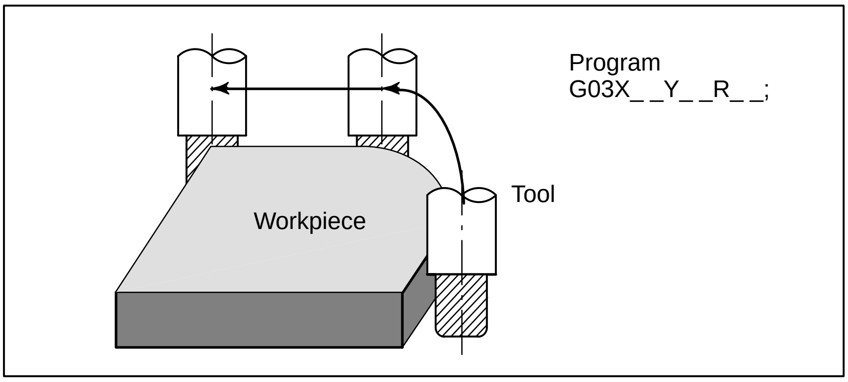


Fig. 1.1 (b) Tool movement along an arc

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

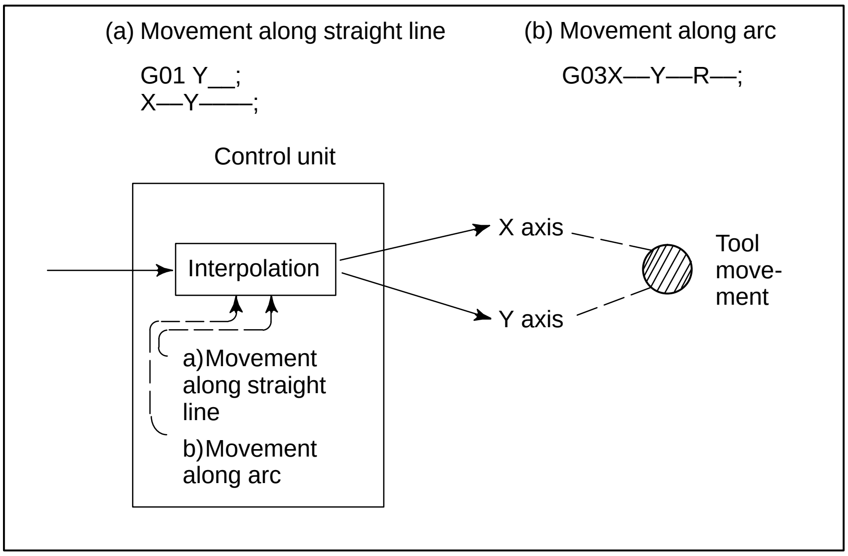


Fig. 1.1 (c) Interpolation function

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

1.2 FEED– FEED FUNCTION

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

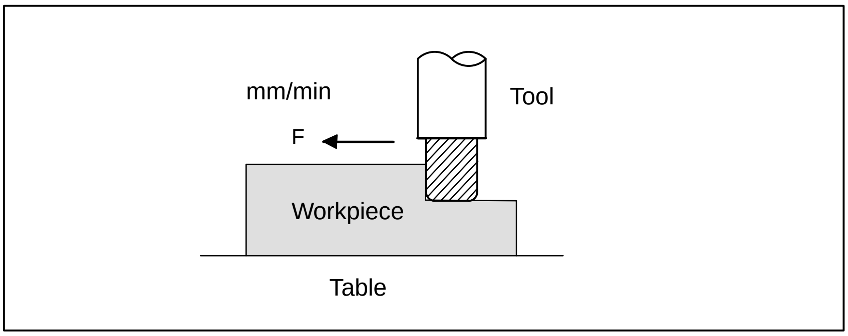


Fig. 1.2 Feed function

Feedrates can be specified by using actual numerics. For example, to feed the tool at a rate of 150 mm/min, specify the following in the program:
F150.0
The function of deciding the feed rate is called the feed function (See II–5).

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine–Specific Position)

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

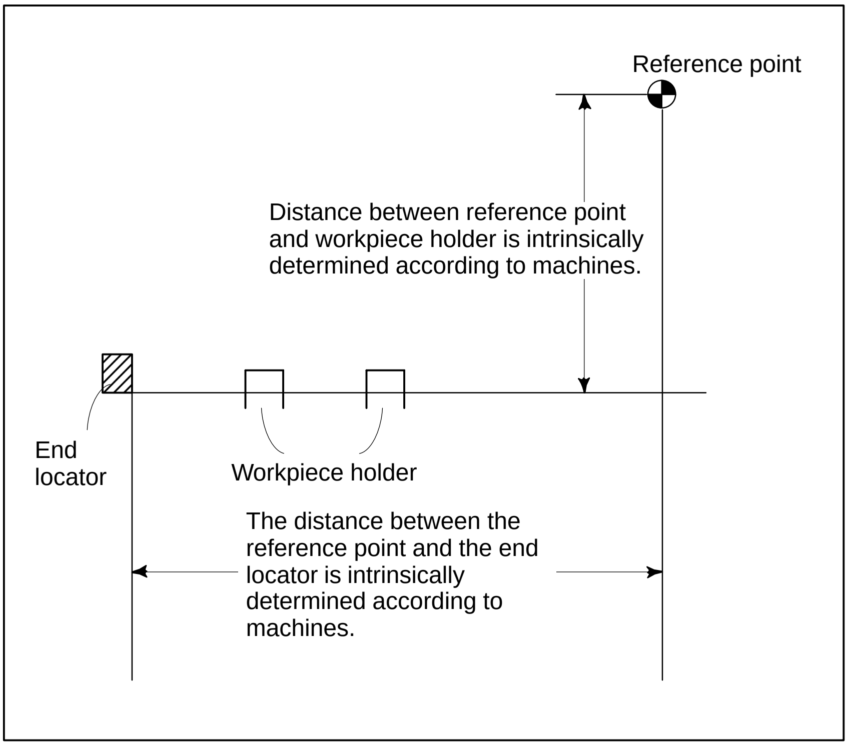


Fig. 1.3.1 Reference position

Explanations

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

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

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

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

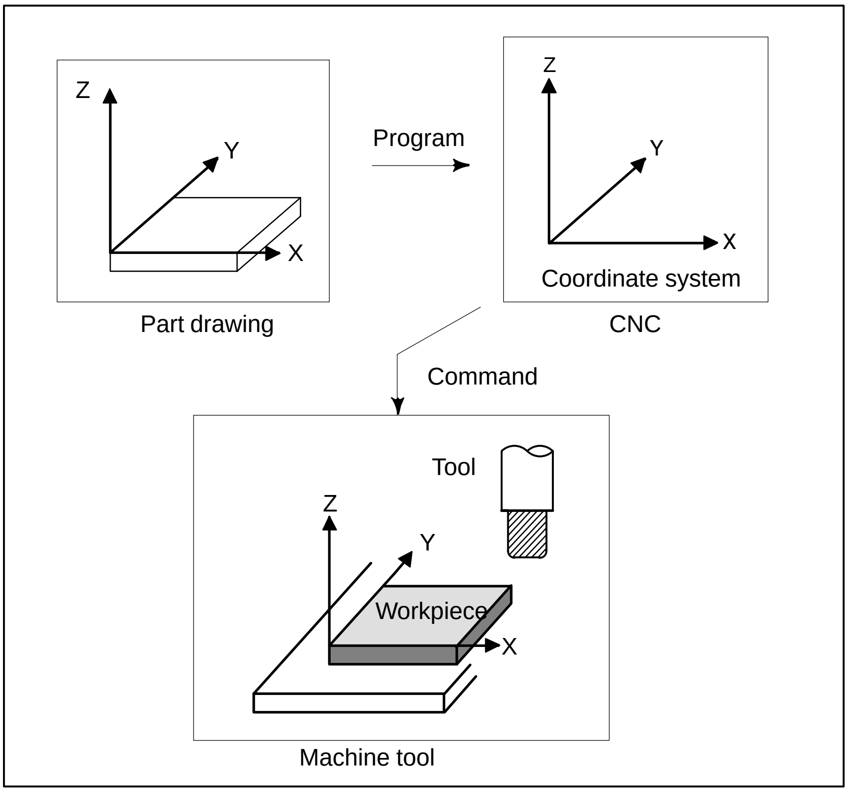


Fig. 1.3.2 (a) Coordinate system

Explanations

- **Coordinate system**

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

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

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

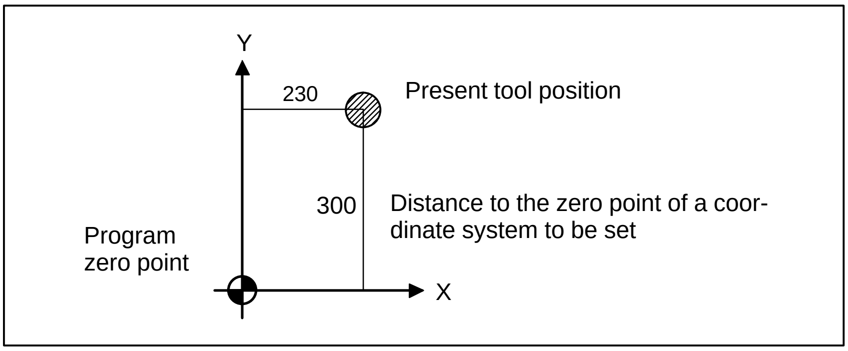


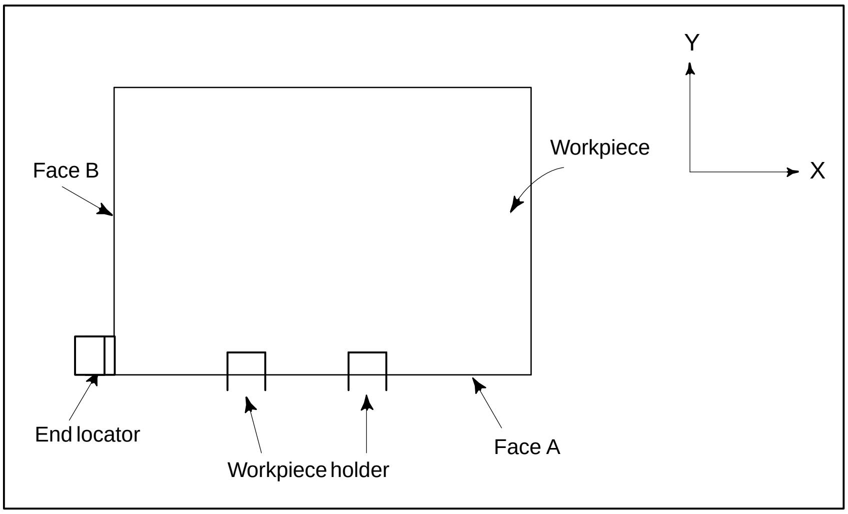
Fig. 1.3.2 (b) Coordinate system specified by the CNC

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

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

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

When a workpiece is set on the table, these two coordinate systems lay as follows:
The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cut 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.
To set the two coordinate systems at the same position, when setting a workpiece to be machined to general turret punch press, the workpiece is held by the workpiece holders after positioning it by applying the end face of the workpiece to the end locator and workpiece holders mounted on the machine as illustrated below.



Generally, the distance between the reference point and the and locator as well as the distance between the reference point and the workpiece holders are intrinsically determined according to machines, and they are separated from each other by a fixed distance.

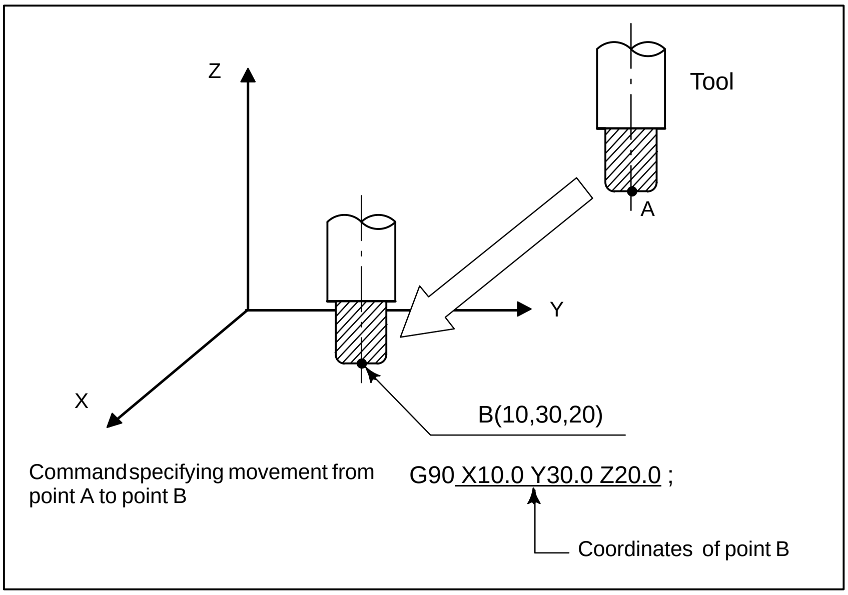
1.3.3 How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands

Explanations

Coordinate values of command for moving the tool can be indicated by absolute or incremental designation (See II–8.1).

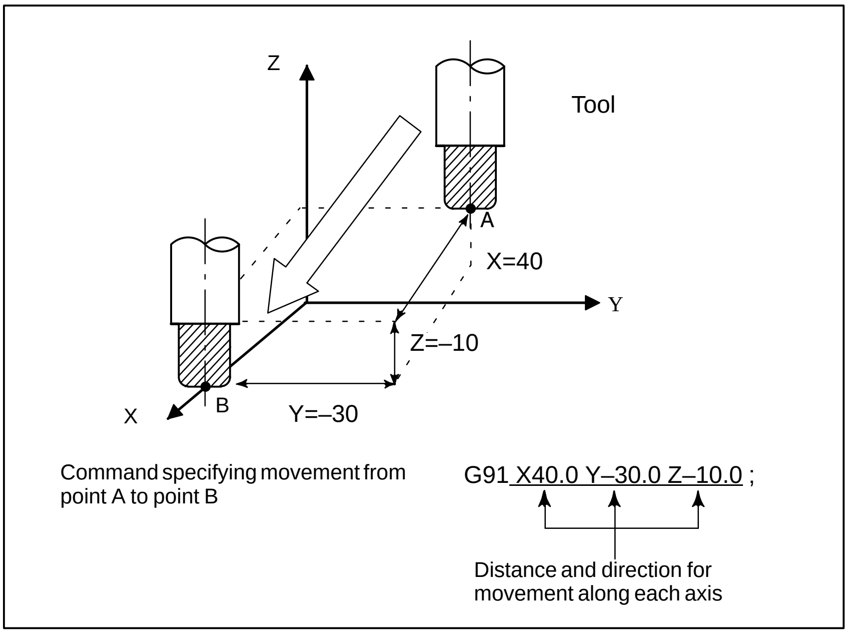
- **Absolute coordinates**

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



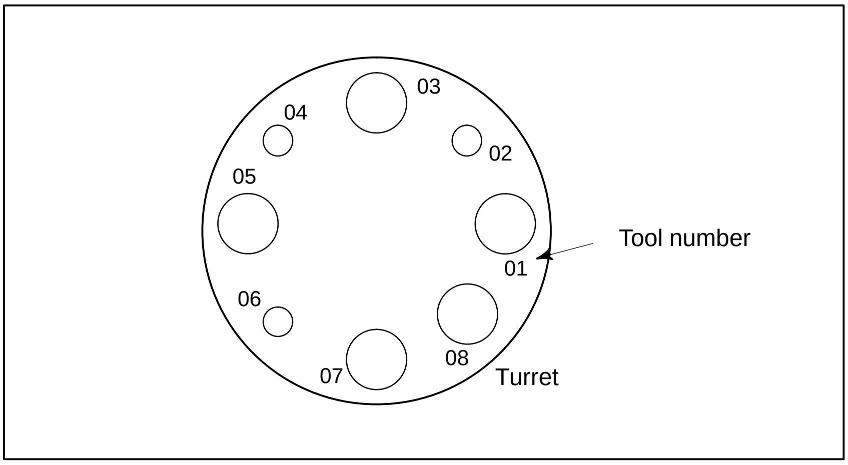
- **Incremental coordinates**

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



1.4 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION

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

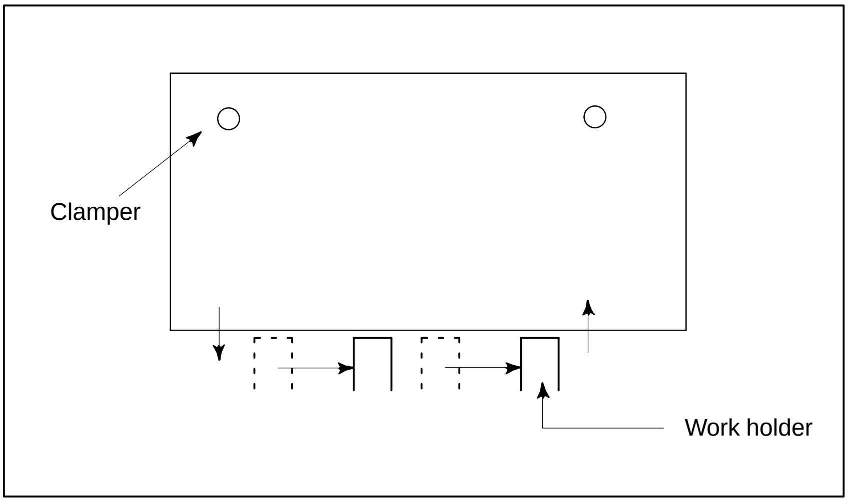


Examples

<When No.01 is assigned to a punching tool>
When the tool is stored at location 01 in the turret, the tool can be selected by specifying T01. This is called the tool function (See II–11).

1.5 COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION

During machining, on–off operation of work holder and clamper is performed.
For this purpose, on–off operations of workholder and clamper should be controlled.



The function of specifying the on–off operations of the components of the machine is called the miscellaneous function. In general, the function is specified by and M code.

1.6 PROGRAM CONFIGURATION

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

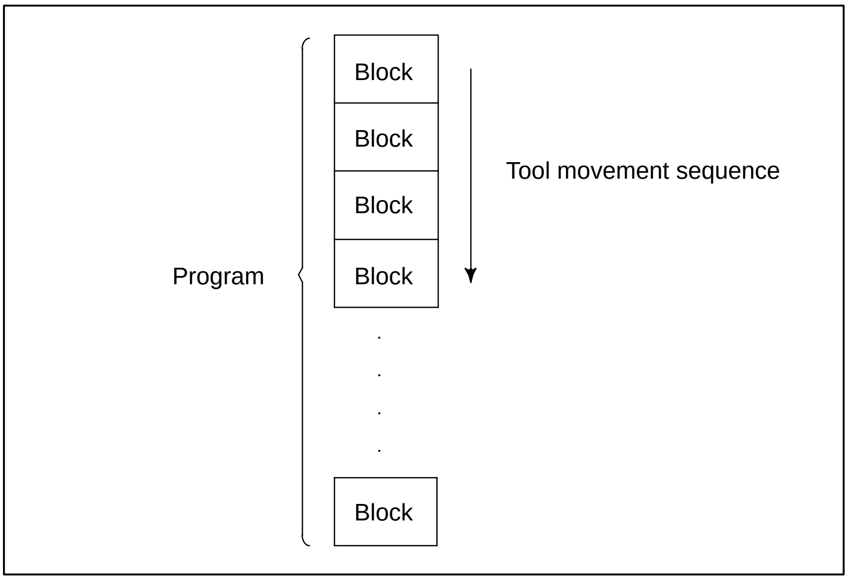


Fig. 1.6 (a) Program configuration

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–13).

Explanations

The block and the program have the following configurations.

• Block

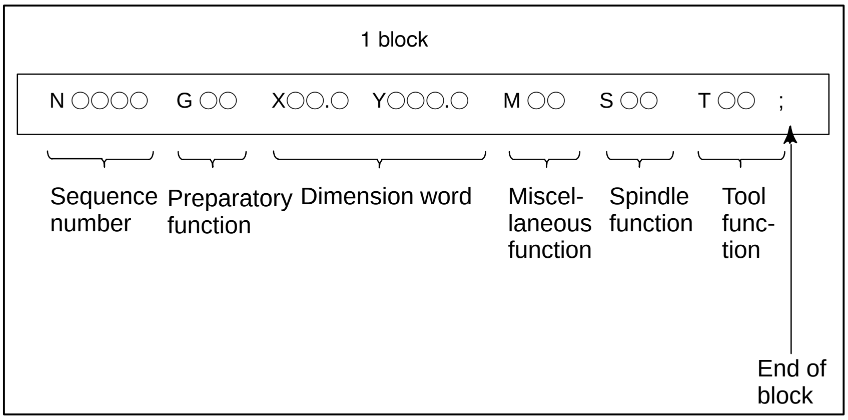


Fig. 1.6 (b) Block configuration

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

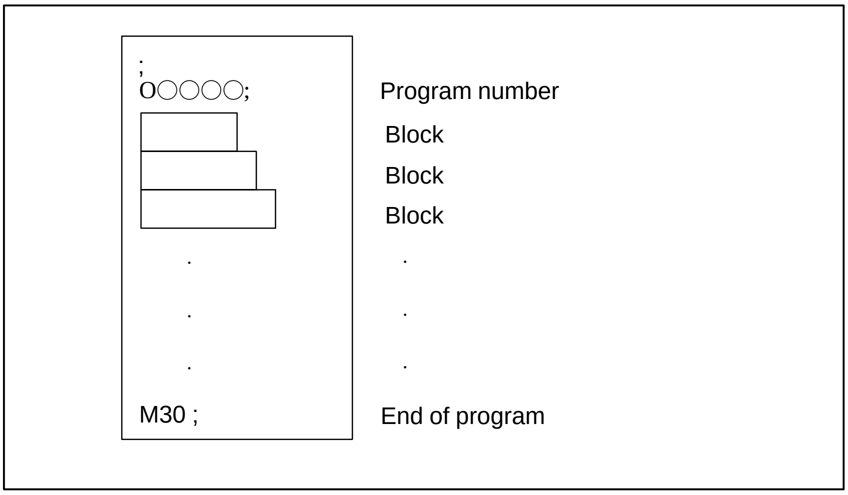
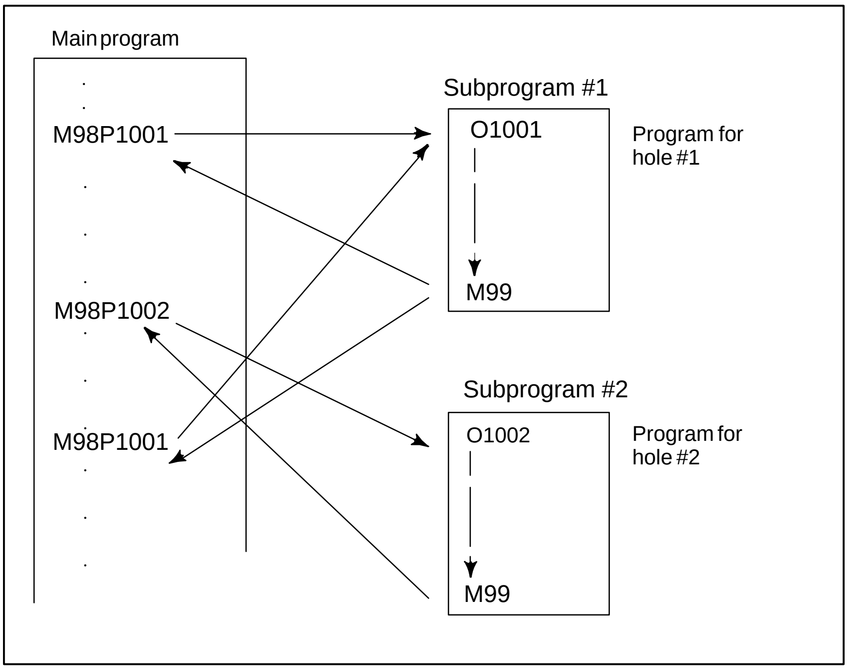


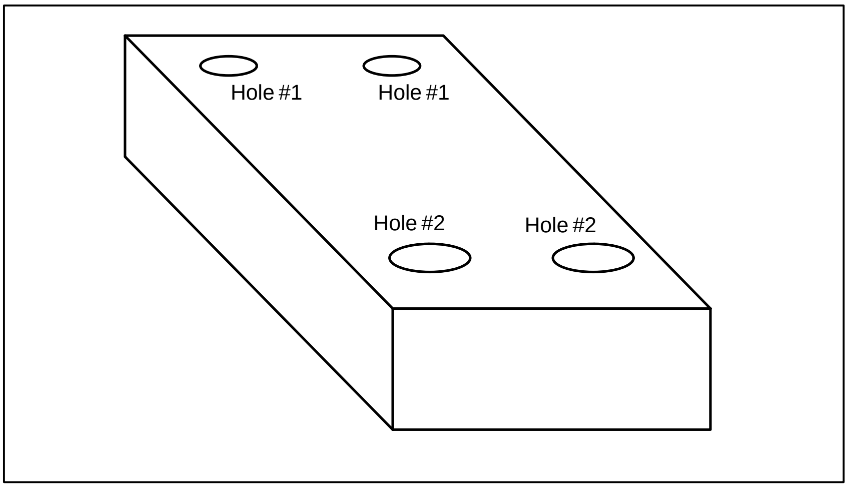
Fig. 1.6 (c) Program configuration

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

- **Main program and subprogram**

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



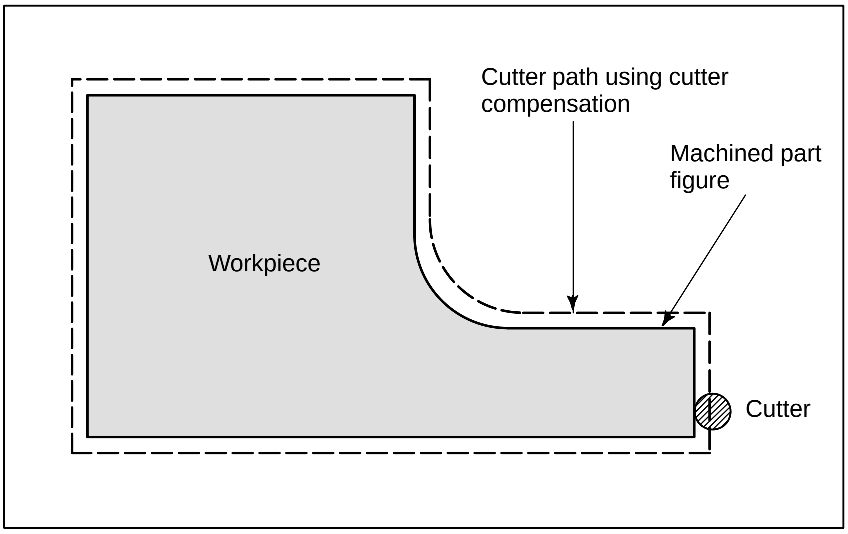


1.7 TOOL FIGURE AND TOOL MOTION BY PROGRAM

Explanations

- **Machining using the side of cutter – Cutter compensation function (See II–15.1, 15.2)**

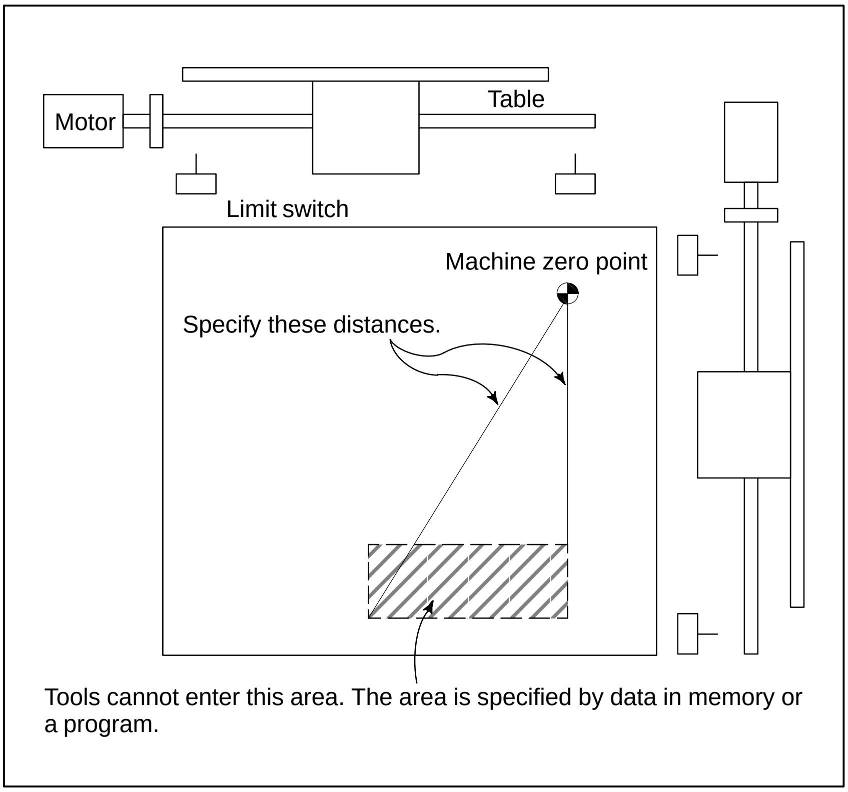
Because a cutter has a radius, the center of the cutter path goes around the workpiece with the cutter radius deviated.



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

1.8 TOOL MOVEMENT RANGE – STROKE

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



Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory (see Section III–11). This function is called stroke check.

2 CONTROLLED AXES

2.1 CONTROLLED AXES

Item	0*i*–PC
No. of basic controlled axes	3 axes
Controlled axes expansion (total)	Max. 4 axes
Basic simultaneously controlled axes	2 axes
Simultaneously controlled axes expansion	Max. 4 axes

2.2 AXIS NAME

The names of the two basic axes are fixed to X and Y, while the names of additional axes can be set to any of A, B, C, U, V, W, and T.
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 4 is assigned by default.

Limitations

- **Default axis name**

When a default axis name (1 to 4) is used, operation in the MEM mode, MDI mode and 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.
If A, B, U, V and W is specified an axis name, the punch press macro function is not available.

2.3 INCREMENT SYSTEM

Name of increment system	Least input increment	Least command increment	Maximum stroke
IS–A	0.01mm 0.001inch 0.01deg	0.01mm 0.001inch 0.01deg	999999.99mm 99999.999inch 999999.99deg
IS–B	0.001mm 0.0001inch 0.001deg	0.001mm 0.0001inch 0.001deg	99999.999mm 9999.9999inch 99999.999deg

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.

Limitations

- T axis is the axis for turret indexing.
- The least input increment is not provided for the turret axis. Neither movement direction nor amount on the turret axis is commanded after address T, but the tool number is commanded. The control system moves the turret axis to the location being preset by a tool registering screen according to the specified tool number, and selects the specified 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–;
    Y–;  } G01 is effective in this range.
    X–;
G00Y–;
```

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)
   For G22 and G23, G22 is set at power–up. 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 (G18) and bit 1 (G19) of parameter No. 3402.
2. G codes other than G10 and G11 are one–shot G codes.
3. When a G code not listed in the G code list is specified, or a G code that has no corresponding option is specified, alarm No. 010 is output.
4. Multiple G codes can be specified in the same block if each G code belongs to a different group. If multiple G codes that belong to the same group are specified in the same block, only the last G code specified is valid.
5. G codes are indicated by group.

Table 3 G code list (1/2)

G code	G code	Group	Meaning	
G00	G00	01	Positioning (Rapid traverse)	
G01	G01		Linear interpolation (Cutting feed)	
G02	G02		Circular interpolation (CW) / Helical interpolation (CW)	
G03	G03		Circular interpolation (CCW) / Helical interpolation (CCW)	
G04	G04	00	Dwell	
G08	G08		Advanced preview control	
G09	G09		Exact stop	
G10	G10	00	Programmable data input	
G11	G11		Programmable data input mode cancel	
G17	G17	02	XpYp plane	Where Xp : X–axis or an axis parallel to it
G18	G18		ZpXp plane	Yp : Y–axis or an axis parallel to it
G19	G19		YpZp plane	Zp : Z–axis or an axis parallel to it
G20	G20	06	Input in inch	
G21	G21		Input in inch	
G22	G22	04	Stored stroke limit function on	
G23	G23		Stored stroke limit function off	
G26	G26	00	Bolt hole circle	
G28	G50		Automatic reference point return	
G32	G32		Automatic safety zone setting	
G33	G33		Skip function	
G38	G38		Bending compensation X	
G39	G39		Bending compensation Y	
G40	G40	07	Cutter compensation cancel	
G41	G41		Cutter compensation left	
G42	G42		Cutter compensation right	
G40.1 (G150)	G40.1 (G150)	19	Normal direction control canceled	
G41.1 (G151)	G41.1 (G151)		Left–side normal direction control turned on	
G42.1 (G152)	G42.1 (G152)		Right–side normal direction control turned on	
G50	G34	11	Scaling on	
G51	G35		Scaling off	
G52	G93	00	Local coordinate system setting	
G53	G53		Machine coordinate system selection	
G54	G54	14	Work coordinates system 1 selection	
G55	G55		Work coordinates system 2 selection	
G56	G56		Work coordinates system 3 selection	
G57	G57		Work coordinates system 4 selection	
G58	G58		Work coordinates system 5 selection	
G59	G59		Work coordinates system 6 selection	

Table 3 G code list (2/2)

G code	G code	Group	Meaning
G61	G61	15	Exact stop mode
G62	G62		Automatic corner override
◤G64	◤G64		Continuous cutting mode
G65	G95	00	Custom macro simple call
G66	G96	12	Custom macro modal call
◤G67	◤G97		Custom macro modal call cancel
G68	G68	00	Circular nibbling
G69	G69		Linear nibbling
G70	G70		Positioning & press off
G72	G72		Standard point command
G73	G75		Multi–piece machining command X
G74	G76		Multi–piece machining command Y
G75	G27		Automatic repositioning
G76	G28		Line at angle
G77	G29		Arc
G78	G36		Grid I
G79	G37		Grid II
G84	G84	16	Coorrdinate rotating on
◤G85	◤G85		Coordinate rotating off
G86	G66	00	Share proof
G87	G67		Square
G88	G78		Radius
G89	G79		Cut at angle
◤G90	◤G90	03	Absolute command
◤G91	◤G91		Incremental command
G92	G92	00	Coordinate system setting
G98	G98		Coordinate system setting (Multi–piece machining)

4 INTERPOLATION FUNCTIONS

4.1 POSITIONING (G00)

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

Format

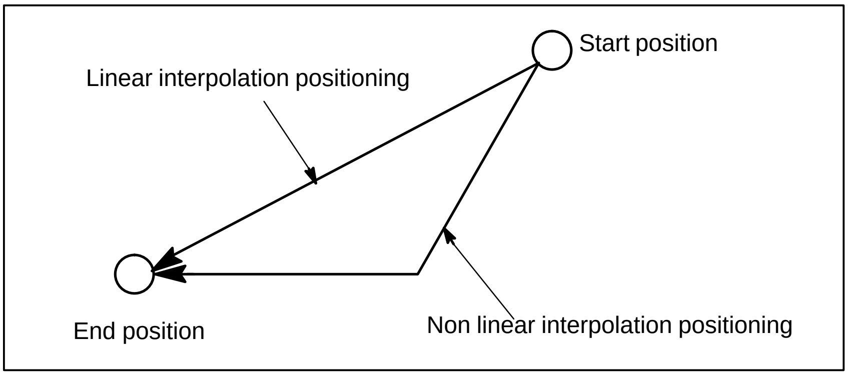
G00 IP_;

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

Explanations

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

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



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

When G00X_Y_T ; is specified in a machine having a turret axis (T–axis), the X and Y axes move to the specified positions at rapid traverse rate and also the T–axis moves at the predetermined rapid traverse rate in such a way as to select a specified tool number.
In a machine provided with a die angle index (C–axis), if “G00X_Y_ ; ” is specified, the X, Y, and C axes move simultaneously at the predetermined rapid traverse rate.
Refer to “C axis control” for the details. Since this control system treats the turret punch press as a controlled system, the tool moves to the commanded position as fast as possible for punching as the basic principle.
Accordingly, the tool is positioned at rapid traverse, punching is done after axis movement in the G00 mode, in principle.
Refer to “Punch function” for details.

The rapid traverse rate in the G00 command is set for each axis independently by the machine tool builder (parameter No. 1420).
Accordingly, the rapid traverse rate cannot be specified in the address F.
In the positioning 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) (Parameter No. 1827)

CAUTION
For T– or C–axis command blocks, nonlinear interpolation positioning is performed, even if linear interpolation positioning is specified.
And, for block including G28 or G53 command, nonlinear interpolation positioning is performed.

4.2 LINEAR INTERPOLATION (G01)

Tools can move along a line

Format

G01 IP_F_;

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

F_:Speed of tool feed (Feedrate)

Explanations

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

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

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 axisβ (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/min.

A calcula;tion 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:

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

The feed rate for the C axis is

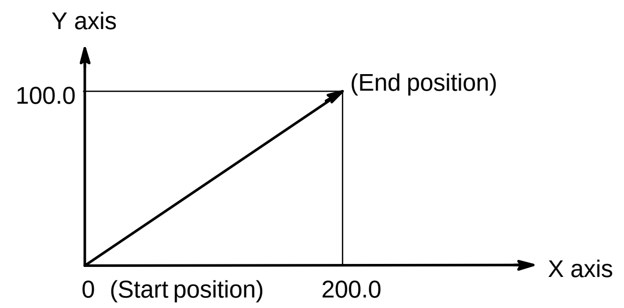
$$\frac{40}{0.14907} \doteqdot 268.3 \text{ deg/min}$$

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 ;

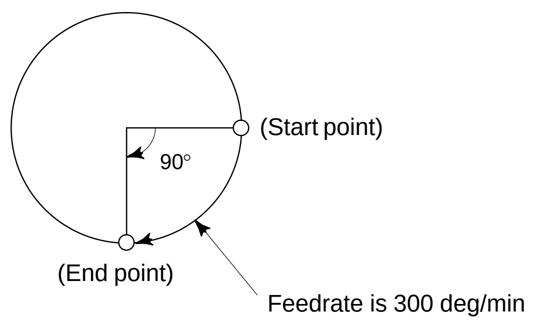


Limitations

- Punching (1–cycle pressing) is not performed in G01 mode.
- T code can’t be specified in G01 mode. If specified, an alarm (No. 4600) occurs.
  However, when T code is specified independently and NMG (No. 16181#0) is set, an alarm does not occur.

• Feedrate for the rotation axis

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



4.3 CIRCULAR INTERPOLATION (G02, G03)

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

Format

Arc in the XpYp plane

$$\mathbf{G17} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \ \mathbf{Xp\_Yp\_} \ \begin{Bmatrix} \mathbf{I\_\ J\_} \\ \mathbf{R\_} \end{Bmatrix} \ \mathbf{F\_\ ;}$$

Arc in the ZpXp plane

$$\mathbf{G18} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \ \mathbf{Xp\_\ p\_} \ \begin{Bmatrix} \mathbf{I\_\ K\_} \\ \mathbf{R\_} \end{Bmatrix} \ F\_$$

Arc in the YpZp plane

$$\mathbf{G19} \begin{Bmatrix} \mathbf{G02} \\ \mathbf{G03} \end{Bmatrix} \ \mathbf{Yp\_\ Zp\_} \ \begin{Bmatrix} \mathbf{J\_\ K\_} \\ \mathbf{R\_} \end{Bmatrix} \ F\_$$

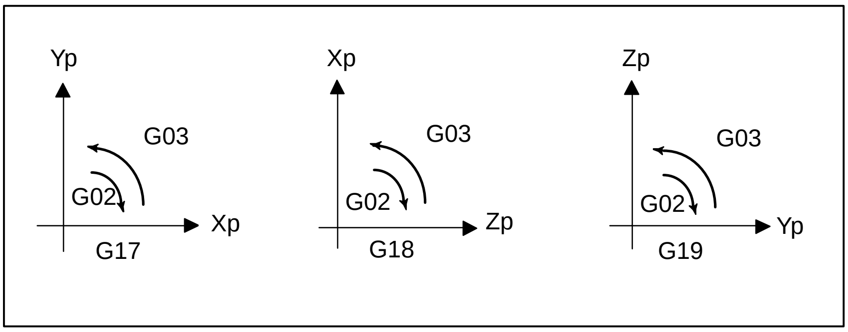
Table. 4.3 Description of the Command Format

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 fixed to radius designation.
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.

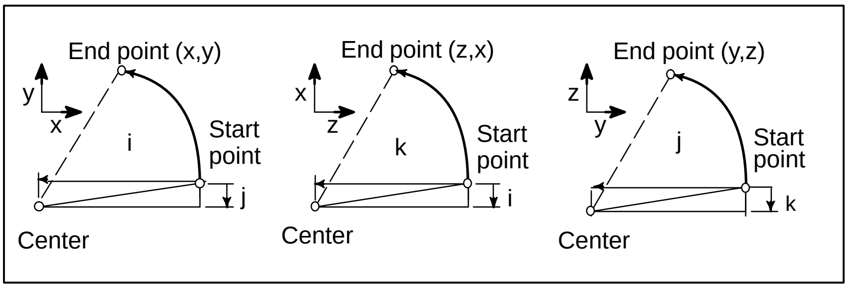


- **Distance moved on an arc**

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.

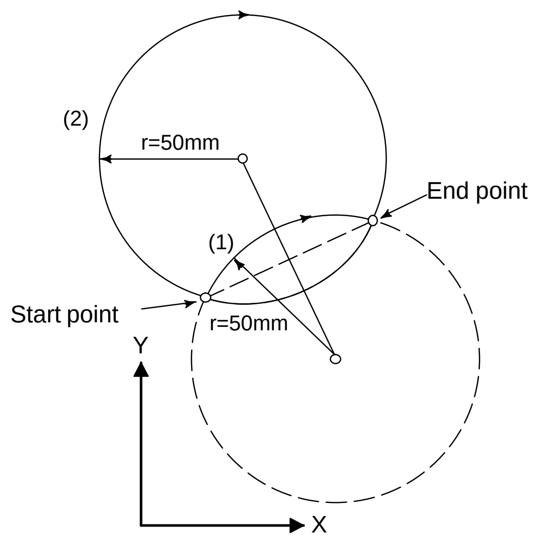


I0,J0, and K0 can be omitted. When X_p, Y_p , and Z_p are omitted (the end point is the same as the start point) and the center is specified with I, J, and K, a 360° arc (circle) is specified.
G02I; Command for a circle
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm (No.024) occurs.

• Arc radius

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I, J, and K. In this case, one arc is less than 180°, and the other is more than 180° are considered. When an arc exceeding 180° is commanded, the radius must be specified with a negative value. If 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 R50.0 F300.0 ;
For arc (2)(greater than 180°)
G91 G02 X_P60.0 Y_P20.0 R–50.0 F300.0 ;



• Feedrate

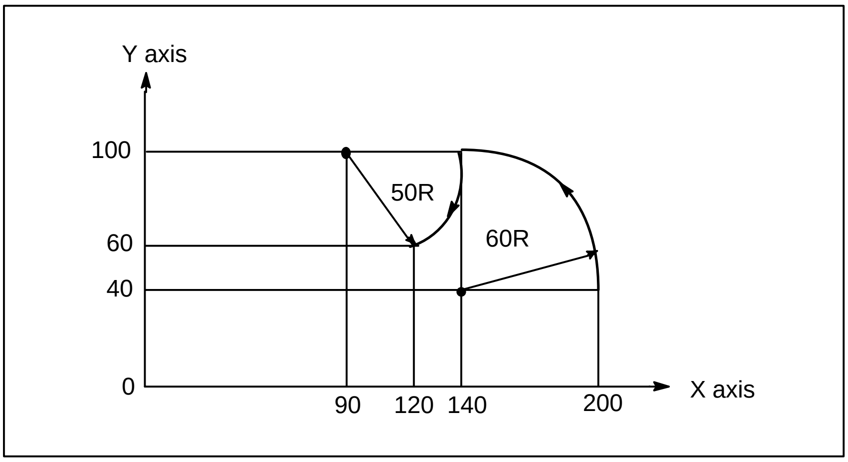
The feedrate in circular interpolation is equal to the feed rate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feed rate is measured along the arc after the cutter compensation is applied

Limitations

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 alarm (No.028)is displayed.

- Punching (1–cycle pressing) is not performed in G02 and G03 mode.
- If T command is specified in G02 and G03 mode, however, when T code is specified independently and NMG (No. 16181#0) is set, an alarm (No. 4600) doesn't occur.

Examples



The above tool path can be programmed as follows ;

(1) In absolute programming

G92X200.0 Y40.0 ;
G90 G03 X140.0 Y100.0R60.0 F300.;
G02 X120.0 Y60.0R50.0 ;
or
G92X200.0 Y40.0 ;
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 ;

4.4 SKIP FUNCTION (G33)

Linear interpolation can be commanded by specifying axial move following the G33 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

G33 IP_ ;

G33: 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 and #5062, as follows:
#5061 X axis coordinate value
#5062 Y axis coordinate value

WARNING
Disable feedrate override, dry run, and automatic acceleration/deceleration (with parameter No. 6200 and subsequent parameters) when the feedrate per minute is specified, allowing for an error in the position of the tool when a skip signal is input.

NOTE
If G33 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 G33 command is specified.

Examples

- **The next block to G33 is an incremental command**

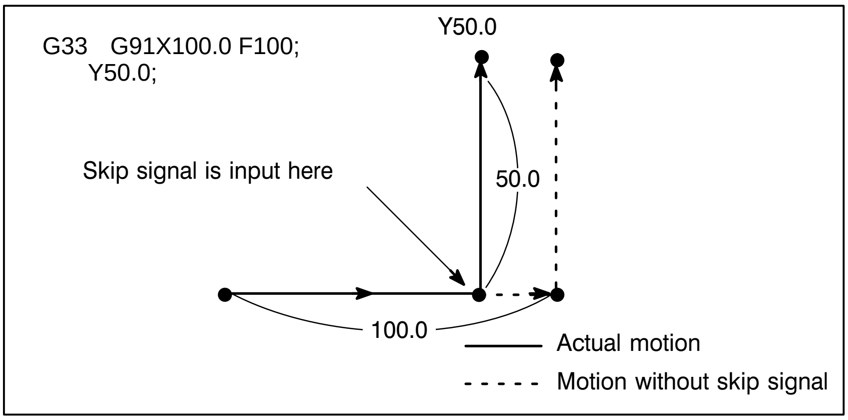


Fig. 4.4 (a) The next block is an incremental command

- **The next block to G33 is an absolute command for 1 axis**

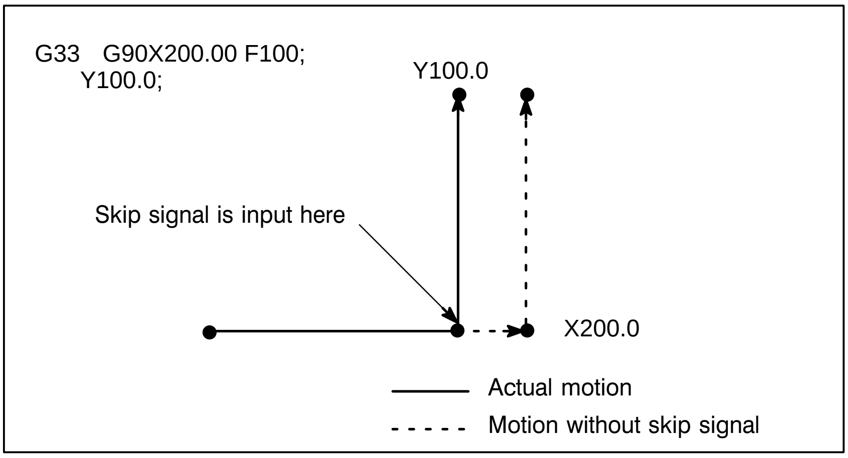


Fig. 4.4 (b) The next block is an absolute command for 1 axis

- **The next block to G33 is an absolute command for 2 axes**

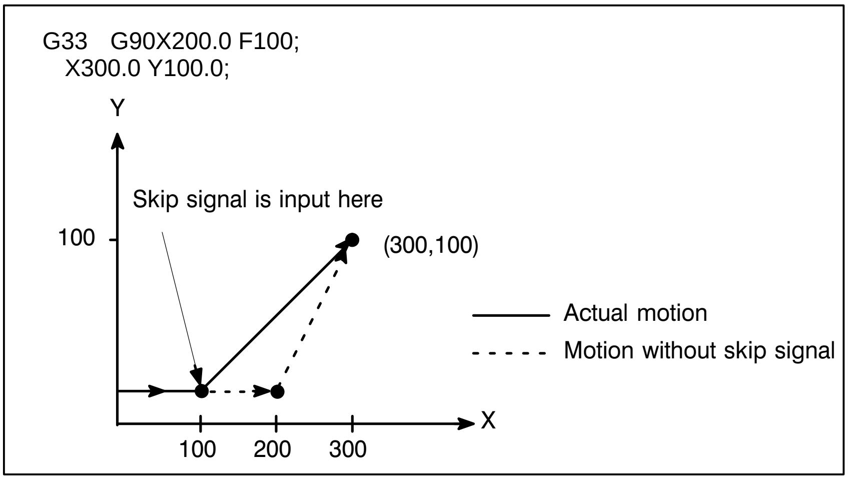


Fig. 4.4 (c) The next block is an absolute command for 2 axes

4.5 HIGH SPEED SKIP SIGNAL (G33)

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

G33 IP_ ;

G33: One–shot G code (If is effective only in the block in which it is specified)

4.6 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

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

Synchronously with arc of ZpXp plane

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

Synchronously with arc of YpZp plane

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

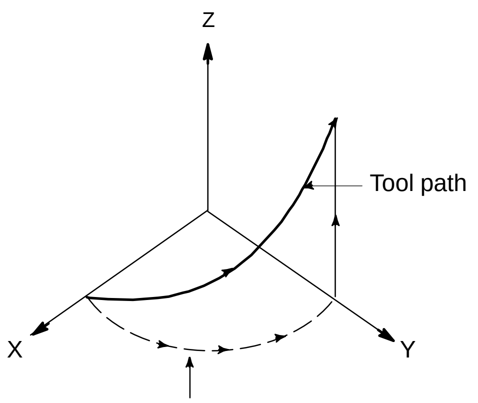
α,β: Any one axis where circular interpolation is not applied.
Up to two other axes can be specified.

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.



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

Restrictions

- Cutter compensation is applied only for a circular arc.
- T axis command and C axis command cannot be used in a block in which a helical interpolation is commanded.

5 FEED FUNCTIONS

5.1 GENERAL

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

- **Feed functions**

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

- **Override**

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

- **Automatic acceleration/ deceleration**

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 5.1 (a)).

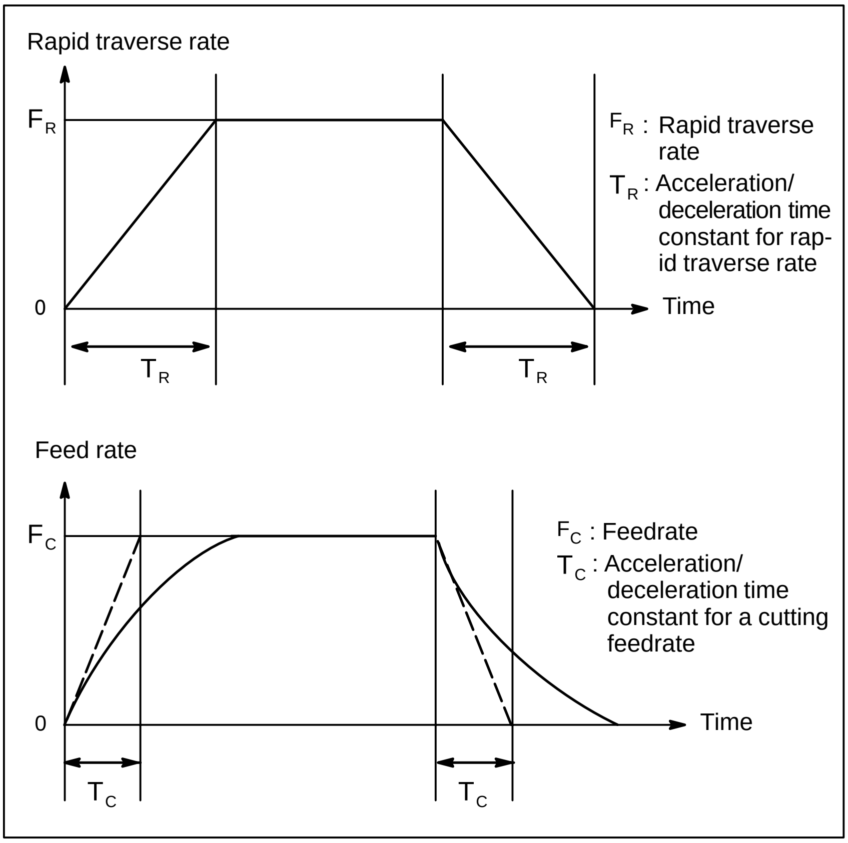


Fig. 5.1 (a) Automatic acceleration/deceleration (example)

- **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)).

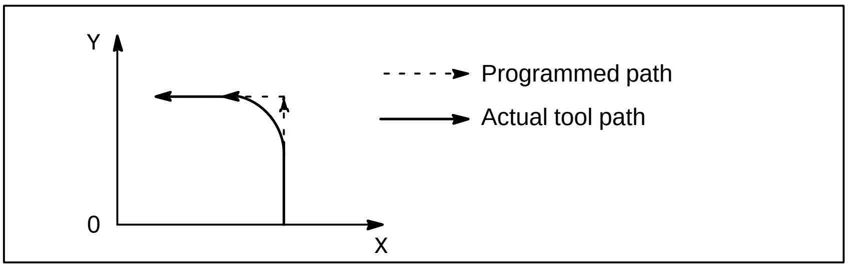


Fig. 5.1 (b) Example of Tool Path between Two Blocks

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

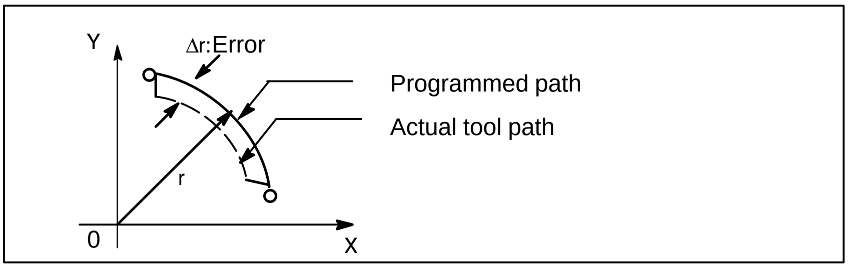


Fig. 5.1 (c) Example of Radial Error in Circular Interpolation

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 tool 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 tool by rapid traverse and punching is performed. 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: 25, 50, 75, 100%

5.2.1 Rapid Traverse Rate by F Command

Each axis rapid traverse rate of rapid traverse command (G00) are set independently to parameter by machine tool builders.

Whereas, by setting parameter G0F (No. 16050#0) to 1, the rapid traverse rate of X and Y axes to rapid traverse command (G00) can be designated by F code. Refer to the manual issued by a machine tool builder for this function.

There are following specifications notices for this function.

1) The feedrate specified by F code is the each axis rapid traverse rate of X and Y axes.
2) 4-step rapid traverse override can be applied to the rapid traverse rate designated by F code, using signals (ROV2, ROV1) from the machine side.
3) When axial move of rapid traverse (G00) is specified in the tape, memory and MDI modes, the rapid traverse rate may not specified by F code or when the speed command is 0, an alarm (No. 011) occurs.
4) In circular nibbling (G68), linear nibbling (G69) and nibbling by M function, the speed to nibbling pitch after the first punch point corresponds to the rapid traverse rate preset by the parameter (No. 1420).
5) F1-digit function for programmable rapid traverse override is ineffective.
6) When the rapid traverse rate designated by F code exceeds the speed preset by a parameter (set by a machine tool builder), it is clamped to the speed preset by the parameter (No. 1420).

5.2.2 Rapid Traverse Override

In the automatic operation, the rapid traverse override is applied to the rapid traverse rate by the switch on the machine operator's panel or F1-digit command (See Subsection 5.2.3).

Either rapid traverse override being set by the switch on the machine operator's panel or rapid traverse override being set by F1-digit command, whichever is lower, becomes effective.

One digit F command	Rapid traverse override switch on machine operator's panel	X-axis, Y-axis	T-axis, C-axis
F1	100%	100%	100%
F2	75%	75%	100%
F3	50%	50%	50%
F4	25%	25%	50%

Examples1

If F3 command is given when the switch on the machine operator's panel is set to 100%, the rapid traverse override of the X and Y axes becomes 50%, and also that of T-axis and C axis becomes 50%.

Examples2

If F1 command is given when the switch on the machine operator's panel is set to 25%, the rapid traverse override of the X and Y axes becomes 25%, while that of T-axis and C axis becomes 50%.

In manual operation mode, the rapid traverse override by the switch on the machine operator's panel and by one-digit F command is ineffective.

WARNING

For the T-axis and C axis, the rapid traverse override can always be set to 100% by setting a parameter TCO (No. 16052#1).

5.2.3 F1-digit (Programmable Rapid Traverse Override)

By specifying one-digit number from 1 to 4 following F, and override can be applied to the rapid traverse rate in automatic operation.

One-digit F command	Rapid traverse override	
	X axis, Y axis	T axis, C axis
F1	100%	100%
F2	75%	100%
F3	50%	50%
F4	25%	50%

An override can be applied to the rapid traverse rate by the switch on the machine operator's panel as well as by F1-digit command in automatic operation.

Either rapid traverse override being set by the switch or the rapid traverse override being set by F1-digit command, whichever lower, becomes effective (see 5.2.2) .

WARNING

1 For the T and C axis, the override can always be set to 100% by setting a parameter TCO (No. 16052#1) .
2 F0 is equivalent to F1, while F5 to F9 are equivalent to F4.
3 When power is turned on, the machine is placed to the F1 command state.
If parameter CLR (No. 3402#6) is set to 1, this F1 state is obtained after depressing the reset button. If CLR is set to 0, the state remains unchanged as before reset.

5.3 CUTTING FEED

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

Format

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

Explanations

- **Tangential speed constant control**

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

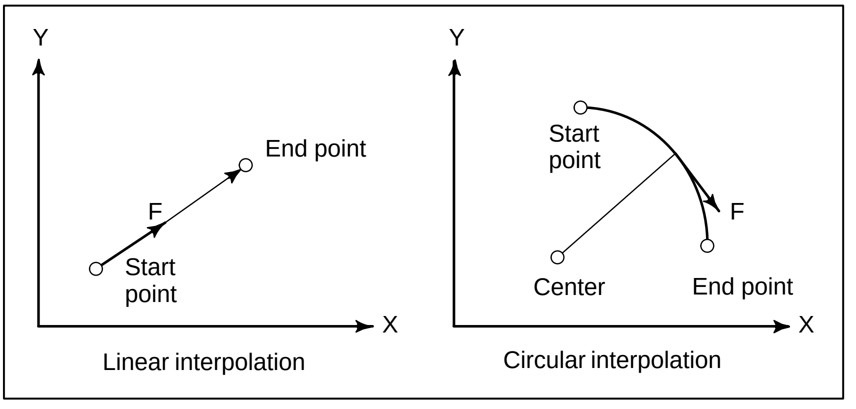


Fig. 5.3 (a) Tangential feedrate (F)

- **Feed per minute**

The amount of feed of the tool 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.

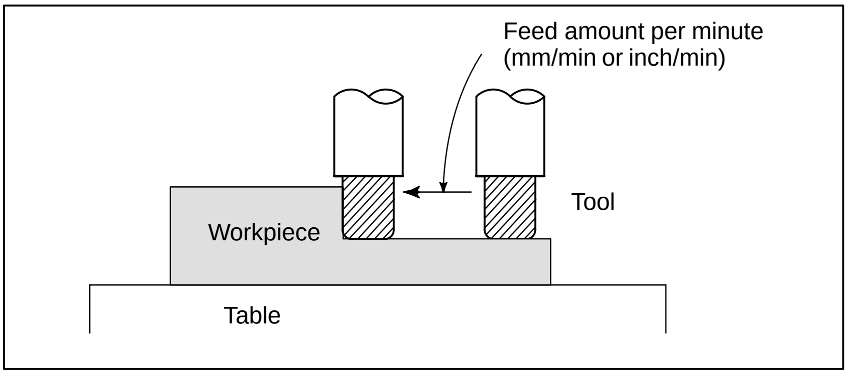


Fig. 5.3 (b) Feed per minute

> **WARNING**
> Cutting feed is invalid for the turret axis (T–axis) and C–axis. T–axis and C–axis commands, therefore, cannot be specified in linear interpolation (G01) mode and circular interpolation (G02, G03) mode.
> However, when the parameter CIP (No.16360#5) is set to 1, C–axis can be specified.

- **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:

5.4 CUTTING FEEDRATE CONTROL

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

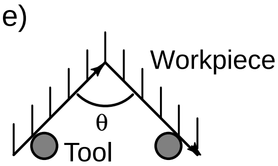
Table 5.4 Cutting Feedrate Control

Function name		G code	Validity of G code	Description
Exact stop		G09	This function is valid for specified blocks only.	The tool is decelerated at the end point of a block, then an in–position check is made. Then the next block is executed.
Exact stop		G61	Once specified, this function is valid until G62 or G64 is specified.	The tool is decelerated at the end point of a block, then an in–position check is made. Then the next block is executed.
Cutting mode		G64	Once specified, this function is valid until G61 or G62 is specified.	The tool is not decelerated at the end point of a block, but the next block is executed.
Auto–matic	Automatic override for inner corners	G62	Once specified, this function is valid until G61 or G64 is specified.	When the tool moves along an inner corner during cutter compensation, override 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 cutting feedrate change	G62	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). When parameter NCI (No. 1601#5) is set tool, in–position check is not executed.

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



Format

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

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

Explanations

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

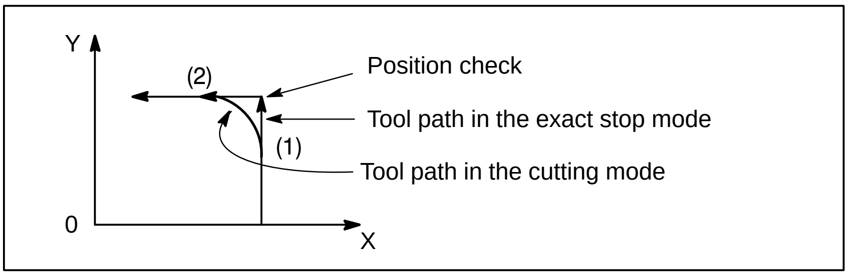


Fig. 5.4.1 Example of Tool Paths from Block (1) to Block (2)

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

5.4.2 Automatic Corner Override

This function enables producing a smooth cutting surface by decelerating tool movement automatically between an inside corner and an inside arc to reduce the load on the cutter during cutter compensation.

5.4.2.1 Inside–corner Override (G62)

Explanations

- **Override condition**

When G62 is specified, and the tool 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.

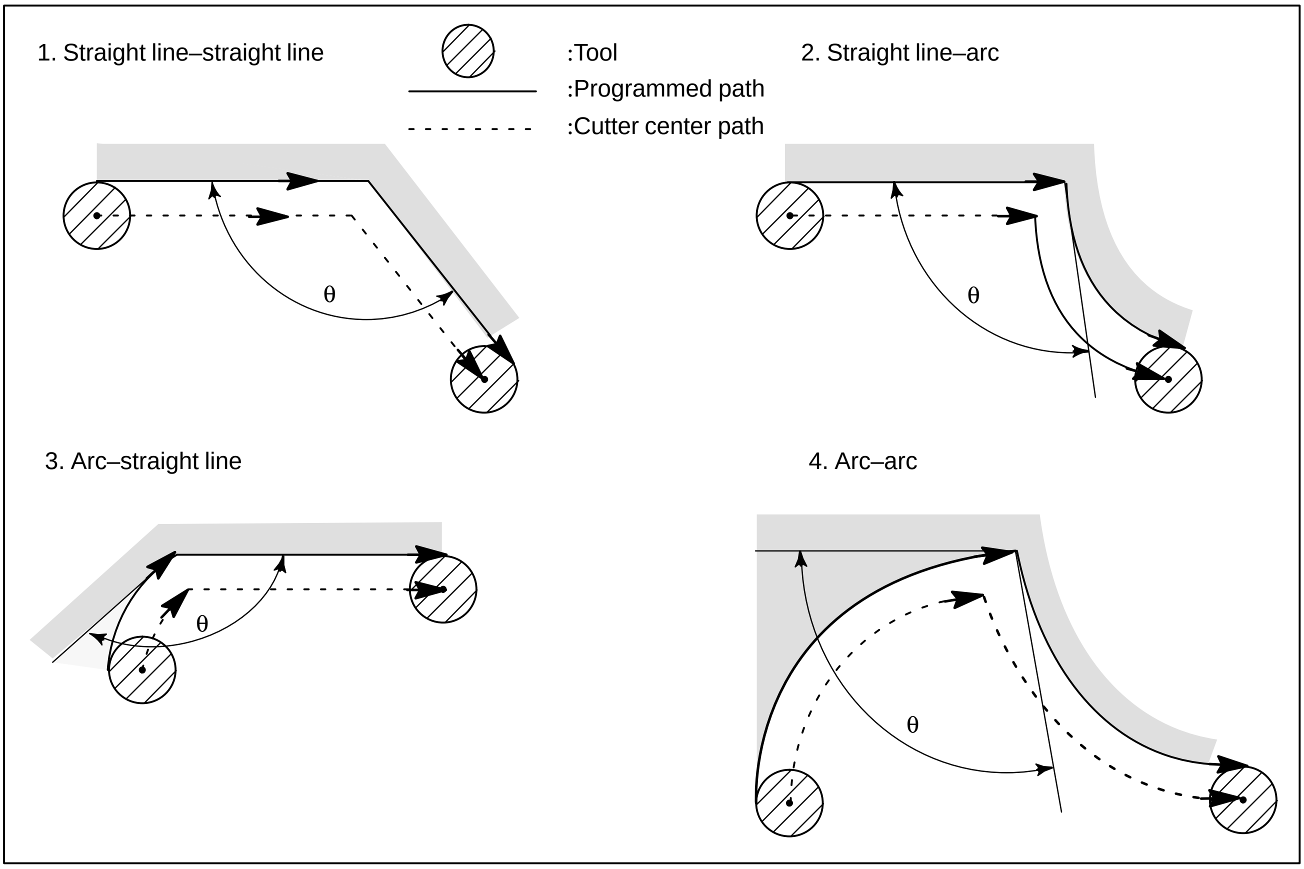


Fig. 5.4.2.1 (a) Inner corner

WARNING
When the block before a corner is a start–up block, or the block after a corner includes G41 or G42, the feedrate is not overridden. The feedrate override function is disabled when the offset value is 0.

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 cutter 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.

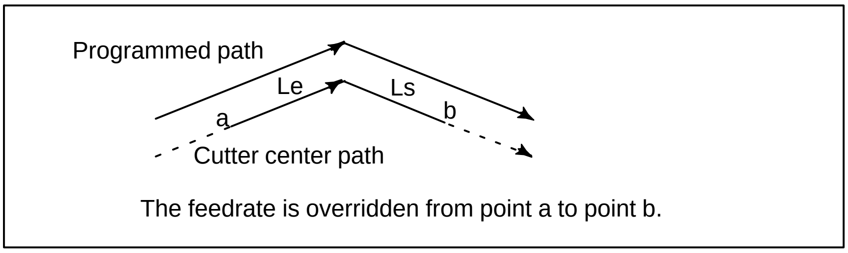


Fig. 5.4.2.1 (b) Override Range (Straight Line to Straight Line)

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)).

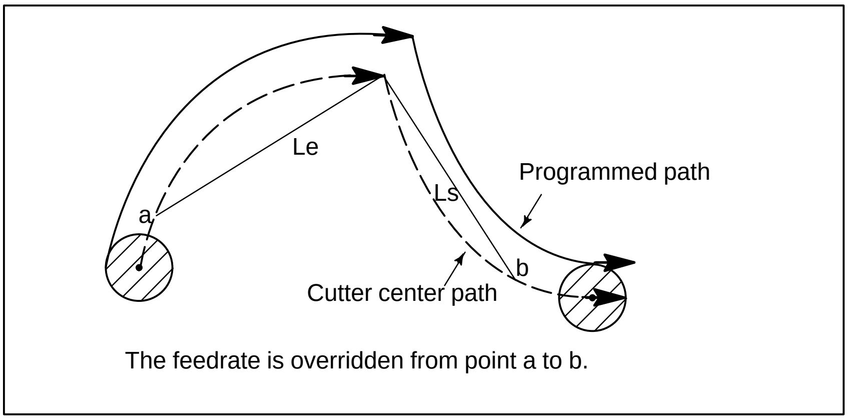


Fig. 5.4.2.1 (c) Override Range (Arc to Arc)

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)).

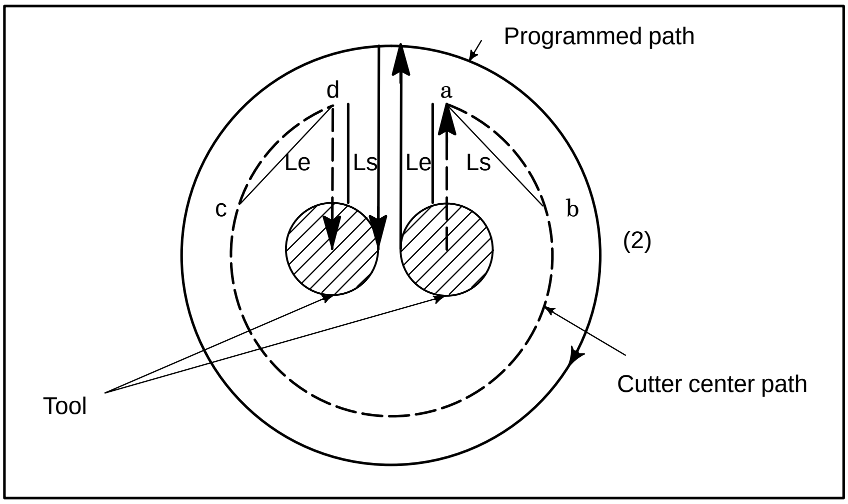


Fig. 5.4.2.1 (d) Override Range (Straight Line to Arc, Arc to Straight Line)

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)

Restrictions

- **Pre–interpolation acceleration/deceleration**

The inside–corner override function is disabled during pre–interpolation acceleration/deceleration.

- **Start–up or G41 and G42**

The inside–corner override function is disabled if a block before the corner is a start–up block for cutter compensation or a block after the corner contains G41 or G42.

- **Offset data**

The inside–corner override function is disabled if the offset data is 0.

5.4.2.2 Internal Circular Cutting 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 : Cutter center path radius
Rp : Programmed radius

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

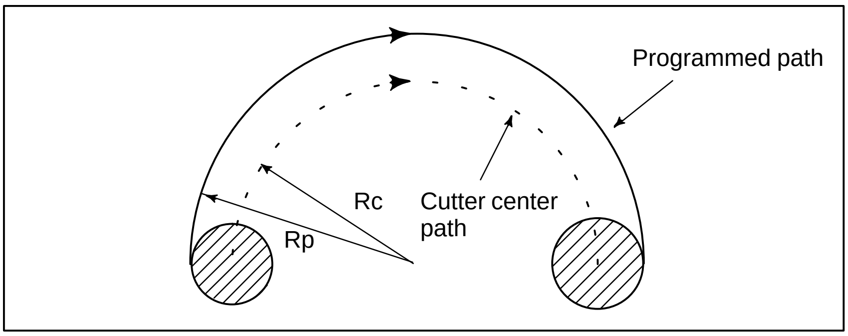


Fig. 5.4.2.2 Internal circular cutting feedrate change

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

> **WARNING**
> When internal circular cutting must be performed together with automatic override for inner corners, the feedrate of the tool is as follows:
> $$F \times \frac{Rc}{Rp} \times \text{(automatic override for the inner corners)} \times \text{(feedrate override)}$$

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.

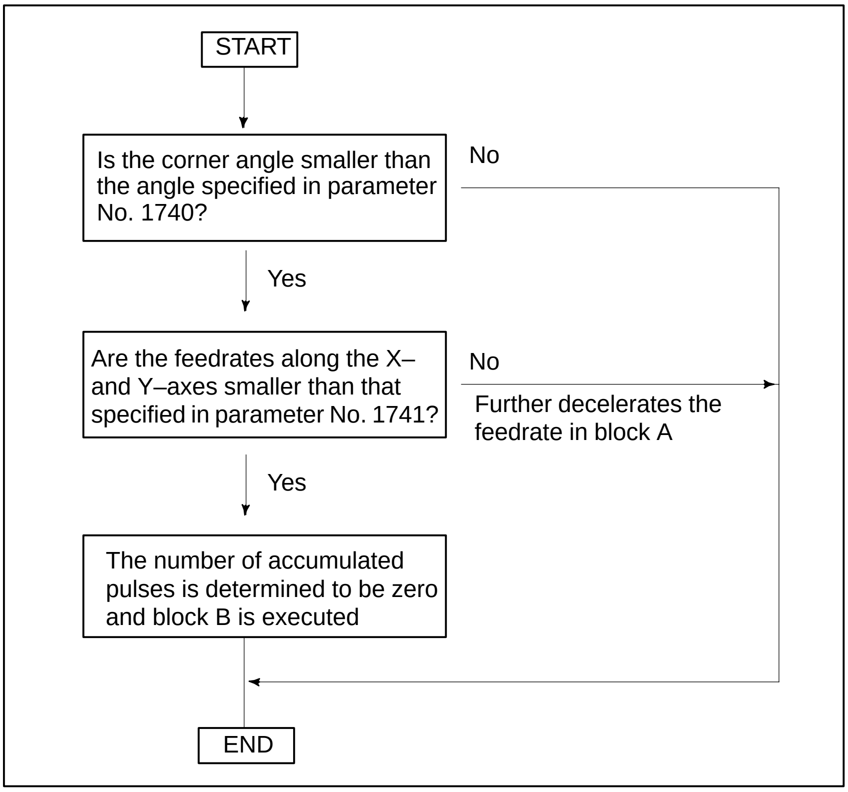
5.4.3.1 Corner deceleration according to the corner angle

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

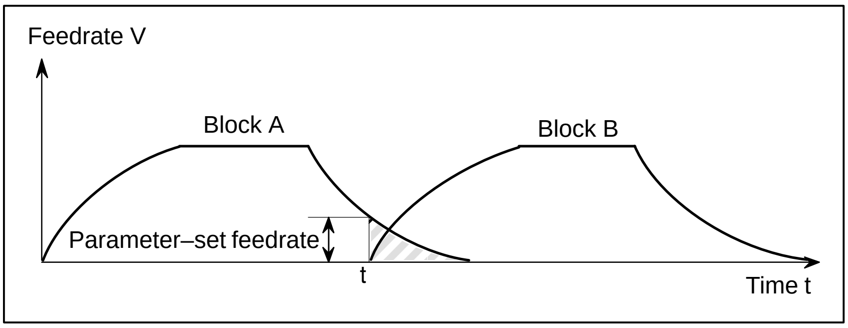
- **Flowchart for feedrate control**

The flowchart for feedrate control is shown below.



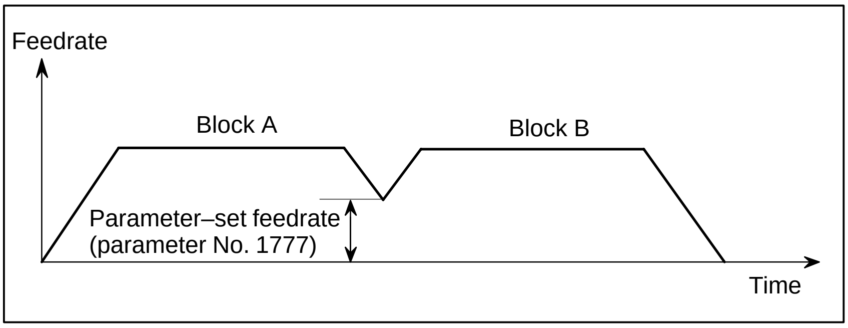
- **Feedrate and time**

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.



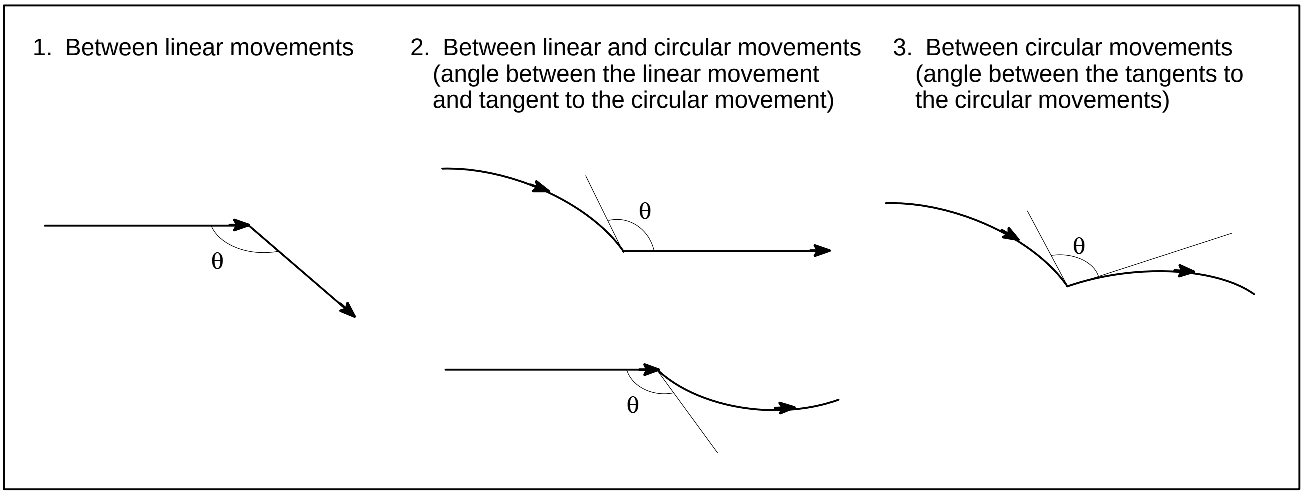
• Acceleration/ deceleration before interpolation

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	1602#4	1602#4
Lower limit of feedrate in automatic corner deceleration based on the angle	1777	1778
Limit angle in corner deceleration based on the angle	1740	1779

Limitations

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

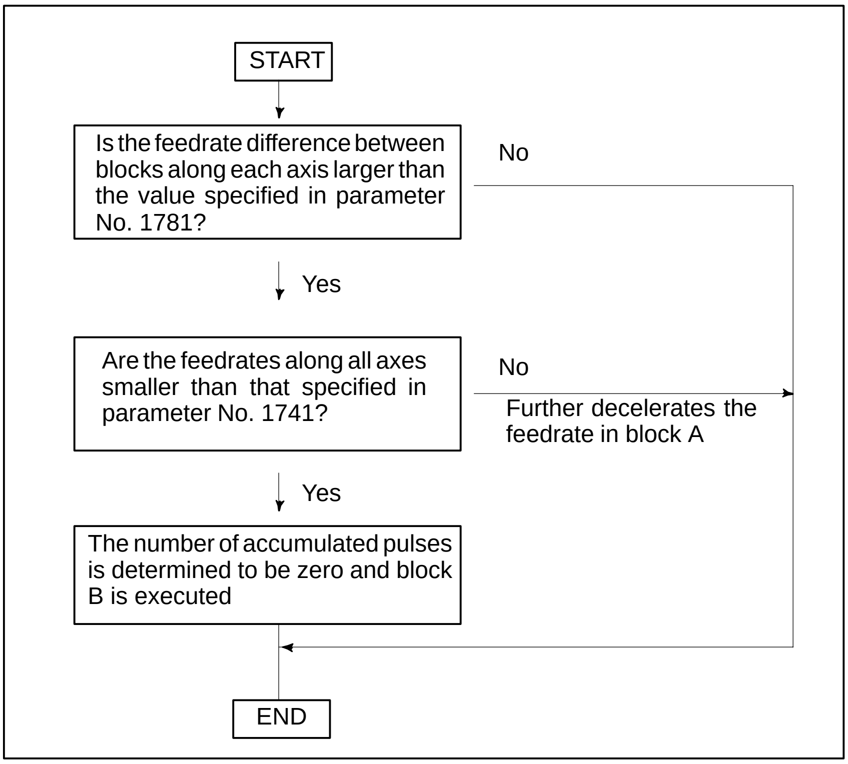
5.4.3.2 Corner deceleration according to the feedrate difference between blocks along each axis

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

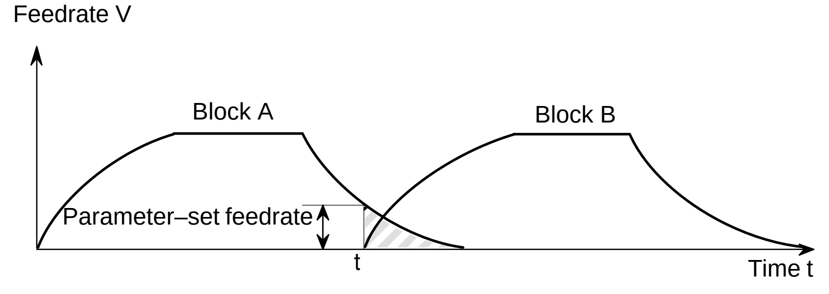
- **Flowchart for feedrate control**

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

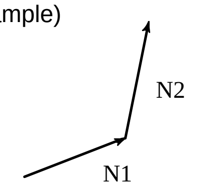
When acceleration/deceleration before interpolation is effective, the relationship between the feedrate and time is as described below.
When the feedrate difference between blocks A and B along each axis is larger than the value specified in parameter No. 1780, the feedrate is decelerated to the corner 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:

$$Fc=F^*\frac{1}{Rmax}$$

(Example)



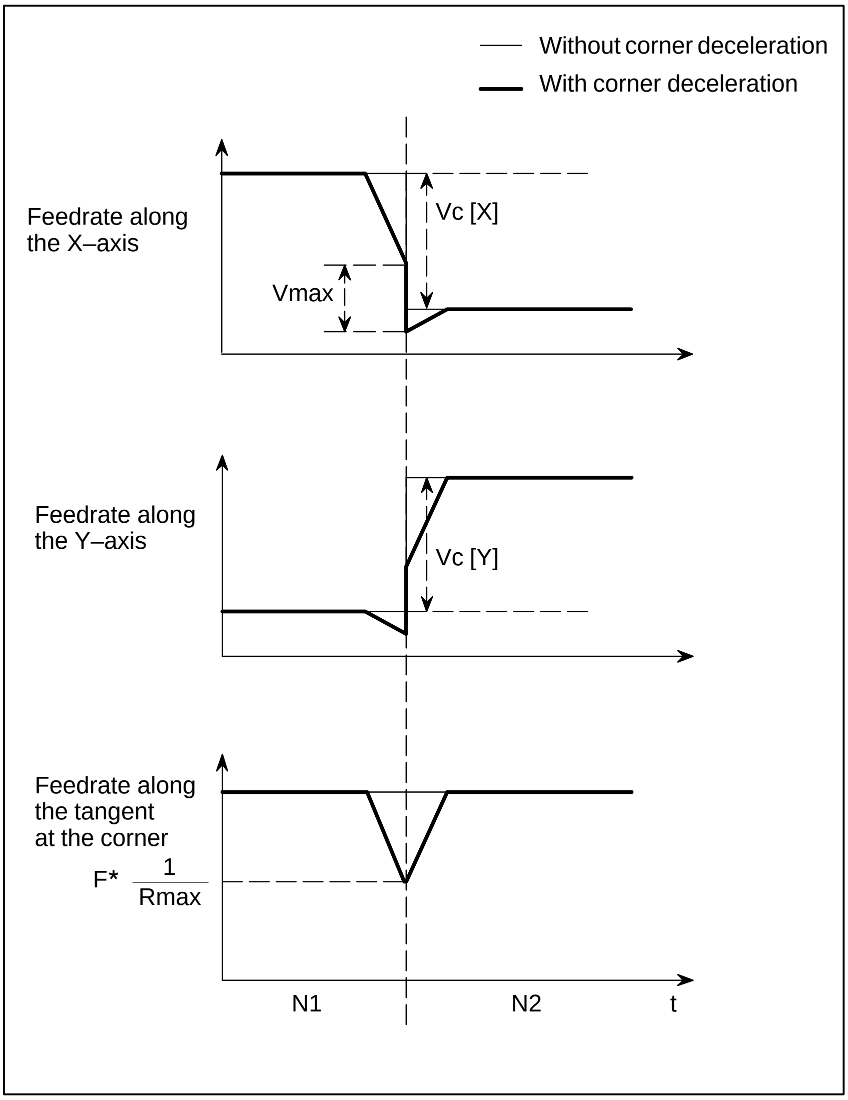
N1 G01 G91 X80. Y20. F3000 ;
N2 X20. Y80. ;

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

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

$$F^*\frac{1}{Rmax}$$

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	1602#4	←
Allowable feedrate difference (for all axis) in automatic corner deceleration based on the feedrate difference	1780	←
Allowable feedrate difference (for each axis) in automatic corner deceleration based on the feedrate difference	1783	←

Limitations

This function is not effective for a single block.

5.5 DWELL (G04)

Format

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

Explanations

By specifying a dwell, the execution of the next block is delayed by the specified time. In addition, a dwell can be specified to make an exact check in the cutting mode (G62 mode).
When neither P nor X is specified, exact stop is performed.

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

Increment system	Command value range	Dwell time unit
IS–A	0.01 to 999999.99	s
IS–B	0.001 to 99999.999	

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

Increment system	Command value range	Dwell time unit
IS–A	1 to 99999999	0.01 s
IS–B	1 to 99999999	0.001 s

6 REFERENCE POSITION

6.1 REFERENCE POSITION RETURN

- **Reference position**

The reference position is a certain fixed point on the machine. It is defined as the point, to which a tool can be moved easily by the reference point return.

When setting a workpiece to be machined to general turret punch press, the workpiece is held by the workpiece holders after positioning it by applying the end face of the workpiece to the end locator and workpiece holders mounted on the machine as illustrated below.

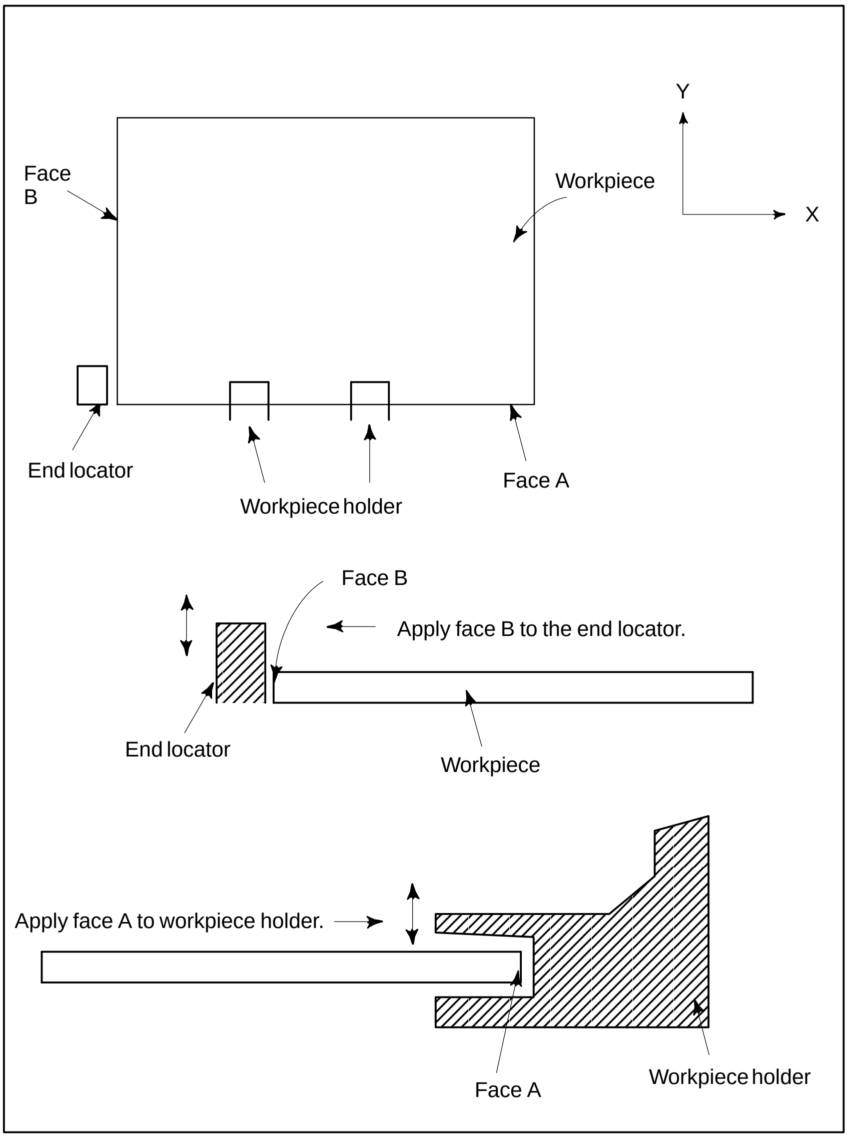


Fig. 6.1 (a)

Generally, the distance between the reference position and the end locator as well as the distance between the reference position and the workpiece holders are intrinsically determined according to machines, and they are separated from each other by a fixed distance.

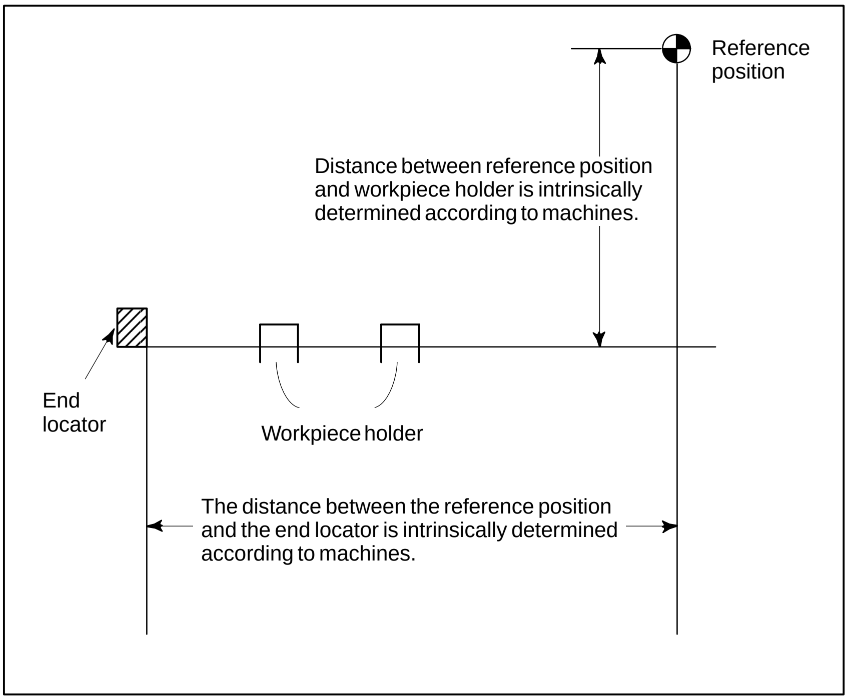


Fig. 6.1 (b)

Accordingly, if the start point is at the reference position and the point located at the left lower side of the workpiece is presumed as the zero point of the workpiece coordinate system, tool position at the start point can be taught to NC as a position in the workpiece coordinate system by giving the following command at the initial stage of programming.

G92X $\underline{x_R}$ Y $\underline{y_R}$;

where, x_R : Distance from end locator to reference position along X-axis

y_R : Distance from workpiece holder to reference position along Y-axis

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

Tools are automatically moved to the reference position. When reference position return is completed, the lamp for indicating the completion of return goes on.

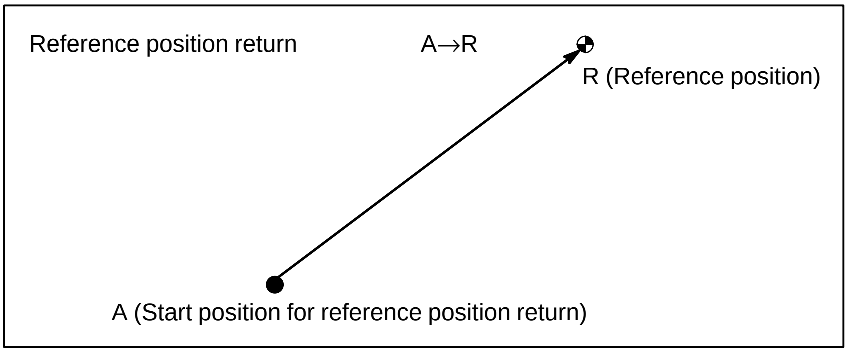


Fig. 6.1 (c) Reference position return

Format

- **Reference position return**

G28_ ;	**Reference position return**

Explanations

- **Reference position return (G28)**

Reference positions are performed at the rapid traverse rate of each axis. When using this command, usually cancel the cutter compensation.
Example G28 M30;

Limitations

- **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.

- **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.

- **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 1μ. This is because the least input increment of the machine tool system is smaller than its least command increment.

Reference

- **Manual reference position return**

See III–3.1.

7 COORDINATE SYSTEM

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

X_Y_Z_

This command is referred to as a dimension word.

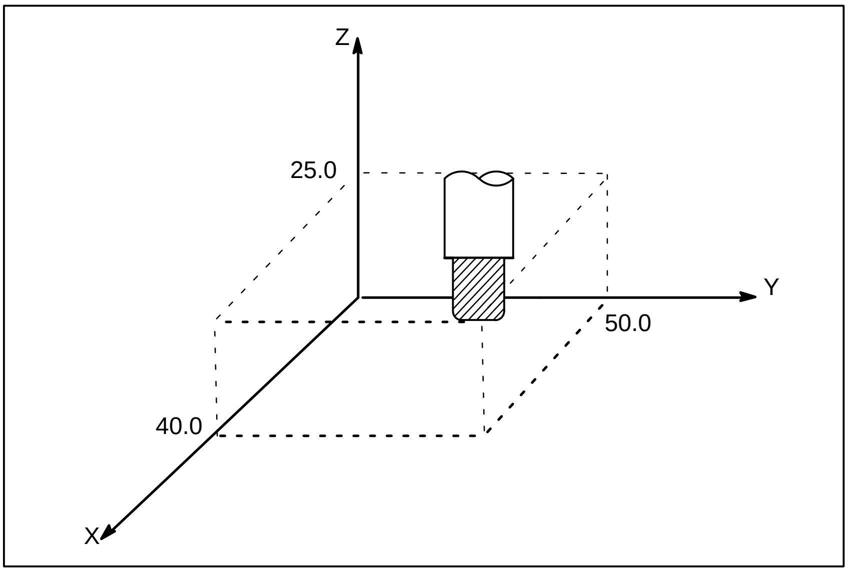


Fig. 7 Tool position specified by X40.0Y50.0Z25.0

Coordinates are specified in one of following three coordinate systems:

(1) Machine coordinate system

(2) Workpiece coordinate system

(3) Local coordinate system

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

7.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine 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

G53 IP_ ;

IP_ : Absolute dimension word

Explanations

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

When a command is specified based on a machine coordinate system, the tool moves 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. The absolute command (G90) is valid. If the incremental command (G91) is specified, G53 is not executed. When the tool is to be moved to a machine–specific position such as a tool change position, program the movement in a machine coordinate system based on G53.

Limitations

- **Cancel of the compensation function**

When the G53 command is specified, cancel the cutter compensation, 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.

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 NC 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 No. 1201 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed (see Part III–3.1.).

(3) Method using G54 to G59
After six workpiece coordinate systems are set from the MDI panel, the program commands G54 to G59 are used to select which workpiece coordinate system is used (see III–11.4.7). Before specifying the absolute command, use one of the above methods to establish the workpiece coordinate system.

Format

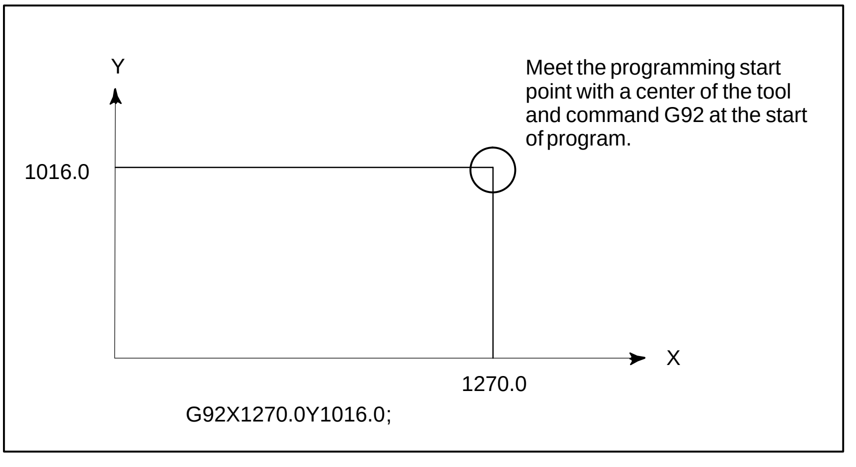
- **Setting a workpiece coordinate system by G92**

(G90) G92 IP_

Explanations

A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates.
Cutter compensation is cancelled temporarily with G92.
M.S and T code cannot be specified in G92 block.

Examples



7.2.2 Selecting a Workpiece Coordinate System

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

(1) Selecting a workpiece coordinate system set by G92 or automatic workpiece coordinate system setting

Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

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

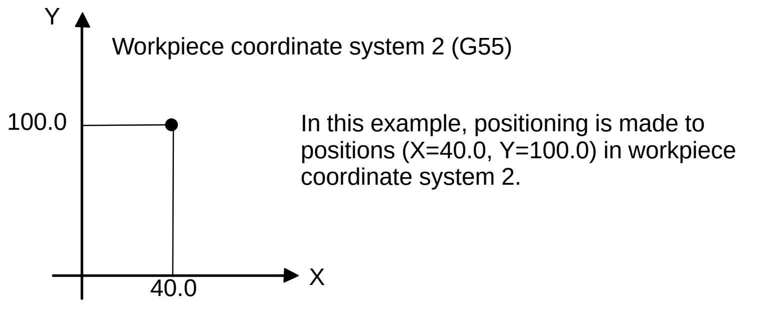


Fig. 7.2.2

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–11.4.7)
(2) Programming by G10 or G92
(3) Changing an external workpiece zero point offset value (refer to machine tool builder's manual)

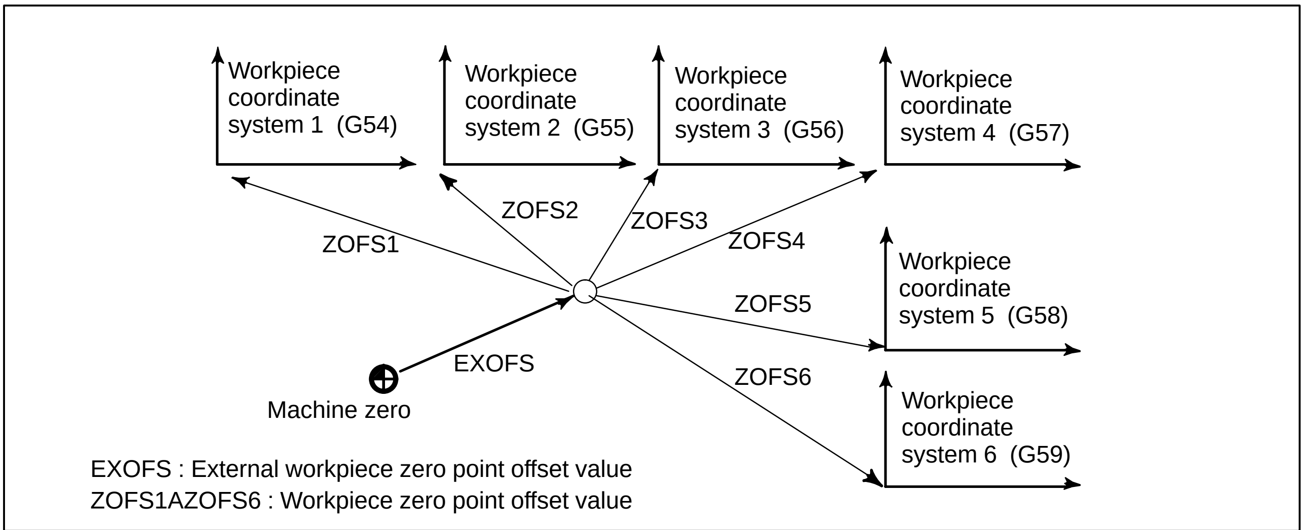


Fig. 7.2.3 Changing an external workpiece zero point offset value or workpiece zero point offset value

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**

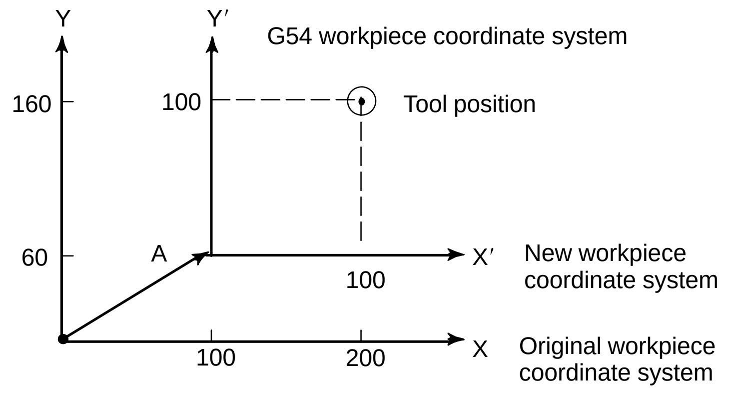
By specifying G92 IP_;, a workpiece coordinate system (selected with a code from G54 to G59) is shifted to set a new workpiece coordinate system so that the current tool position matches the specified coordinates (IP_).

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 G92X100.0Y80.0; is specified, for example, the coordinate system having its current tool reference position at X = 100.0 and Y = 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.



7.3 LOCAL COORDINATE SYSTEM

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

Format

G52 IP_; Setting the local coordinate system
......
G52 IP 0 ; Canceling of the local coordinate system
IP_ : Origin of the local coordinate system

Explanations

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

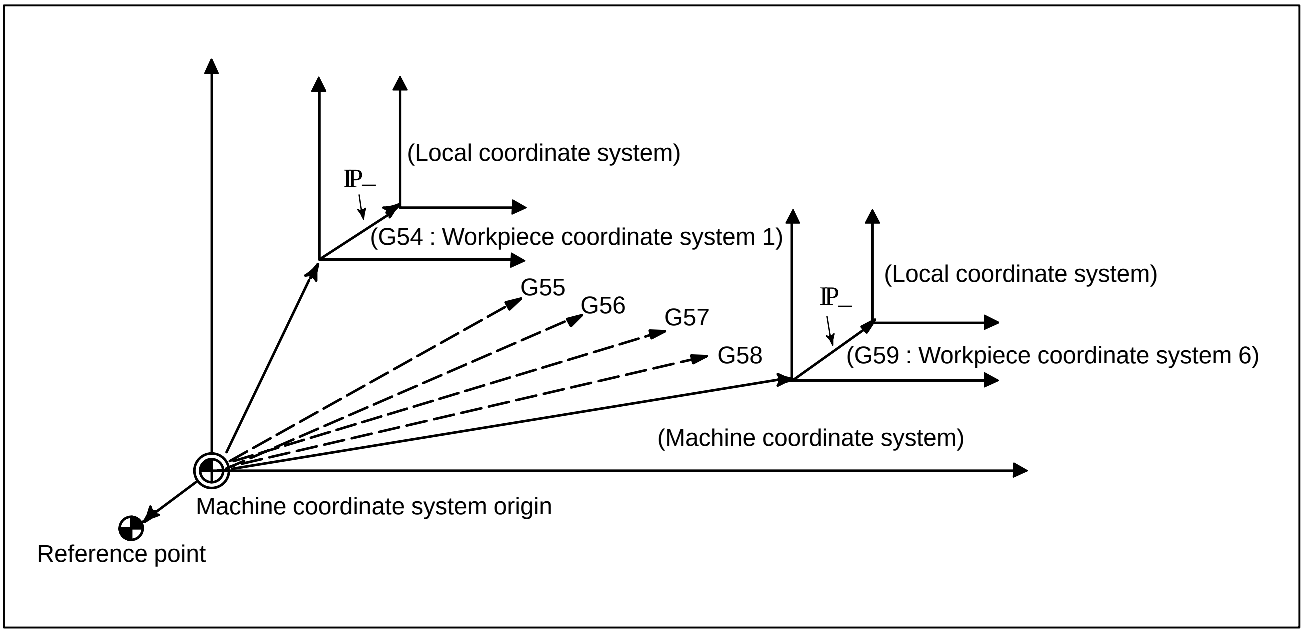


Fig. 7.3 Setting the local coordinate system

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 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.

4 G52 cancels the offset temporarily in cutter compensation.

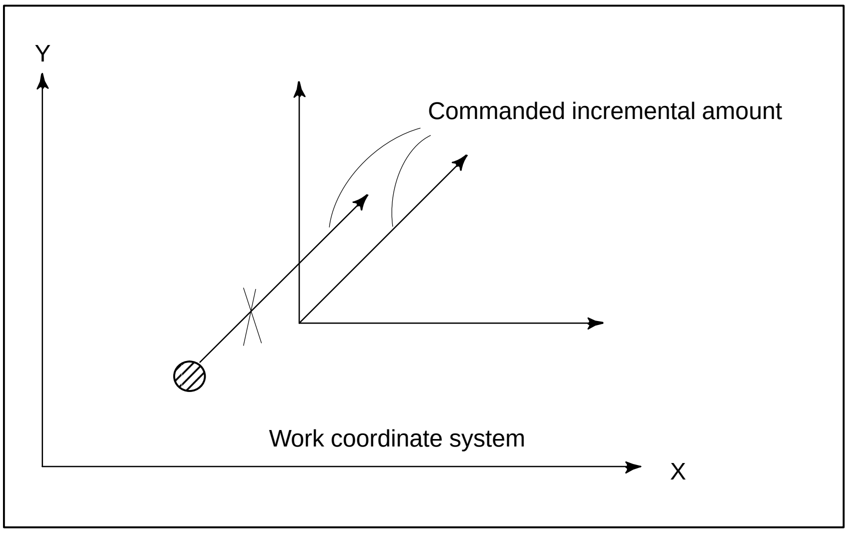
5 Command a move command immediately after the G52 block in the absolute mode.

6 If parameter CLR (No. 3402#6) is set in such a manner that NC is cleared by reset, the local coordinate system is cancelled by reset.

7 If X–axis or Y–axis was not commanded by G52 command, the local coordinate system of the un–commanded axis remains unchanged as before.
If X–axis or Y–axis was not commanded by G92 command, the local coordinate system of the uncommanded axis remains unchanged as before, and it is not cancelled.

8 The M, S or T code cannot be commanded in the same block as G52 command block.

9 The incremental command in a block just after G52 does not produce the incremental amount from the present position of the tool at that time, but it produces the incremental amount from the zero point of the set local coordinate system.
Provided that the incremental amount from the present position of the tool can be produced by parameter LIP (No. 16201#0) setting.



7.4 PLANE SELECTION

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

Explanations

Table 7.4 Plane selected by G code

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

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

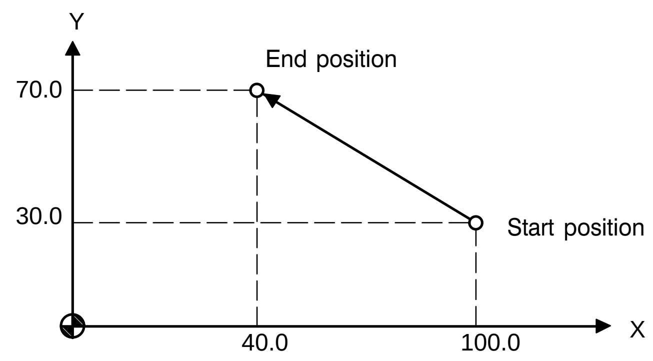
Format

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

Examples

G90 X40.0 Y70.0 ; **Absolute command**

G91 X–60.0 Y40.0 ; **Incremental command**



8.2 INCH/METRIC CONVERSION (G20,G21)

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

Format

G20 ;	Inch input
G21 ;	mm input

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

1 G20 and G21 must not be switched during a program.
2 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 reset.
3 Reference position return is performed at a low speed for the first G28 command after the inch input is switched to the metric input or vice versa.

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.

8.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Y, Z, 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. 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:

X1.2345; Truncated to X1.234 when the least input increment is 0.001 mm.
Processed as X1.2345 when the least input increment is 0.0001 inch.

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; Alarm 003 occurs because more than eight digits are specified.

X123456.7; If the least input increment is 0.001 mm, the value is converted to integer 123456700. Because the integer has more than eight digits, an alarm occurs.

9 PRESSING FUNCTION

9.1 PUNCH FUNCTION (1-CYCLE PRESSING)

This control sends a signal “Start press and punch” to the machine after moving a tool to the position commanded in a predetermined block. When the machine receives this signal, it starts pressing. As a result, punching is made on a workpiece by the selected tool. After punching, the press motion stops, and a signal returns to the NC to indicate that “punch has finished”.
Thus, NC proceeds to the execution of the next block. In this manner, punching on a workpiece by press motion is executed by data transfer between the NC and the machine, and it is necessary to know the blocks to be punched, in advance.
This description is made from the viewpoints of the NC side. Since details may differ depending upon the machine tool builders, refer to the machine tool builder’s manual without fail.

> **NOTE**
> This section covers one-cycle punch only. For nibbling (punching by sequential repeated press motion), refer to “9.3 NIBBLING FUNCTION”.

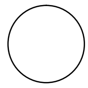
9.1.1 Block in which Punching is Made

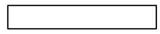
Punching is made in a block where the X-axis or Y-axis is positioned at rapid traverse, in principle.
In other words, punching is not done in a block where the X-axis or Y-axis is not positioned at rapid traverse. Blocks where punching is done are as follows:

(1) Block where X-axis or Y-axis is positioned in the positioning mode (G00)

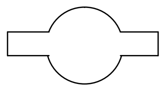
> **WARNING**
> If the same position as the present tool position is commanded by address X or Y, positioning is not done, but punching is executed. (This is regarded as the positioning command with movement amount 0)
> G00G91X0; . . . Punching is made.
> This applies to such a case that punching is done at the same position using a different tool.

Examples

Tool 01 profile 

Tool 02 profile 

N711G00G90X50.0Y30.0T02; . . . Punching is done using tool 02
N712X50.0Y30.0T01; . . . Punching is made using tool 01
The punch profile at (50, 30) position is as shown below.



No punching is made in case of N712T01;, N712T01C50.01;

WARNING
Punching is not done in T single block where the X-axis or Y-axis moves for tool offset.

(2) Block where pattern function G26, G76, G77, G78, G79, G86, G87 or G89 was commanded

Punching is made after positioning to respective points on a pattern.

Punching is not done in the following cases, even if the block corresponds to (1) or (2).

(a) MDI mode is selected.

(b) M code is commanded.

(c) Blocks inserted between M code of workpiece clamp and M code of workpiece unclamp which are employed for repositioning of workpiece.

(d) Block where positioning & punch off (G70) was commanded.

WARNING
Punching is not done even in G00 mode if the block is irrespective of positioning such as coordinate system setting (G92), local coordinate system setting (G52), standard point command (G72), dwell (G04), etc.

9.2 POSITIONING & PRESSING OFF (G70)

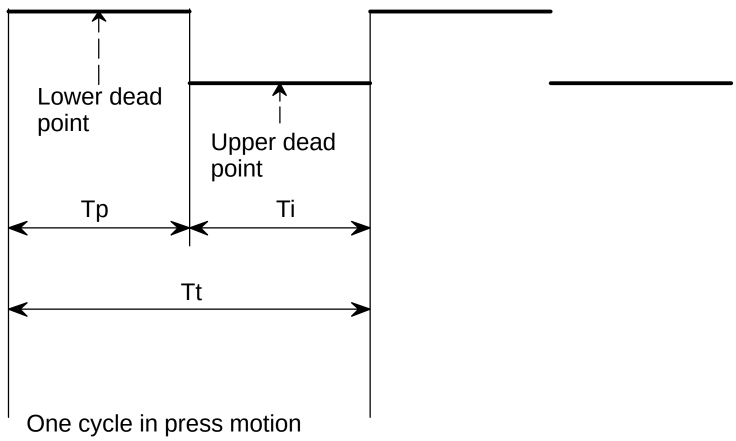
Punching is made in a block where the X-axis or Y-axis if positioned at rapid traverse, in principle.
Command the following code, if it is not desired to punch a workpiece after positioning a tool to the commanded position at rapid traverse.

G70X__Y__;

WARNING
1 G70 is an one–shot G code.
2 Rapid traverse is made in a G70 block even if in G01, G02 or G03 mode.

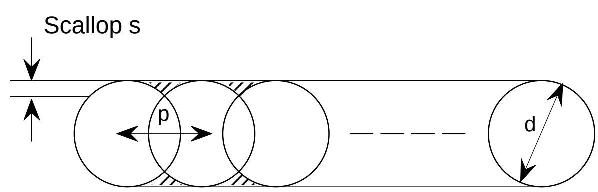
9.3 NIBBLING FUNCTION

Nibbling means sequential repeated punching without stopping press motion.
Assume Tt be the time required for one-cycle press motion. The remaining time obtained by subtracting punching time Tp from Tt (or, Ti = Tt – Tp) is the time allowable for positioning.



One cycle in press motion

The maximum distance (maximum pitch) which can be positioned in time Ti is limited by various conditions, such as machine, servo motor, and others as well as time Ti.
In this NC, the maximum nibbling pitch determined by these conditions is preset as a parameter.
On the other hand, the nibbling pitch is commanded by a program. If the commanded pitch exceeds the maximum pitch preset by the parameter, an alarm is produced.
Since this pitch can be specified directly, programming can be done, while taking the scallop into consideration.

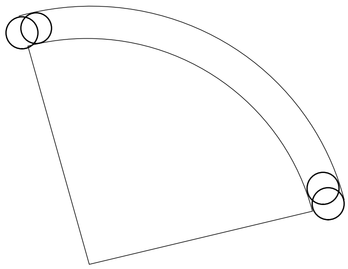
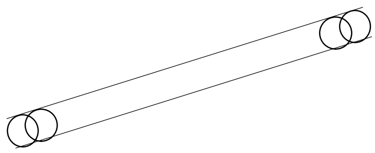


The relation between pitch p and scallop s is as shown below in case of linear nibbling

$$p = 2\sqrt{ds - s^2}$$

where d: Tool diameter

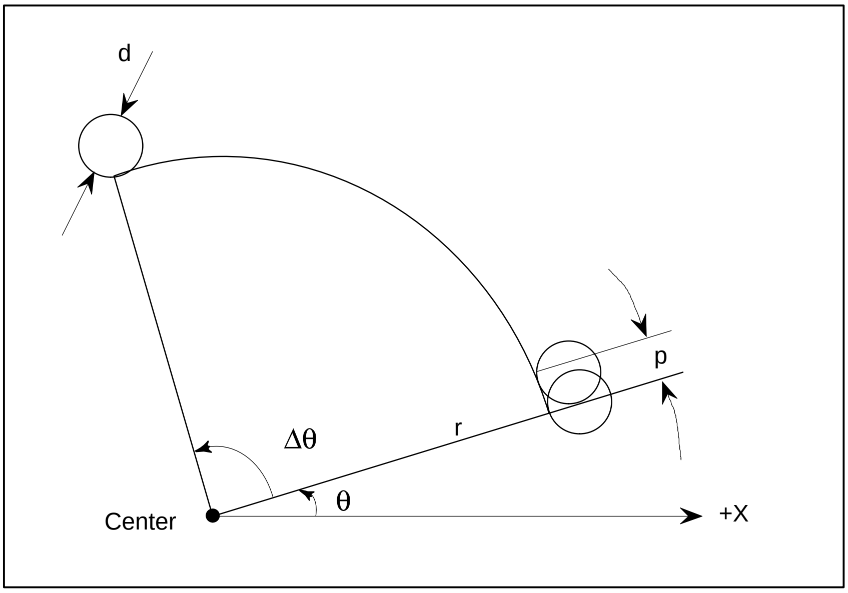
The following functions are prepared for nibbling.

Functions	Description
Circular nibbling (G68)	
Linear nibbling (G69)	
Nibbling by M function	M12; . . . . Nibbling is performed in these blocks. . . . . M13; (Note) Other M codes may be used instead of M12 and M13 depending upon machine tool builders.

9.3.1 Circular Nibbling (G68)

G68I r J θ K Δθ P d Q p ;

Nibbling is made at pitch p using a tool having diameter d, starting with the point which forms angle θ with reference to the X-axis on the circumference having radius r, with the preset tool position or the position specified by G72 being set as the circle center, to the point which forms angle θ + Δθ with reference to the X-axis.



r: Radius of arc
The unit is input unit by a positive number.

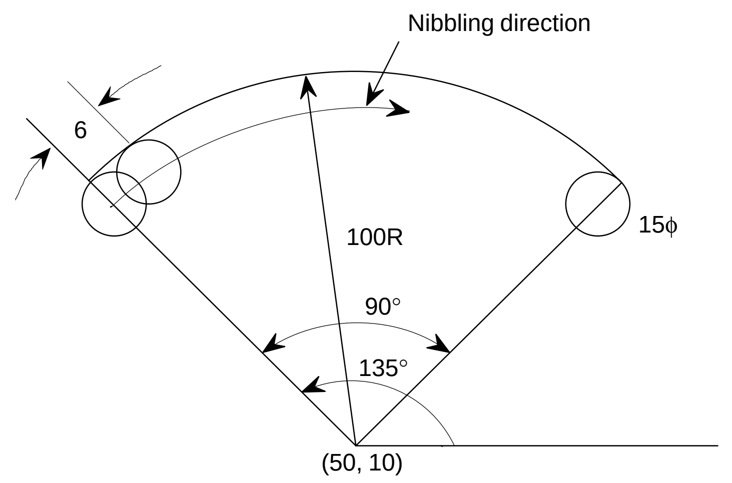
θ: Angle formed between the first punch point and the + X axis.
The unit is input (deg), and the counterclockwise direction is commanded by a positive number.

Δθ: Incremental angle from the first punch point to the least punch point
The unit is input unit (deg). Counterclockwise nibbling is made when this angle is commanded by a positive number.

d: Tool diameter value
The unit is input unit.
Nibbling is made outside the arc when this value is positive, inside the arc when this value is negative, and on the circumference when this value is 0.

p: Pitch
The unit is 0.01 mm in mm input and 0.01 inch in inch input. This pitch is specified as an arc length. For the commandable maximum value, since it has been set by parameters No. 16186 (for metric input), No. 16187 (for inch input), refer to the machine tool builder's manual.

Example 1

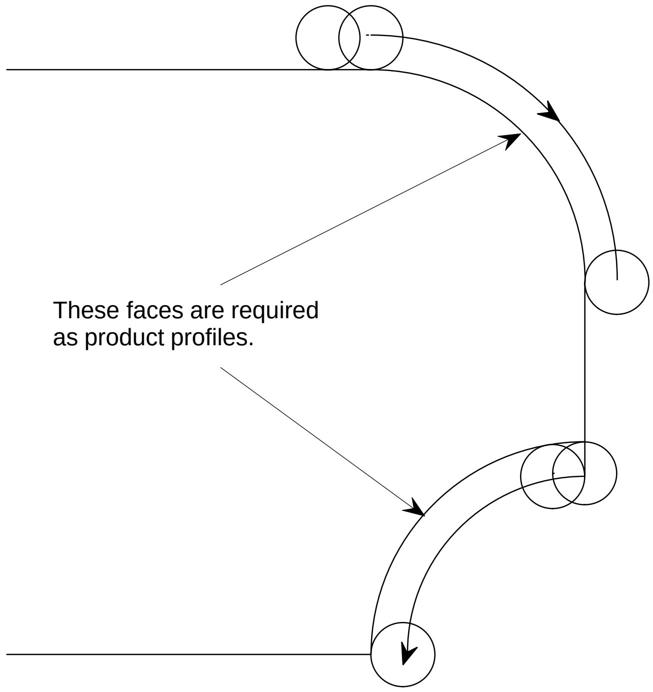



```
N711G72G90X50.0Y10.0;
N712G68I100.0J135.0K-90.0P-15.0Q6.0;
```

WARNING

1 G68 is an one-shot G code.
2 The standard point of G68 is the center of arc.
3 Pitch specification

The pitch is specified by the arc length.
The pitch is defined as the divided length of the arc having radius r specified in address I. The pitch does not mean the arc length between the centers of adjacent punch points, i.e., the divided length of the arc having a radius of r + d/2 (d: tool diameter).
This reason is shown below.



The commanded arc faces are required as a finel product, irrespective of whether the tool moves along outside of the arc (right upper part) or inside of the arc (right lower part).
Accordingly, it is desirable that the pitch along the specified arc is equal to the commanded pitch, when taking the scallop into consideration.

WARNING

4 Pitch compensation

When the circumferential length of the specified arc having radius r is divided by pitch p, a remainder may be produced in general. However, it is not desirable from the viewpoints of the machine and product profile to compensate this remainder by the movement to the last punch point, etc.

This NC automatically compensates these pitches to the same value as much as possible by the following method.

(i) Assume that arc = $2\pi r \frac{|\Delta\theta|}{360000}$ (Circumferential length of arc having radius r)

ARC = $2\pi r \frac{|\Delta\theta|}{360000}$ (Circumferential length of arc having radius r + $\frac{d}{2}$)

where,

π : Circular constant

r : Command value in address I Unit: input unit

$\Delta\theta$: Command value in address K Unit: input unit

$R = r + \frac{d}{2}$

d : Tool diameter Unit: input unit

(ii) Assume n be quotient and m be the remainder of $\frac{arc}{P}$

n = n when m = 0 (No remainder exists)

n = n + 1 when m $\neq$ 0 (A remainder exists)

(iii) Pt = $\frac{ARC}{n}$ … (Equation 1)

If Pt > maximum nibbling pitch (Pm) (This may be produced when nibbling is made outside an arc), perform the following calculation again.

Assume n' be the quotient and m' be the remainder of $\frac{ARC}{Pm}$

n = n' when m' = 0 (No remainder exists)

n = n'+ 1 when m' $\neq$ 0 (A remainder exists)

Then, calculate equation 1.

Pt is actual tool pitch.

Provided that remainder ℓ = 0, 1, . . . or n – 1 is produced in equation 1. (input unit)

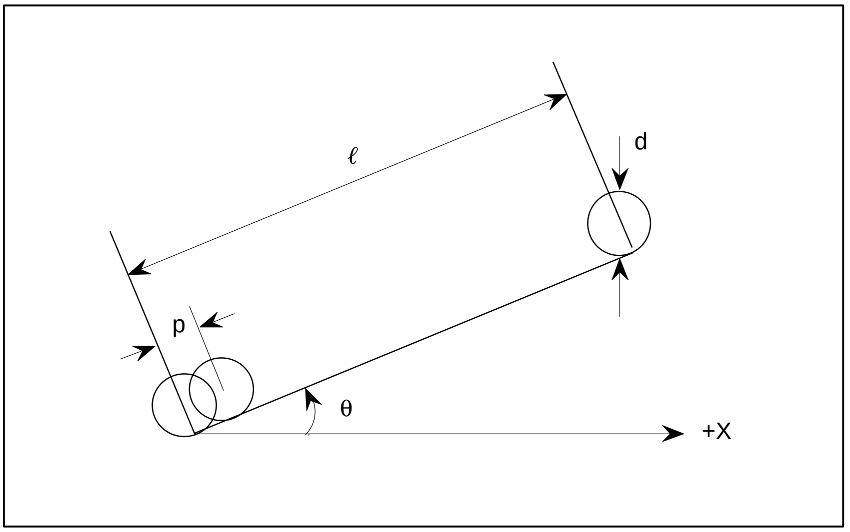
This remainder is compensated by setting the first ℓ-times pitch to Pt = Pt + 1 in the movement of n times.

5 If radius is 0, or if the pitch is 0 or more than a specified value, alarm (No. 4523) is produced.

9.3.2 Linear Nibbling (G69)

G69I $\underline{\quad \ell \quad}$ J $\underline{\quad \theta \quad}$ P $\underline{\quad d \quad}$ Q $\underline{\quad p \quad}$;

By the above command, nibbling is made at pitch p using a tool having diameter d along a straight line of length ℓ which forms angle θ with reference to the X-axis, starting with the present tool position or the position specified by G72 as the start point.



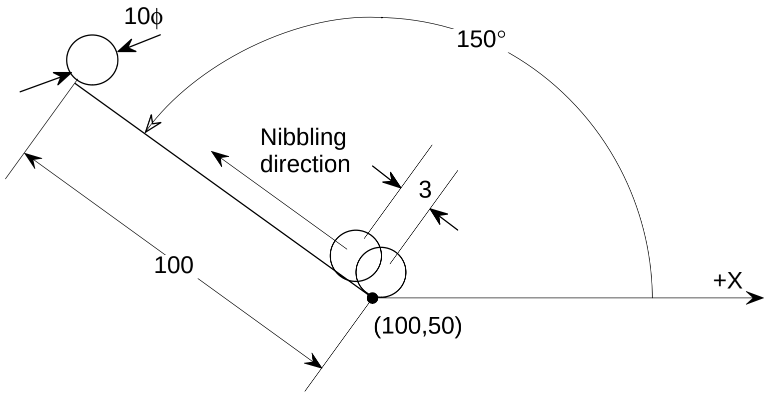
ℓ: Length of straight line
The unit is input unit.
If a negative number is commanded, the angle between the straight line and the +X axis becomes θ + 180 deg.

θ: Angle formed between straight line and the + X axis
The unit is input unit (deg), and the counterclockwise direction is commanded by a positive number.

d: Tool diameter value
The unit is input unit. Nibbling is made leftwards by d/2 when this value is positive or rightwards by d/2 when this value is negative as viewed from the start point to the end point of the straight line.
Nibbling is made along the straight line when this value is 0.

p: Pitch
The unit is input unit. This is commanded by the length in the straight line direction.
For the maximum commandable value, since it has been set by parameters (No. 16186: for metric input, No. 16187: for inch input), refer to the machine tool builder's manual.

Example 2



N721G72G90X100.0Y50.0;
N722G69I100.0J150.0P-10.0Q3.0;
The N722 block may also be commanded as
G69I-100.0J-30.0P-10.0Q3.0;

WARNING

1 G69 is a one-shot G code.
2 The standard point of G69 is the start point.
3 The pitch compensation is the same as in circular nibbling (G68); except that the circumferential length of an arc in G68 is changed to the linear length (For details, refer to Warning 4 in 9.3.1)
4 If pitch is 0 or more than specified, alarm (No. 4524) is produced.

9.3.3 Notes on Circular Nibbling (G68) and Linear Nibbling (G69)

WARNING

1 The maximum pitches in G68 and G69 are set by parameters No. 16186 (for mm input) and No. 16187 (for inch input).
2 If T code is commanded in G68 or G69 block, nibbling is started after the X and Y axes have moved to the first punch point and also a tool has been selected.
3 M code is not commandable in G68 and G69 blocks.
4 For the rapid traverse to the first punch point, the rapid traverse override is effective when it is specified by the rapid traverse override switch on the machine operator's panel or by F1-digit specification. For the pitch movement up to the final point, the rapid traverse override is ineffective and fixed to 100%.
5 If G68 or G69 is commanded using the single block operation, nibbling is made up to the last punch point, and then, stopped.
6 If feed hold is applied halfway during the movement to the first punch point, the X and Y axes stop at once.
These axes also stop immediately when the feed hold is applied halfway during the pitch movement from the first point to the last point.
However, this can be changed by parameter NSP (No. 16181#2) in such a way that the X and Y axes stop after pitch movement.
7 In a block just after G68 or G69, the tool does not move by the incremental amount from the tool position when nibbling ends, but moves from the programmed end point of the arc or straight line by the incremental amount.

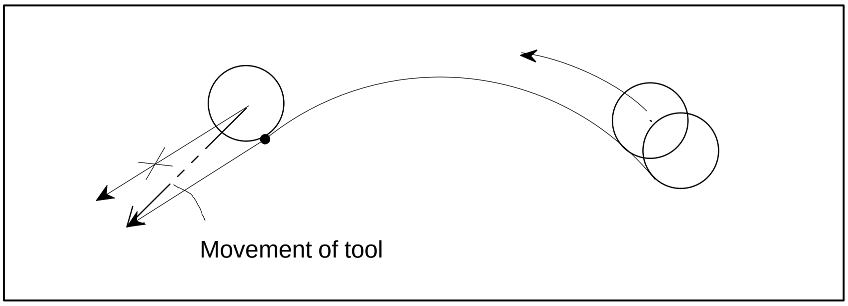


Fig. 9.3.3 (a) Incremental command just after circular nibbling (G68)

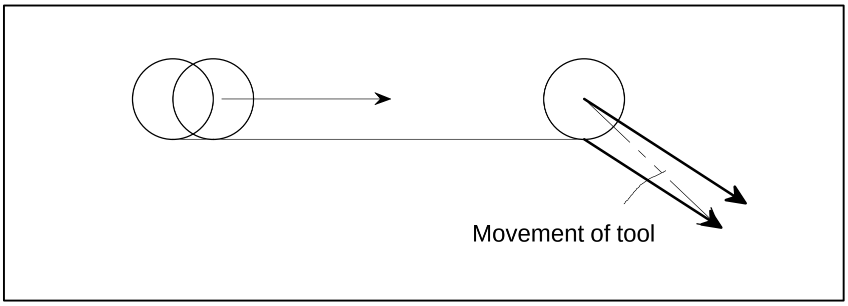
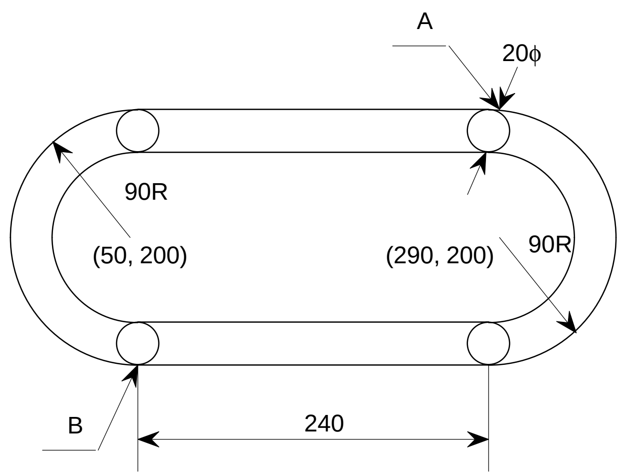


Fig. 9.3.3 (b) Incremental command just after linear nibbling (G69)

Example 3




```
N731G72G90X290.0Y200. ;
N732G68I90. J–90. K180. P–20. Q5. ;
N733G69I240. J180. P20. Q5. ;
N734G72X50. Y200. ;
N735G68I90. J90. K180. P–20. Q5. ;
N736G69I240. J0P20. Q5. ;
```

It is not necessary to command point A by G72 in the block next to N732 nor point B by G72 in the block next to N735.

WARNING

8 Since radius, straight line length in G68 and G69 are not modal data, they must be commanded correctly in every block.

9 It is possible in certain machines that nibbling is not performed by G68 or G69 command, but one-cycle punching is executed at respective positioning points.

9.4 NIBBLING BY M FUNCTION

In addition to the circular or linear nibbling according to the G68 or G69 command, this control can perform nibbling by M function. In other words, it can execute nibbling in the blocks from a block with the M code of nibbling mode to a block with the M code of nibbling mode cancel as shown below.

M12; (M code of nibbling mode)

```
    . . . . . . . . . . . .
    . . . . . . . . . . . .
    . . . . . . . . . . . .   Nibbling is done in these blocks.
    . . . . . . . . . . . .
    . . . . . . . . . . . .
    . . . . . . . . . . . .
```

M13; (M code of nibbling mode cancel)

In this manual, the M code of nibbling mode is described as M12, while the M code of the nibbling mode cancel is described as M13. However, since these M codes may be different from those specified above in certain machine tool builders, you are requested to read these M codes correspondingly according to the manual prepared by these machine tool builders.

Don’t use this nibbling by M function in a different way other than specified in this manual, since there are certain restriction about its use.

WARNING

Each of the M codes for nibbling mode and nibbling mode cancel must be commanded in a single block.

9.4.1 G00 Command in Nibbling Mode

Example4

N100G00G90X$\underline{\ x_1\ }$ Y$\underline{\ y_1\ }$;
N110M12;
N120X$\underline{\ x_2\ }$ Y$\underline{\ y_2\ }$ T$\underline{\quad}$;
N130X$\underline{\ x_3\ }$ Y$\underline{\ y_3\ }$;
N140X$\underline{\ x_4\ }$ Y$\underline{\ y_4\ }$;
N150X$\underline{\ x_5\ }$ Y$\underline{\ y_5\ }$;
N160X$\underline{\ x_6\ }$ Y$\underline{\ y_6\ }$;
N170X$\underline{\ x_7\ }$ Y$\underline{\ y_7\ }$;
N180M13;

(1) The first punch point of nibbling is commanded in the block next to M12. A T code can also be commanded in this block concurrently.

The first punch point (x_2, y_2) of nibbling is previously commanded in N120 block, and nibbling is started when the positioning to (x_2, y_2) has been completed and the motion corresponding to the T code has been finished.

The movement amounts along the X-axis and Y-axis in this block are not restricted by the maximum positioning distances (parameter Nos. 16188, 16184) of the X-axis and Y-axis in the nibbling mode.

(2) In the blocks after the first punch point of nibbling has been commanded, the positioning command is given to correspond to the pitches by G68 and G69 commands.
If the movement amounts along the X-axis and Y-axis in these blocks exceed the parameter set values (parameter Nos. 16188, 16189), alarm (No. 4521) results.

In N130 to N170 blocks, the positioning corresponding to the nibbling pitch is commanded. If the movement amounts along X-axis x_3-x_2, x_4-x_3, x_5-x_4, x_6-x_5, x_7-x_6 or the movement amounts of Y-axis y_3-y_2, y_4-y_3, y_5-y_4, y_6-y_5, y_7-y_6 exceed the parameter set values in each block, alarm (No. 4521) is issued.

(3) M13 is commanded in the block next to the block where the last punch point of nibbling was commanded.

The last punch point (x_7, y_7) of nibbling is commanded in the N170 block and M13 is commanded in the next block, i.e. in block N180. Punching is done at (x_n, y_n) and nibbling terminates.

The G00 command in the nibbling mode is given to execute the positioning to the commanded position, and the movement amounts along X-axis and Y-axis should not exceed the parameter set values (No. 16188: for millimeter input, No. 16189: for inch input, except for the command to the first punch point of nibbling.

9.4.2 G01, G02, and G03 Commands in Nibbling Mode

Linear nibbling can be done by commanding G01 in the nibbling mode, while circular nibbling can be done by commanding G02 and G03 in the nibbling mode.
The tool diameter cannot be offset by G01, G02, G03 commands. Accordingly, these commands are used together with cutter compensation commands (G40, G41, G42) when nibbling is done by offsetting a continuous straight line or circular arc leftwards or rightwards by the tool diameter.

(i) Straight line

G01X__Y__Q__;

The end point is designated by address X, Y, while the pitch is designated by address Q.

(ii) Circular arc

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

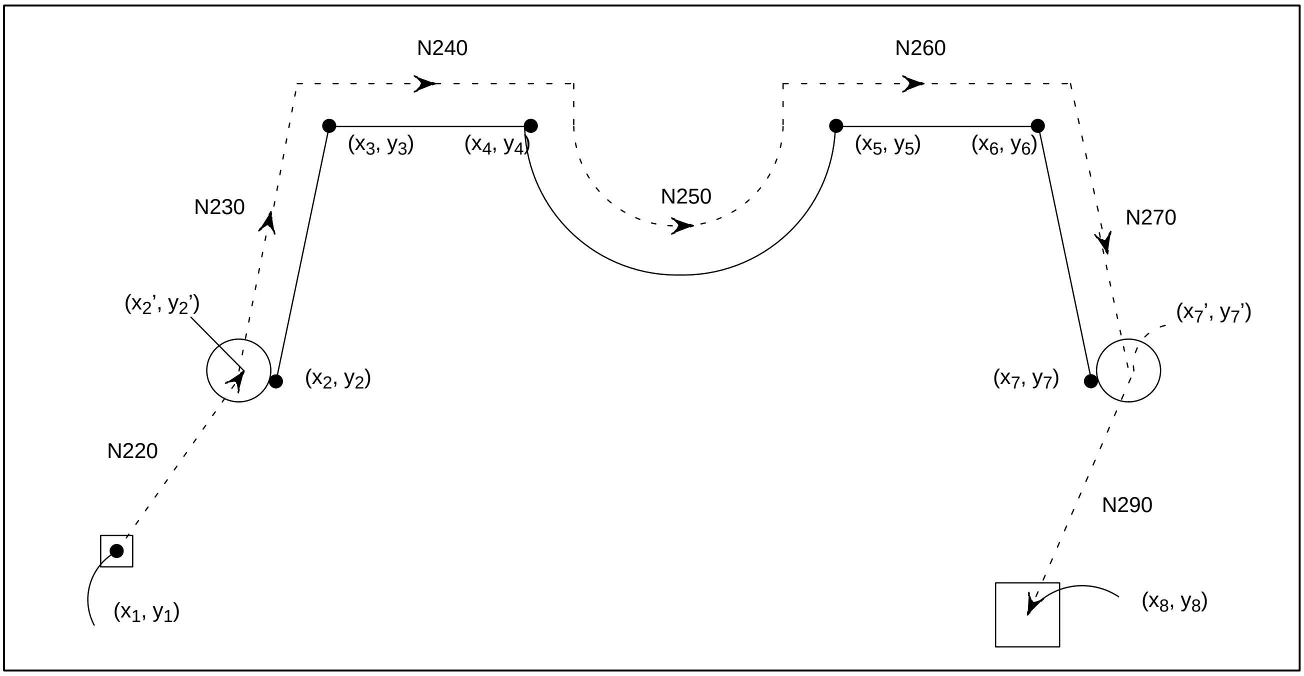
The end point is designated by address X, Y, the radius of circular arc is designated by address I, J, or R (For details, refer to II-4.3 “Circular interpolation”) and also the pitch is designated by address Q.

Q: Pitch

The pitch is commanded by the unit of input.

Example 5

N200G00G90X x_1 Y y_1 ;
N210M12;
N220G41X x_2 Y y_2 T02D02;
N230G01X x_3 Y y_3 Q__;
N240X x_4 Y y_4 ;
N250G03X x_5 Y y_5 I__J__;
N260G01X x_6 Y y_6 ;
N270X x_7 Y y_7 ;
N280M13;
N290G40G00X x_8 Y y_8 T03;



The G40, G41, and G42 codes function as follows.
For details, refer to 15.1 Cutter compensation.

G code	Function
G40	Cutter compensation cancel
G41	Leftward offset for moving direction of tool
G42	Rightward offset for moving direction of tool

Assume that the diameter (radius) of the tool selected by T02 is preset to offset No. 02.

(1) The first punch point of nibbling and either G41 or G42 to offset the tool leftwards or rightwards as viewed from the moving direction of the tool are commanded in the block next to M12 block. The first punch point is commanded by G00.

The first punch point (x_2, y_2) of nibbling is commanded and also G41 is commanded in the N220 block.
Accordingly, positioning is done at rapid traverse to the point (x_2', y_2') being offset leftwards to the commanded position by the tool diameter being preset to offset No. 02.
Nibbling is started when the positioning has been completed and the motion corresponding to T code has been finished.

(2) The straight line is commanded by G01, the circular arc is commanded by G02, G03, and the pitch is commanded by Q, starting with the block next to the block where the first punch point was commanded.
Since pitch Q is a modal data, it is no longer needed to designate pitch Q after pitch Q was designated once.
The axis movement to pitch Q is done at rapid traverse.

The straight line and circular arc along which nibbling is done are commanded in N230 to N270 blocks. The straight line and circular arc obtained by offsetting the commanded straight line and circular arc leftwards by the tool diameter being preset to offset No. 02, are divided by pitch Q.
The axis movement to pitch is done at rapid traverse.

(3) M13 is commanded in the block next to the block where the end point of nibbling was commanded.

The end point (x_7, y_7) of nibbling is commanded in N270 block, and M13 is commanded in the N280 block next to the N270 block. Nibbling is completed after punching was done at point (x_7', y_7') being offset leftwards by the tool diameter at end point (x_7, y_7).

For G01, G02, and G03 in the nibbling mode, the straight line and circular arc are divided by the pitch commanded by address Q, and the movement to respective pitches is done at rapid traverse. If the pitch commanded by address Q exceeds the parameter set value No. 16186: for mm input, No. 16187: for inch input, alarm (No. 4523) is issued.
Now, the difference of the motion will be described when the program shown in example 5 is presumed to have been programmed as shown in example 6.

Example 6

N200G00G90X $\underline{\ x_1\ }$ Y $\underline{\ y_1\ }$;
N210G41X $\underline{\ x_2\ }$ Y $\underline{\ y_2\ }$ T02D02;
N220M12;
N230G01X $\underline{\ x_3\ }$ Y $\underline{\ y_3\ }$ Q__;
. . . .
. . . .
. . . .

In the program shown in example 5, nibbling is started after the positioning has been completed to (x_2', y_2') in Fig. 9.4.2. In case of the program shown in example 6, on the other hand, one-cycle punching only is done after the positioning has been completed to point (x_2', y_2'), and nibbling is started after the axis movement by pitch Q along the straight line in N230 block.

9.4.3 Notes on Nibbling by M Function

WARNING

1 The following commands only are executable in nibbling mode.
   (i) X, Y positioning command by G00
   Provided that the T code and F1-digit command can be included in the same block where the X, Y positioning is made by G00 to the first punch point of nibbling.
   (ii) G26 (bolt hole circle), G76 (line at angle), G77 (arc), G78, G79 (grid), G86 (share proofs), G87 (square), G88 (radius), G89 (cut at angle)
   The movement amounts along the X-axis and Y-axis to respective positioning points should not exceed the parameter set value (Nos. 16188, 16189), except when the first positioning point is equivalent to the first punch point of nibbling.
   (iii) G01, G02, G03, G41 and G42 commands in the range described in 9.4.2.

2 The positioning distance commandable by the X-Y positioning command by G00 is not composite distance $\sqrt{\Delta x^2 + \Delta y^2}$ obtained by the movement amounts along X-axis and Y-axis.
   If the absolute value of the movement amount along either X-axis or Y-axis exceeds the parameter set value (Nos. 16188, 16189), alarm (No. 4521) is issued.
   This provision also applies to G26, G76, G77, G78, G79, G86, G87, G88 and G89, correspondingly.

3 When offset was made by the tool diameter by G41 or G42 to G01, G02 and G03 as described in example 5 the offset straight line or circular arc is divided by the pitch commanded by address Q.
   Be careful since the above division differs form such a case that a commanded circular arc is divided by a commanded pitch, like in G68 (circular nibbling).

9.5 EXTERNAL MOTION FUNCTION

Section 9.1 “PUNCH FUNCTIONS (1-CYCLE PRESSING)” explained the blocks, in which punching is made after positioning. In certain cases, no punching is made, but tapping and other mechanical motion may be executed in these blocks.

Example

```
M80;
G00X__Y__T__;
X__Y__;
. . . . . . . . . . . . . .
. . . . . . . . . . . . . .
X__Y__;
M81;
```

Tapping is made instead of punching.

Since this function depends upon the machine tool builders, refer to the machine tool builder’s manual.

10 S FUNCTION

10.1 SPECIFYING THE S CODE WITH A BINARY CODE

S code can be specified by address S followed by a binary code. A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the number of digits in an S code or the execution order when a move command and an S code command are in the same block.

11 TOOL FUNCTION (T FUNCTION)

11.1 TOOL SELECTION FUNCTION

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

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

(ii) Executing T function commands upon completion of move command execution.

The selection of either (i) or (ii) depends on the machine tool builder's specifications. Refer to the manual issued by the machine tool builder for details.

<u>The T command must be given without fail to the block before a block where punching is first made by press motion in one program or to the same block where punching is first made by press motion.</u>
If the T command is not given to these blocks, the press start signal which instructs "Punch by press motion", is not sent to the machine, and machining does not proceed to the next block.

Examples

```
0100G92  . . . . .  ;
N1G00G90X__Y__;
 . . . . . . . . .
 . . . . . . . . .
```

Punching should be made in N1 block.
However, since no T command is given to a block before N1 block or N1 block, no punching is made, and machining does not proceed to the next block.

WARNING

1 The correspondence between commandable T codes and tools depends upon machine tool builders.
The commandable T codes are set in tool registering screen before shipment from factory. If a commanded T code was not registered, alarm (No. 4602) is produced.

2 No T code is commandable in the following blocks.
(i) G10 (Offset value setting)
(ii) G22 (Stored stroke limit function on)
(iii) G23 (Stored stroke limit function off)
(iv) G92 (Coordinate system setting)
(v) G52 (Local coordinate system setting)
(vi) G72 (Standard point command)
(vii) G75 (Automatic repositioning)
(viii) G98 (Base point command for multi-piece machining)
(ix) G73, G74 (Multi-piece machining command)

3 If tape or memory operation is made in the T command neglect status, the T command is ignored, and the operation is made as if the T command were not given.
The press start signal is not sent to the machine side in a block to be punched, and processing does not proceed to the next block. If a program is checked by marking to a workpiece by using a marking tool, for example, select the marking punch tool by a T command in the MDI mode in advance, and perform the tape or memory operation without reset operation in the T command neglect status. Now, punching is made using the tool selected in the MDI mode.

4 If automatic operation is applied to the machine side by the cycle start when the cycle start lamp signal is not sent, i.e., in the reset status, the cycle start lamp signal is sent.
If a block to be punched appears before a T command is given after this cycle start lamp signal has been sent, the press start signal is not sent to the machine side, and also machining does not proceed to the next block.
The cycle start lamp signal is stopped by reset operation.

5 The press start signal can be sent by setting a parameter TCF (No. 16003#5) even if no T command is given to a block before the block where the punching is made or the block in which punching is made. In this case, the integrated value of the number of punch times may differ from actual number of punch times about respective tools.

11.2 T COMMAND NEGLECT

This function ignores the T command. Whether the T command is ignored or not is generally selected by a switch on the machine operator's panel.

If the T command is ignored, it is treated, as if no T code command were present on a program. Accordingly, it is not checked that whether the T code be commandable or not.

WARNING
The ignorance of T command is judged when the command is read from a tape reader or memory into buffer register. Accordingly, the selection of the T command ignorance switch is not effective for the block that has been read into the buffer register.

NOTE
By setting a parameter TNM (No. 16260#5), it is checked that whether the T code be commandable or not, only in the machine lock status.

11.3 TOOL OFFSET

Tool offset is applicable to respective T codes in the X-axis and Y-axis directions.

Since use of this tool offset function depends upon machine tool builders, refer to the machine tool builder's manual.

WARNING

1 Tool offset compensation applies to tools numbered from 1 to 9999.
2 Tool offset values are set in the tool registering screen in the unit of the least command increment for each X axis and Y axis before shipment of the machine from the machine tool builder's factory.
3 If data are inputted by inch in a millimeter system machine, or if data are inputted by millimeter in an inch system machine, a tool offset error is produced within the sum of a half of the least input increment and half of the least command increment. This error is not accumulated.

11.4 CONTROLLING THE TURRET-AXIS (T-AXIS)

The CNC uses set parameters to control the turret which is indexed for a tool to be used. A specified T code is output, and at the same time, the turret is positioned at the location which was specified for the tool on the tool registration screen. Up to 136 tools can be registered on the screen. A T code consisting of one to four digits is used for controlling the turret. This control substantially reduces the time required to change tools because changing the tool, which was conventionally performed in the machine tool, is done by positioning the turret using the CNC.
For detailed information, refer to the manual prepared by the machine tool builder.

NOTE
The T-axis can be controlled when TCL, bit 4 of parameter No. 16260, is 1.

11.5 TOOL LIFE MANAGEMENT FUNCTION

Tools are classified into various groups, with the tool life (frequency of use) for each group being specified.

11.5.1 Tool Life Management Data

Tool life management data consists of tool numbers, and tool life value.

- **Tool number**

Specify a four–digit number after T.

11.5.2 Register and Change of Tool Life Management Data

In a program, tool life management data can be registered in the CNC unit, and registered tool life management data can be changed.

Format

- **Addition and change of tool life management data**

Format	Meaning of command
G10L30 ; **N_P7R_ ;** **N_P7R_ ;**	**G10L30 ;** Start of change **N_** : Tool number **P7** : Tool life data selection
N_P7R_ ;	**R_** : Life value
G11 ; **M02 (M30) ;**	**G11** : End of change

Explanations

- **Types**

For tool life management, the type used varies from one machine to another. For details, refer to the appropriate manual of each machinde tool builder.

11.5.3 Tool Life

The life of a tool is specified by a usage frequency (count).

Explanations

- **Usage count**

The usage count is incremented by 1 for each tool used punching.

12 AUXILIARY FUNCTION

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

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

ii) Executing miscellaneous function commands upon completion of move command execution.

The selection of either sequence depends on the machine tool builder's specification. Refer to the manual issued by the machine tool builder for details.

12.1 AUXILIARY FUNCTION (M FUNCTION)

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

Explanations

The following M codes have special meanings.

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

This indicates the end of the main program
Automatic operation is stopped and the CNC unit is reset.
This differs with the machine tool builder.
After a block specifying the end of the program is executed, control returns to the start of the program.
Bit 5 of parameter 3404 (M02) or bit 4 of parameter 3404 (M30) can be used to disable M02 or M30 from returning control to the start of the program.

- **M00 (Program stop)**

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

- **M01 (Optional stop)**

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

- **M98 (Calling of sub–program)**

This code is used to call a subprogram. The code and strobe signals are not sent. See the subprogram section 13.3 for details .

- **M99 (End of subprogram)**

This code indicates the end of a subprogram.
M99 execution returns control to the main program. See the subprogram section 13.3 for details.

- **M08, M09 (Forming mode and forming mode cancel)**

If punching (excluding nibbling) is executed in a block between M08; and M09; it is executed when the preset time by parameter (No. 16032) has passed after completion of positioning, and machining proceeds to the next block when the preset time by parameter (No. 16033) has passed after receiving the punch finish signal from the machine side. (Other M codes may be used for these functions depending upon the machine tool builders.)

- **M10, M11 (Workpiece clamp and workpiece unclamp)**

The movement amounts of the X and Y axes are not added to the absolute coordinate value in a block between by M10; and M11;. No punching is executed, even if punching is specified in the block.
Used to reposition a workpiece. (Other M codes may be used for these functions, depending upon machine tool builders.)

- **M12, M13 (Nibbling mode and nibbling mode cancel)**

Nibbling is executable in a block between M12; and M13;. (Other M codes may be used for these functions depending upon machine tool builders)

WARNING

1 M08, M09, M10, M11, M12 and M13 must be commanded in a single block.

2 No punching is made in a block where an M code is commanded. But punching may be made by some machine tool builder.

3 No M code is commandable in the following blocks.

(i) G10 (Offset amount setting)
(ii) G22 (Stored stroke limit function ON)
(iii) G23 (Stored stroke limit function OFF)
(iv) G26 (Bolt hole circle)
(v) G76 (Line at angle)
(vi) G77 (Arc)
(vii) G78, G79 (Grid)
(viii) G68 (Circular nibbling)
(ix) G69 (Linear nibbling)
(x) G72 (Base point command)
(xi) G75 (Automatic repositioning)
(xii) G86 (Share proofs)
(xiii) G87 (Square)
(xiv) G88 (Radius)
(xv) G89 (Cut at angle)
(xvi) G98 (Base point command of Multi-piece machining)
(xvii) G73, G74 (Multi-piece machining command)
(xviii) G52 (Local coordinate system setting)

NOTE

1 If there is a block following M00, M01, M02, or M30, it is not read into the buffer storage. Similarly, ten M codes which cause the block following them not to enter the buffer storage are available by parameter setting (No. Nos. 3411 - 3421). As for these M codes, refer to the machine tool builder's manual.

2 For M98 code and M99 code, their code signals and strobe signals are not transmitted.

12.2 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 example, M42 can be specified only after the mechanical operation of M41 is completed. 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 ; G00G91X0Y0 ; : : :	M40M50M60 ; G00G91X0Y0 ; : : : : :

12.3 THE SECOND AUXILIARY FUNCTIONS (B CODES)

Indexing of the table is performed by address B and a following 8–digit number. The relationship between B codes and the corresponding indexing differs between machine tool builders.
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

Restrictions

1. Addresses used for the second auxiliary functions (addresses specified with B or parameter No. 3460) cannot be used as controlled axis names (parameter No. 1020).
2. If this function is used, the A/B macro function is disabled.
3. When this function is used, the parameter ABM (No. 16200#6) must be set to 1.

13 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.

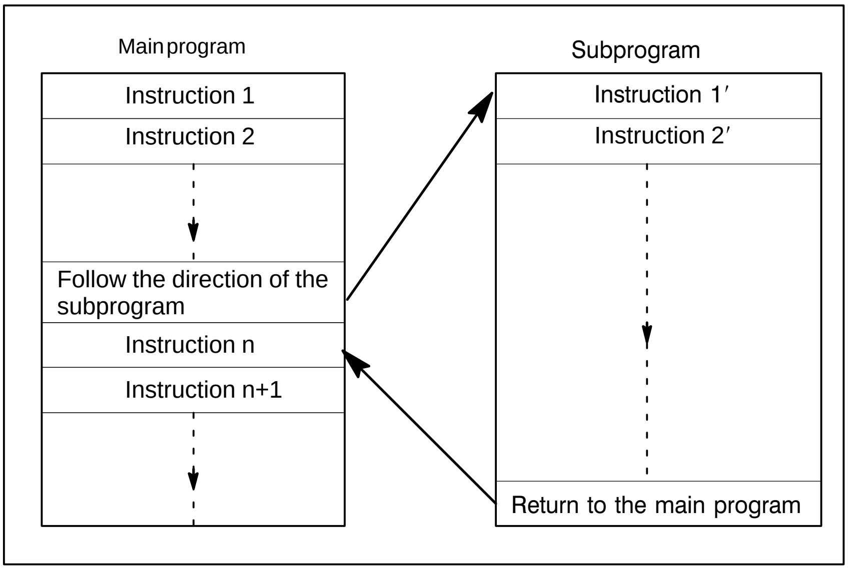


Fig. 13 (a) Main program and Subprogram

The CNC memory can hold up to 400 main programs and subprograms. A main program can be selected from the stored main programs to operate the machine. See Chapter III–10 for the methods of registering and selecting programs.

- **Program components**

A program consists of the following components:

Table 13 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

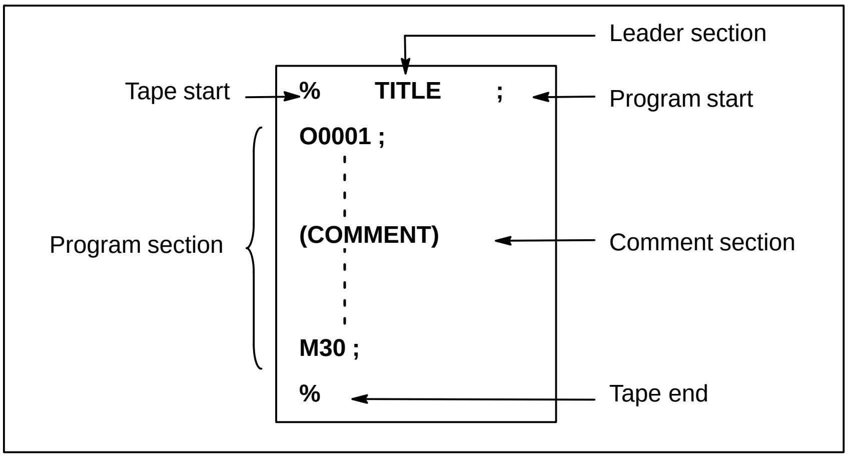


Fig. 13 (b) Program configuration

- **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 G00 X100.T02 ;
:	:
Block n	Nn M2 ;
Program end	M30 ;

A block contains information necessary for machining, such as a move command or coolant on/off command.Specifying a value following a slash (/) at the start of a block disables the execution of some blocks (see “optional block skip” in Section 13.2).

13.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See Section 13.2 for a program section.

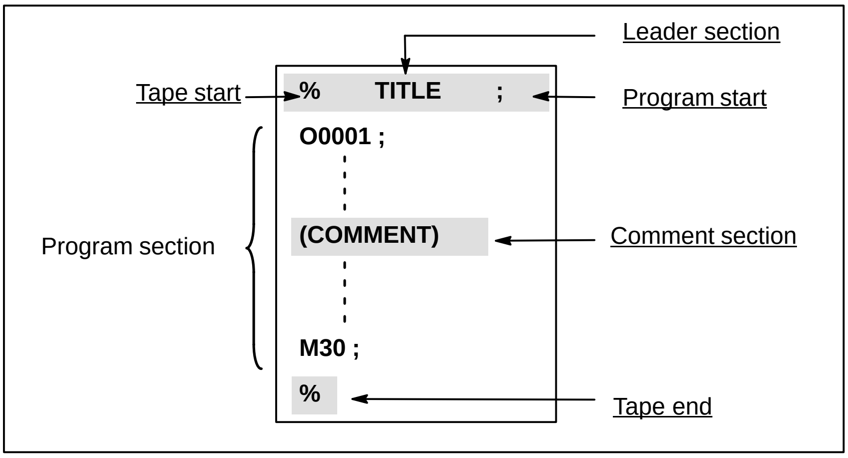


Fig. 13.1 Program configuration

Explanations

• Tape start

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

Table 13.1 (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 ordinary personal computers, this code can be entered by pressing the return key.

Table 13.1 (b) Code of a program start

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

WARNING
If one file contains multiple programs, the EOB code for label skip operation must not appear before a second or subsequent program number. However, an program start is required at the start of a program if the preceding program ends with %.

- **Comment section**

Any information enclosed by the control–out and control–in codes is regarded as a comment and skipped by the CNC.
The user can enter a header, comments, directions to the operator, etc. in a comment section using the EOB code or any other code.
There is no limit on the length of a comment section.

Table 13.1 (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 command tape 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 F are ignored, and thus are not read into memory.
When data in memory is punched out on paper tape with the punch function, the comment sections are also punched out.
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 punched out or displayed on the screen.
During memory operation in memory command mode, 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).

WARNING
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 CNC 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 alarm (No. 5010) is occurred.

Table 13.1 (d) Code of a tape end

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

13.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See Section 13.1 for program components other than program sections.

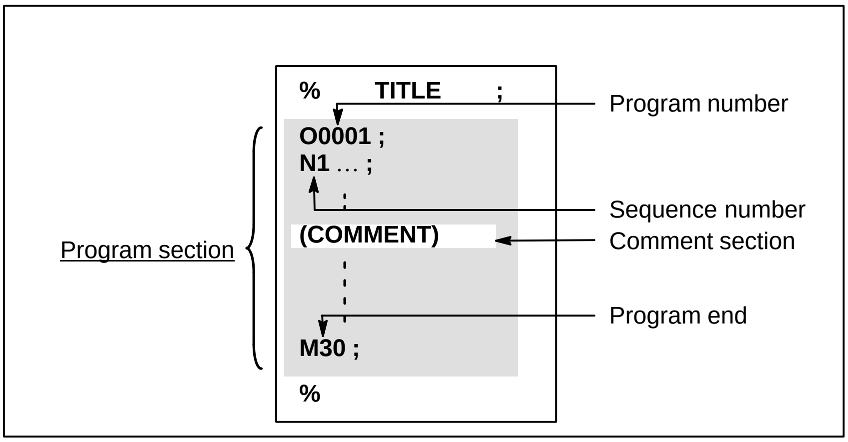


Fig. 13.2 (a) Program configuration

- **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 Section 9.3 in Part III.).

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 13.2 (a) EOB code

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

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

```
N300 X200.0 Y300.0 ;  A sequence number is underlined.
```

Fig. 13.2 (b) Sequence number and block (example)

WARNING

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 horizontally. If the number of characters in one block (starting with the code immediately after an EOB and ending with the next EOB) is odd, an alarm (No.002) is output. No TV check is made only for those parts that are skipped by the label skip function. A comment section enclosed in parentheses is also subject to TV check to count the number of characters. The TV check function can be enabled or disabled by setting on the MDI unit (See Subsec. 11.4.3 in Part III.).

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

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus sign (–) may be prefixed to a number.)
Word = Address + number (Example : X–1000)
For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address. Table 13.2 (b) indicates the usable addresses and their meanings.
The same address may have different meanings, depending on the preparatory function specification.

Table 13.2 (b) Major functions and addresses

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

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

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

Fig. 13.2 (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 13.2 (c) Major addresses and ranges of command values

Function		Address	Input in mm	Input in inch
Program number		O (Note)	1 to 9999	1 to 9999
Sequence number		N	1 to 99999	1 to 99999
Preparatory function		G	0 to 99	0 to 99
Dimen-sion word	Increment system IS–B	X, Y, Z, U, V, W, A, B, C, I, J, K, R,	±99999.999mm	±9999.9999inch
	Increment system IS–A		±999999.99mm	±99999.999inch
Feed per minute	Increment system IS–B	F	1 to 240000mm/min	0.01 to 9600.00 inch/min
	Increment system IS–A		1 to 240000mm/min	0.01 to 9600.00 inch/min
Spindle speed function		S	0 to 20000	0 to 20000
Tool function		T	0 to 99999999	0 to 99999999
Auxiliary function		M	0 to 99999999	0 to 99999999
Offset number		H, D	0 to 400	0 to 400
Dwell	Increment system IS–B	X, P	0 to 99999.999s	0 to 99999.999s
	Increment system IS–A		0 to 999999.99s	0 to 999999.99s
Designation of a program number		P	1 to 9999	1 to 9999
Number of repetitions		P	1 to 9999	1 to 9999
Specification of angle in pattern function and nibbling function		J, P, K	±99999.999deg ±999999.99deg	±99999.999deg ±999999.99deg
Number of punch points in pattern functions		K, P	1 to 9999	1 to 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 tape 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 punching one of the following codes at the end of the program:

Table 13.2 (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.

CAUTION

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

13.3 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 ;**

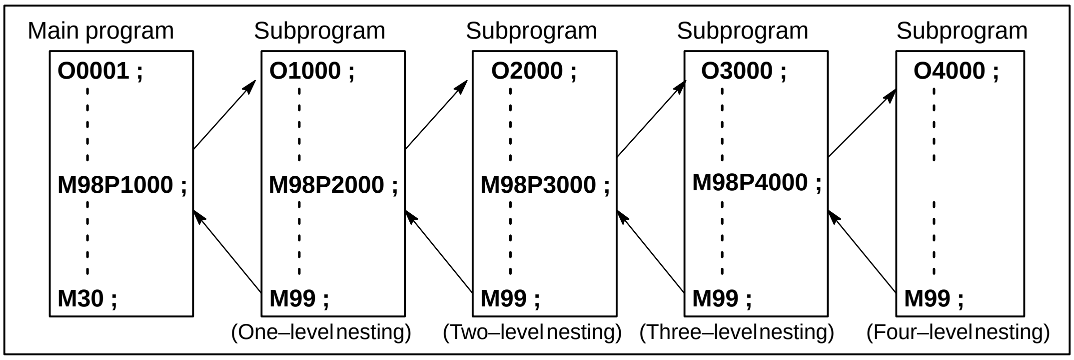
- **Subprogram call**

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

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

Explanations

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



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

- **Reference**

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

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

Examples

☆ **M98 P51002 ;**

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

☆ **X1000.0 M98 P1200 ;**

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

☆ Execution sequence of subprograms called from a main program

Main program	1 2 3	**Subprogram**
N0010 ... ;		**O1010** ... ;
N0020 ... ;		**N1020** ... ;
N0030 M98 P21010 ;		**N1030** ... ;
N0040 ... ;		**N1040** ... ;
N0050 M98 P1010 ;		**N1050** ... ;
N0060 ... ;		**N1060** ... **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/M99P<u>n</u> ; is specified, control returns not to the start of the main program, but to sequence number n. In this case, a longer time is required to return to sequence number n.

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

- **Using a subprogram only**

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

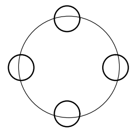
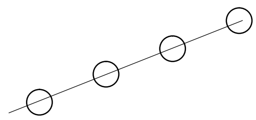
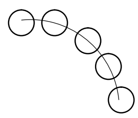
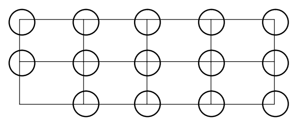
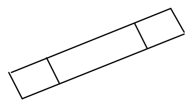
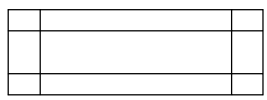
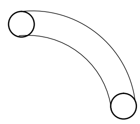
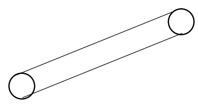
```
  N1010 ... ;
  N1020 ... ;
  N1030 ... ;
/ N1040 M02 ;                Optional block skip
  N1050 M99 P1020 ;          ON
```

14 FUNCTIONS TO SIMPLIFY PROGRAMMING

14.1 PATTERN FUNCTION

The pattern function means a function to punch multiple positions conforming to a certain format by one-block command including G function. This pattern function requires only one block command instead of several-block commands, and thus, its program is simplified.
This control system prepares the following patterns as the standard functions, which are used most frequently.

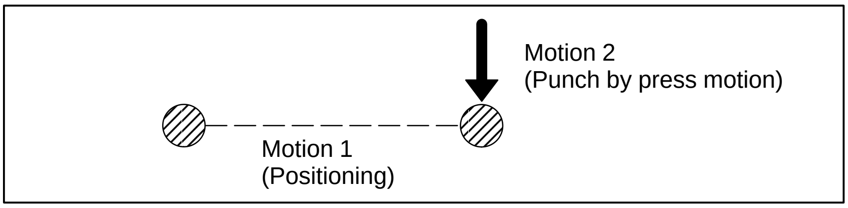
Table 14.1 Pattern functions

G code	Functions	Patterns
G26	Bolt hole circle	
G76	Line at angle	
G77	Arc	
G78 G79	Grid	
G86	Share proofs	
G87	Square	
G88	Radius	
G89	Cut at angle	

WARNING
G26, G76, G77, G78, G79, G86, G87, G88 and G89 are one-shot G codes.

In pattern function, the following two motions are repeatedly done to punch at respective positions.

Motion 1 ... Positioning of X, Y axes (rapid traverse)
Motion 2 ... Punch by press motion



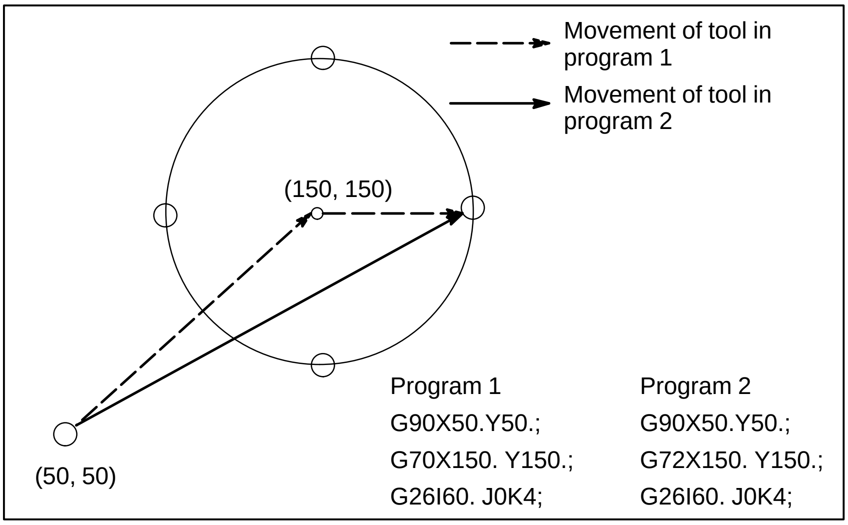
14.1.1 Base Point Command (G72)

Of the pattern function, the center point of the bolt hole circle (G26), arc (G77) and radius (G88), and also the start point of line at angle (G76), grid (G78, G79), share proofs (G86), Square (G87) and cut at angle (G89) are called base point of pattern.
The present position of the tool when the pattern function was commanded, i.e. the tool position in the block just before the pattern function command, is the base point of pattern. If it is desired to set another position as the pattern base point, the base point of pattern is assignable by the following command.

G72X_x_ Y_y_ ;

The (x, y) point in the work coordinate system is the pattern base point in absolute (G90) programming, while the point distant from the present position by (x, y) is the reference point in incremental (G91) programming.
G72 specifies the pattern base point, and the tool does not move.

Examples

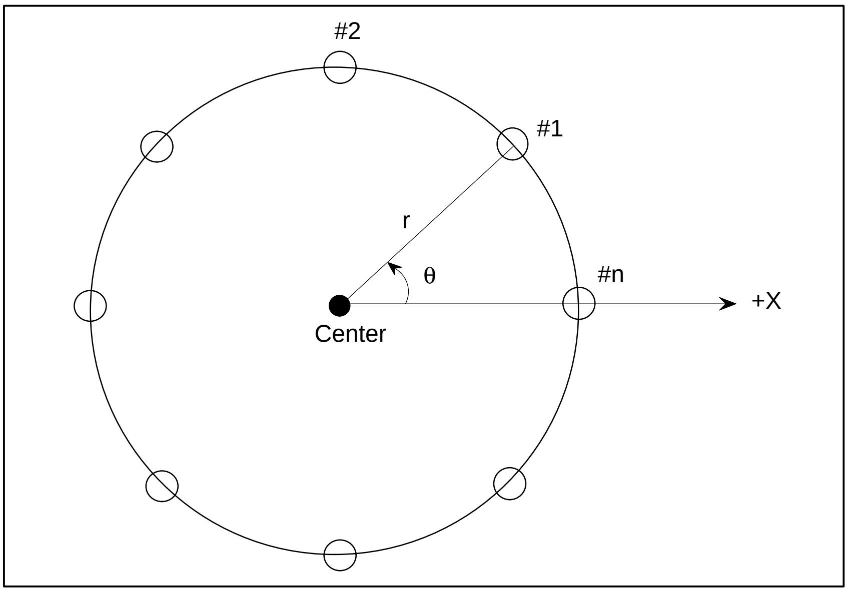


WARNING
1. None of T code, M code, and F code is commandable in G72 block.
2. G72 is one-shot G code.

14.1.2 Bolt Hole Circle (G26)

G26I_r_ J_θ_ K_n_ ;

This G26 punches n pieces of equally divided points on the circumference, starting with the point which forms angle θ with the reference to X axis on the circumference having radius r with the present tool position or the position commanded by G72 being set as the circle center.



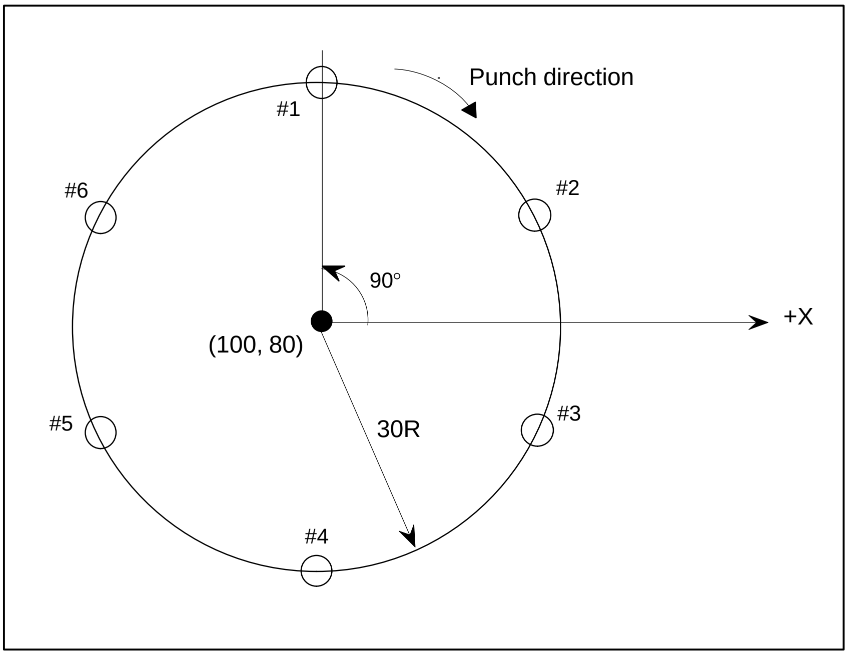
r : Radius of circle
The unit is input unit.
It is commanded by a positive number.

θ : Angle between the first punch point and +X axis
This unit is input unit (deg), and the counterclockwise direction is commanded as positive.

n : Number of punch points (±1 to ±9999)
Counterclockwise punching is made by a positive number, while clockwise punching is made by a negative number.

Examples

N521G72G90X100.0Y80.0 ;
N522G26I30.0J90.0K-6 ;



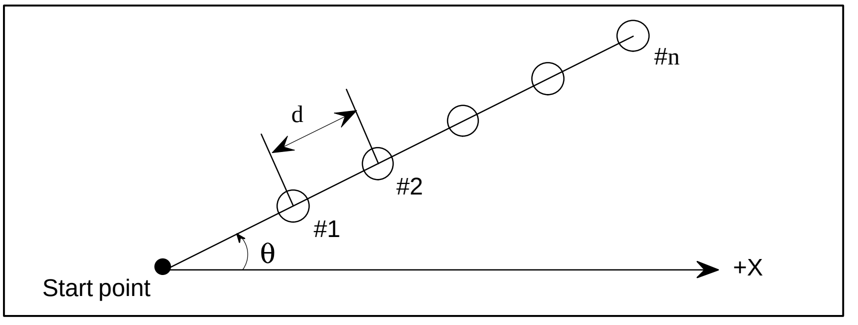
If it is desired to punch the center of the circle, omit G72 of block N521.

NOTE
1 If the radius is 0 or the number of punch points is 0, an alarm (No. 4502) is issued.
2 T and C commands are possible in same block of G26.

14.1.3 Line at Angle (G76)

G76I_d_ J_θ_ K_n_ ;

By the above command, punching is made at n pieces of points which lie every intervals of d along the straight line which forms angle θ with reference to the X axis, starting with the present tool position or the position specified by G72 as the start point.



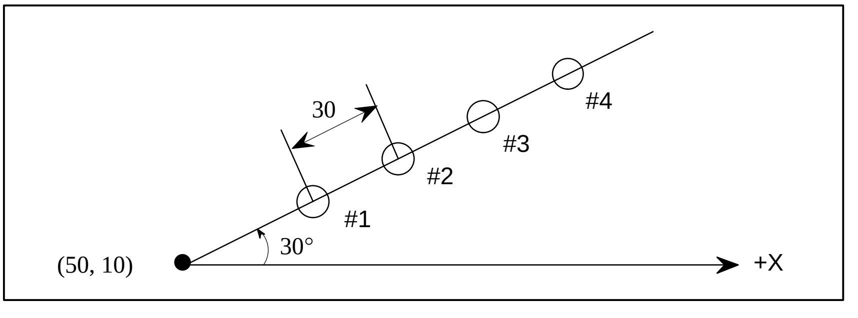
d : Punch point interval
The unit is input unit.
If this value is commanded with negative, punching is made in the θ + 180° direction.

θ : Angle which is formed between the +X axis and the line connecting the start point and punch points
The unit is input unit (deg), and the counterclockwise direction is commanded by a positive number.

n : Number of punch points (1 to 9999)

Examples

N531G00G90X50.0Y10.0T05 ;
N532G76I30.0J30.0K4 ;



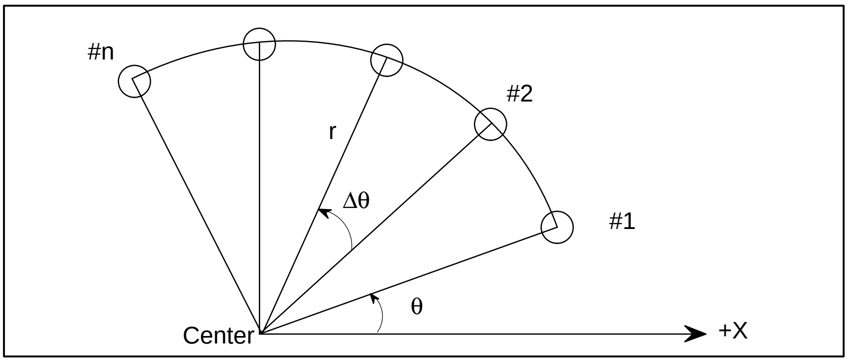
N532 block may be commanded as
G76I–30.0J210.0K4;
or G76I30.0J–330.0K4;
If it is not desired to punch the start point, command as
N531G72G90X50.0Y10.0;
N532G76I30.0J30.0K4T05;

NOTE
If the number of punch points is 0, alarm (No. 4503) is produced.

14.1.4
Arc (G77)

G77I_r_ J_θ_ P_Δθ_ K_n_ ;

By the above command, punching is made at n pieces of points every incremental Δθ angle, starting with the point which forms θ angle with reference to the X-axis on the circumference of radius r, with the present tool position or the position specified by G72 being set as the circle center.



r : Radius of arc
This unit is input unit, and this radius is commanded by a positive number.

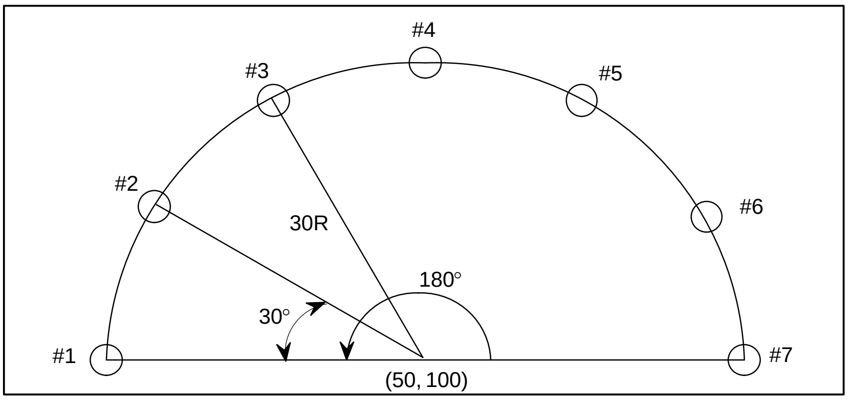
θ : Angle formed between the first punch point and the +X axis
Its unit is input unit (deg), and counterclockwise punching is commanded by a positive number.

Δθ : Angle formed between adjacent punch points
The unit is input unit (deg).
Counterclockwise punching is commanded by a positive number.

n : Number of punch points (1 to 9999)

Examples

N541G72G90X50.0Y100.0 ;
N542G77I30.0J180.0P-30.0K7 ;



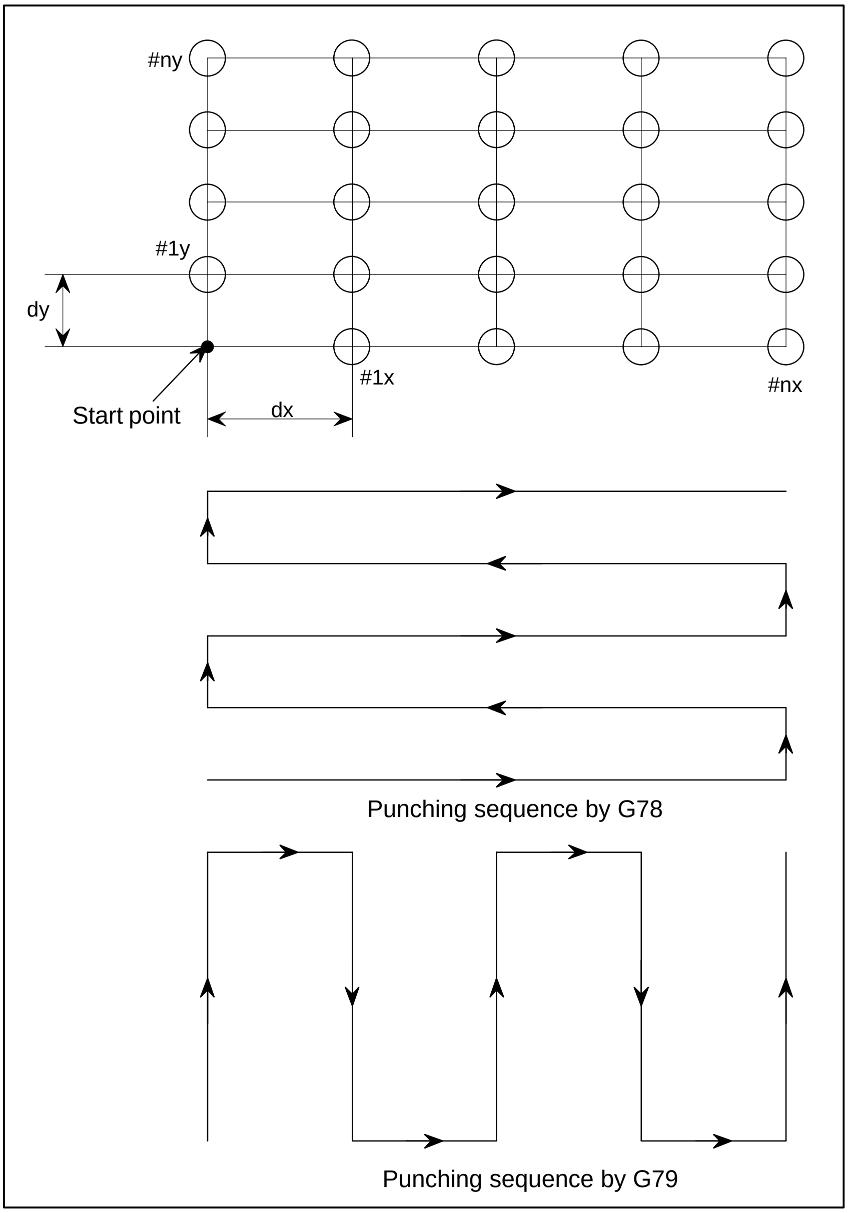
For center punch, command No. 541 block as
N541G00G90X50.0Y100.0;

NOTE
If the radius is 0 or the number of punch points is 0, alarm (No. 4504) is produced.

14.1.5 Grid (G78, G79)

G78I__dx__ P__nx__ J__dy__ K__ny__ ; or
G79I__dx__ P__nx__ J__dy__ K__ny__ ;

By the above command, punching is made at matrix points consisting of nx pieces at intervals of dx in the X-axis direction and ny pieces at intervals of dy in the Y-axis direction, i.e., (nx + 1) × (ny+1) − 1 = nxny + nx + ny pieces in total, starting with the present tool position or the position specified by G72 being set as the start point. G78 commands punching in the X-axis direction, while G79 commands punching in the Y-axis direction.



dx : Punch point intervals in the X-axis direction
This is commanded by a positive number when the first punch point in the X-axis direction is located in the +X direction as viewed from the start point.

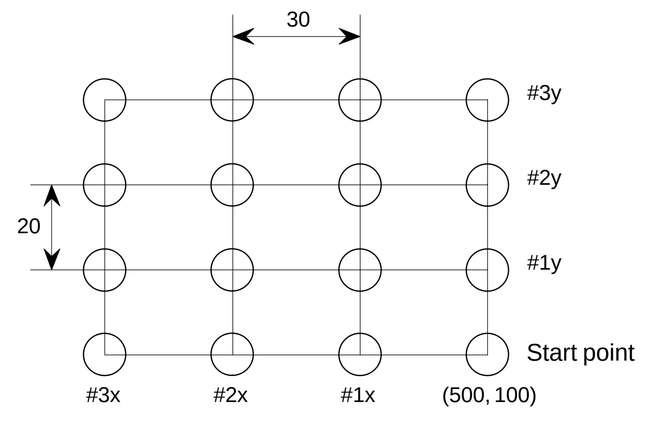
dy : Punch point intervals in the Y-axis direction
This is commanded by a positive number when the first punch point in the Y-axis direction is located in the +Y direction as viewed from the start point.

nx : Number of punch points in the X-axis direction (1 to 9999)
The start point is excluded from the number of punch points.

ny : Number of punch points in the Y-axis direction (1 to 9999)
The start point is excluded from the number of punch points.

Examples

N551X500.0Y100.0 ;
N552G78I-30.0P3J20.0K3 ;



If it is not desired to punch the start point, command N551 block as N551G72X500.0Y100.0;

NOTE
If the number of punch points is 0 in the X-axis or Y-axis direction, alarm (No. 4505) is produced.

14.1.6 Share Proofs (G86)

G86I $\underline{\quad \ell \quad}$ J $\underline{\ \theta\ }$ P $\underline{\ w_1\ }$ Q $\underline{\ w_2\ }$;

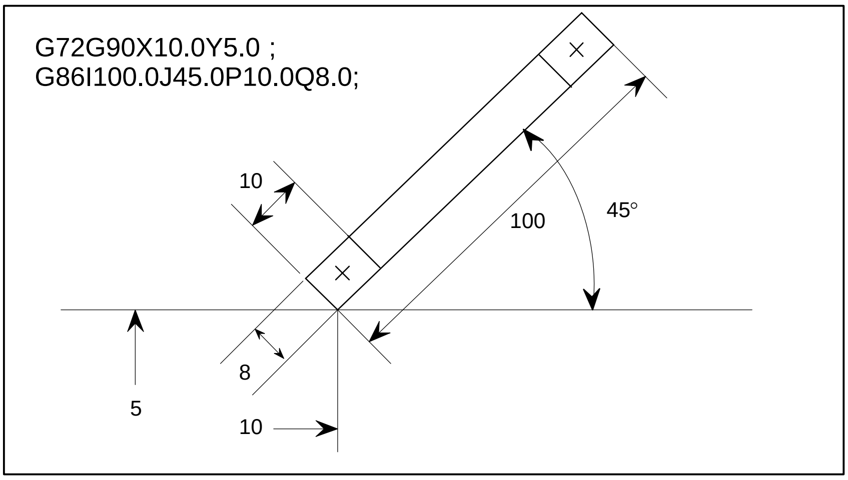
With the current position or the coordinates designated by G72 as a start point, this function allows to punch length ℓ in the direction of angle θ for the X-axis, using a rectangular tool with w_1 as the width and w_2 as the length.

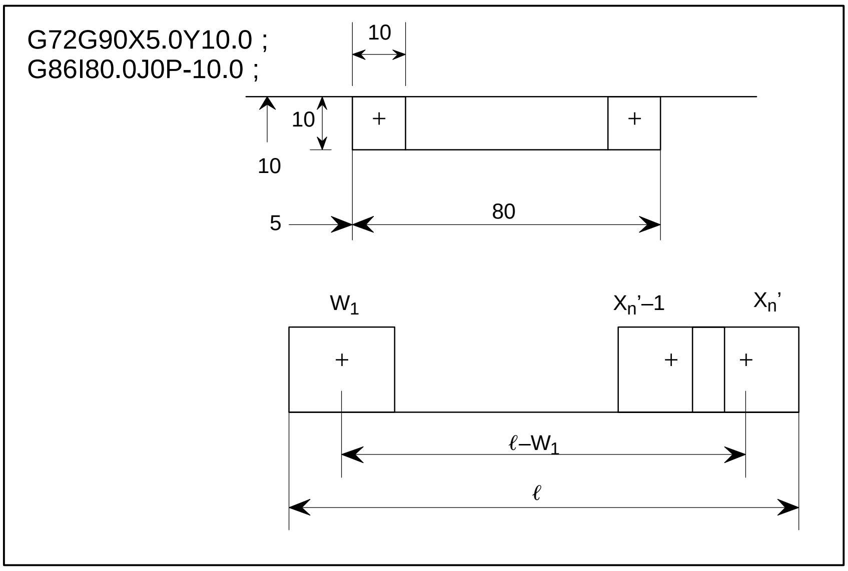
ℓ : The unit of this length is the input unit.
In the case of negative, perform punching operation in a symmetrical direction at the starting point as its center.

θ : The angle for the X-axis is fixed in integers in increments of 0.01 degrees.
The counterclockwise direction corresponds to positive.

w_1, w_2 : Tool profile size __ the unit conforms to the input unit. For positive and negative, punch the left and right sides in the forward direction, respectively. Both w_1 and w_2 must be designated when either positive or negative. For a square tool with $w_1 = w_2$, w_2 may be omitted.

Examples





The punching method is as follows.

1 Punch the first point.

2 Set the pitch to 0.95 w_1.

3 Calculate $\dfrac{\ell - w_1}{0.95\ w_1} = n$

If n ≤ 1, the pitch shall be "$\ell - w_1$". If n is an integer, the pitch shall be 0.95 w_1.
When n is not an integer, [n] + 1 = n': [ ] shows that the decimal place is omitted.

And $\dfrac{\ell - w_1}{n'}$ shall be the pitch.

At this time, there is a small calculation error of nearly 0.01 mm × n', but this error is compensated at the last punching.
That is, the last pitch shall be $X_{n'} - X_{n'-1}$.

4 According to the above method, the pitch is always as follows.
$0.45\ w_1 \leqq p \leqq 0.95\ w_1$

5 Length ℓ must be; $\ell \geqq 1.5 w_1$

14.1.7 Square (G87)

G87I $\underline{\quad \ell x}$ J $\underline{\quad \ell y}$ P $\underline{\ w_1\ }$ Q $\underline{\ w_2\ }$;

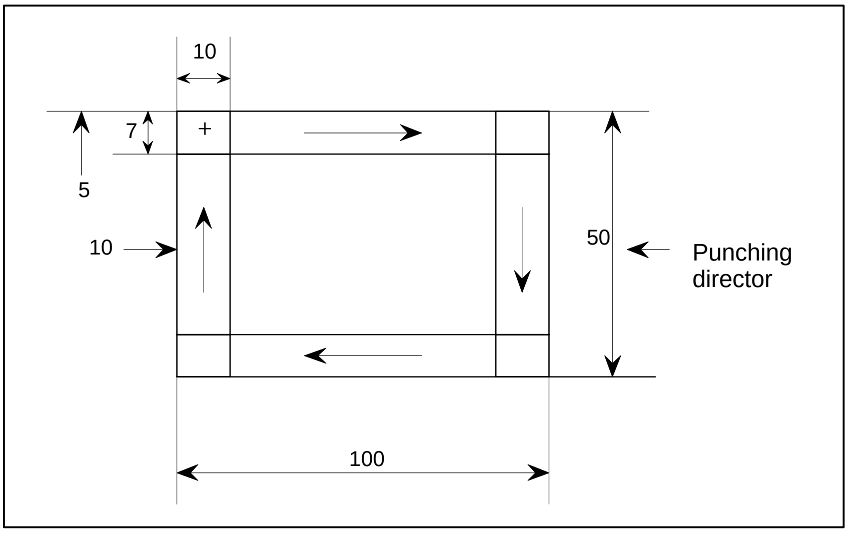
With the current position or the coordinates designated by G72 as a starting point, it allows to punch a rectangle with length ℓx in the X-axis direction and length ℓy in the Y-axis direction, using a tool with w_1 as the width and w_2 as the length.

ℓx, ℓy : Side length of rectangle
The unit is the input unit. For positive and negative, the length shall be in the +X, +Y and –X, –Y directions, respectively.

w_1, w_2: Tool profile size
The unit is the input unit. This size is designated as a positive number. For a square tool with $w_1 = w_2$, w_2 may be omitted.

Examples

G72G90X10.0Y5.0 ;
G87I100.0J-50.0P10.0Q7.0 ;



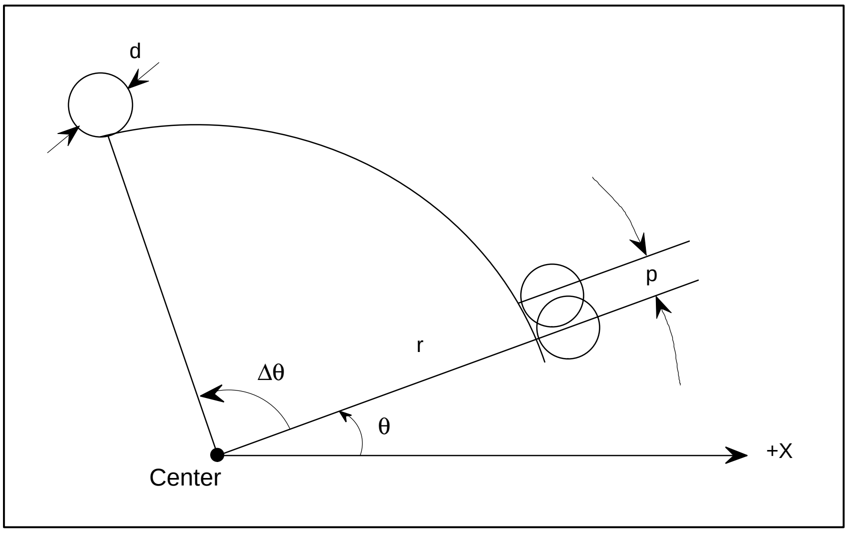
The punching operation is performed from the longitudinal direction. If $\ell x = \ell y$, punching operation will be performed in the X-axis direction. This punching ℓx and ℓy should be:

$\ell x \quad \geq 3w_1$
$\ell y \quad \geq 3w_2$

14.1.8 Radius (G88)

G88I r J θ K Δθ P d Q p ;

The punching operation is performed at pitch P between a point having angle θ for the X-axis on the circumference (diameter r) and a point having angle θ + Δθ for the X-axis with the current tool position or position designated by G72 as a center according to the above, using a tool with diameter d.



r : Radius of circle
The radius is designated in positive at input unit.

θ : Angle of the first punch point for the X-axis
The counterclockwise direction is designated in positive form inincrements of input unit (deg).

Δθ : Incremental angle from the first punch point to the last punch point
Punching operation is performed counterclockwise when positive in increments of input unit (deg).

d : Tool diameter
The incremental unit is input unit.
It allows to punch the circle outside, the circle inside, and the circumference, when positive, negative, and 0, respectively.

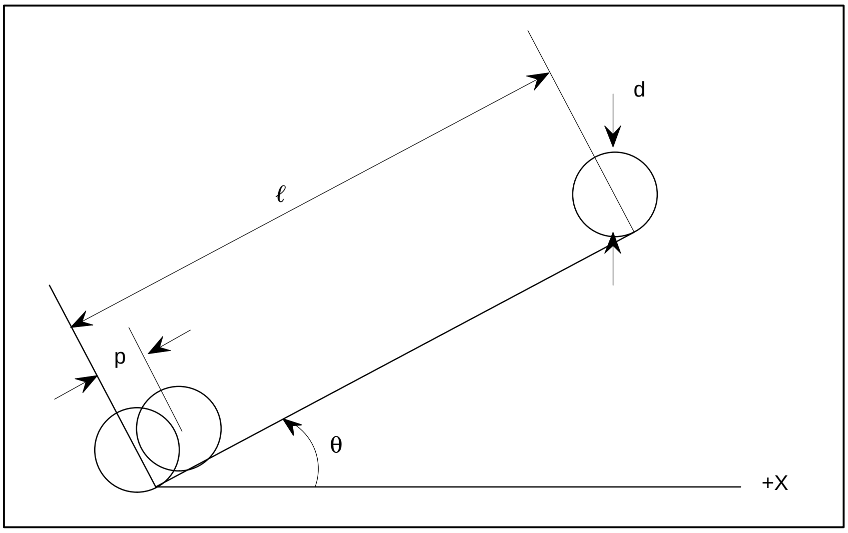
p : Pitch
Designate the circular length in input unit, respectively.

Radius (G88) is the same as with circular nibbling (G68) except that there is no pitch limit value by parameter setting (No. 468 and 469), and it stops at each punch point for a single block. Therefore, also refer to the Circular Nibbling (G68) paragraph given under 9.3.1.

14.1.9 Cut at Angle (G89)

G89I__ℓ__ J__θ__ P__d__ Q__p__ ;

This function allows to punch a straight line with length ℓ having angle θ to X-axis with the current tool position or the position designated by G72 as a starting point, at pitch P, using a tool with diameter d.



ℓ : Straight line length
The unit is input unit. When a negative number is designated, the angle θ to a straight line and the +X axis is θ + 180 degrees.

θ : Angle of straight line to the +X-axis
Designate the counterclockwise direction at positive in increments of input unit (deg).

d : Tool diameter
The unit is input unit.
The tool is offset by d/2 to the left when positive and to the right when negative from the straight line start point to the end point for nibbling.
Perform punching operation on the straight line at 0.

p : Pitch
The unit is input unit. Designate the pitch with the length in the straight line direction.

Cut at angle (G89) is the same as with straight line nibbling (G69) except that there is no pitch limit value by parameter setting (No. 468 and 469) and it stops at each punching point for a single block; therefore, refer to 9.3.2 “Straight Line Nibbling (G69)” as well.

14.1.10 Incremental Command Just After Pattern Function

If an incremental command is given in a block just after the pattern function, the tool may not move by the incremental amount from the end point of the pattern function. In case of bolt hole circle (G26), share proofs (G86), square (G87), radius (G88) and cut at angle (G89).

The amount of movement attained by an incremental command just after the bolt hole circle is the commanded incremental amount from the center of the bolt hole circle.

In case of bolt hole circle (G26) and square (G87), the incremental amount is from the base point of pattern, namely, center and start point respectively.

In case of share proofs (G86), radius (G88) and cut at angle (G89), the incremental amount is from the end point of the program.

The incremental amount is all started with the end point of the pattern function in line at angle (G76), arc (G77), and grid (G78, G79).

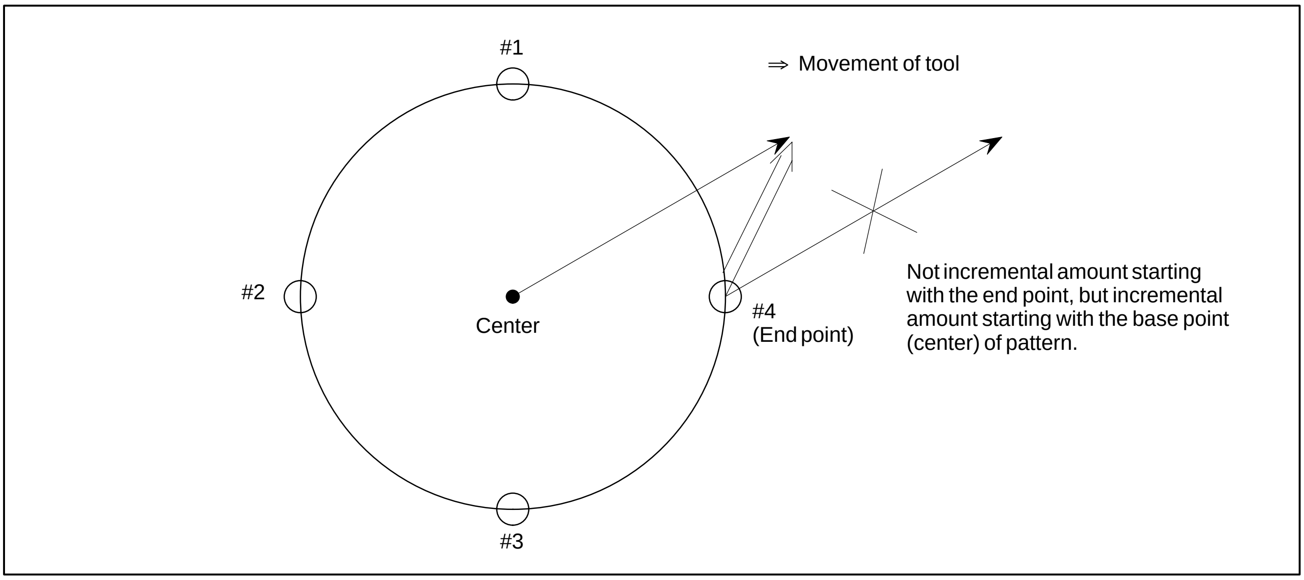


Fig. 14.1.10 (a) Incremental command just after bolt hole circle (G26)

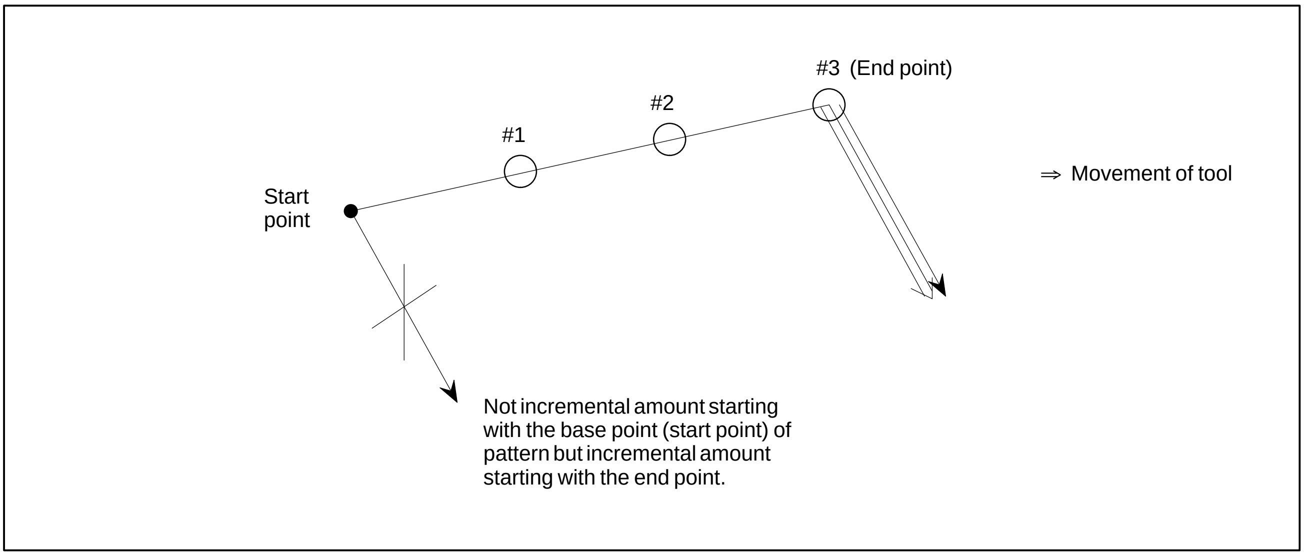
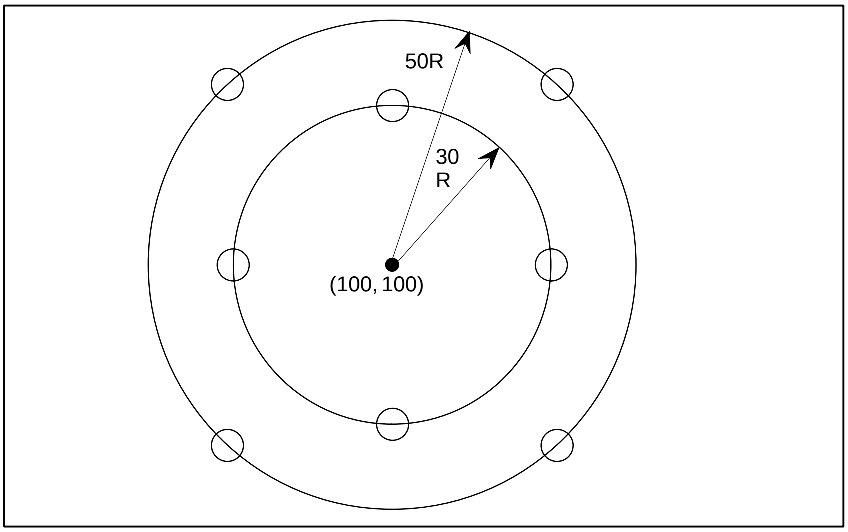


Fig. 14.1.10 (b) Incremental command just after line at angle (G76)

When the block execution of the bolt hole circle (G26) has been finished, the tool is located at the end point, in practice.
However, a programmer shall make a program assuming that the tool be located at the base point, i.e., the center of the pattern.
Thus, programming is simplified in machining, like the following example.

Examples

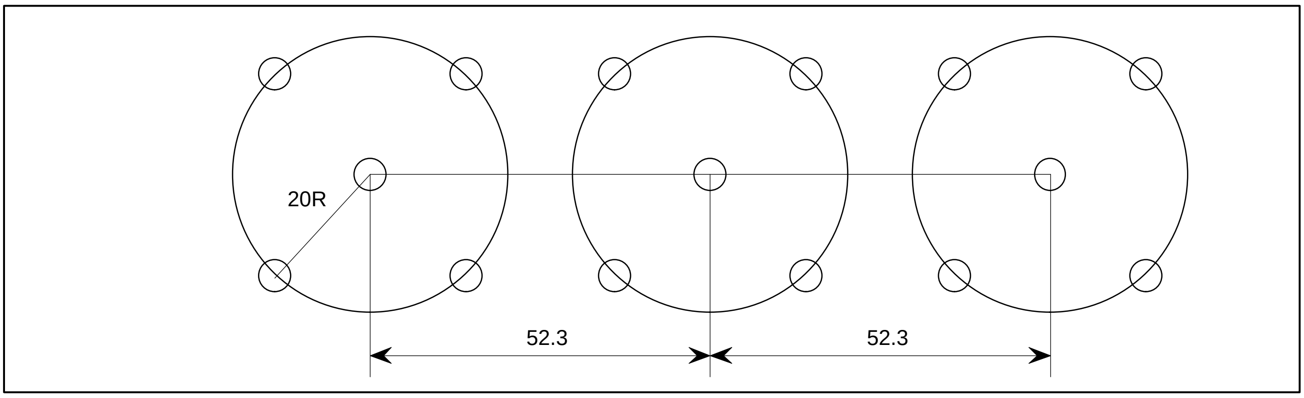
Concentric bolt hole circles



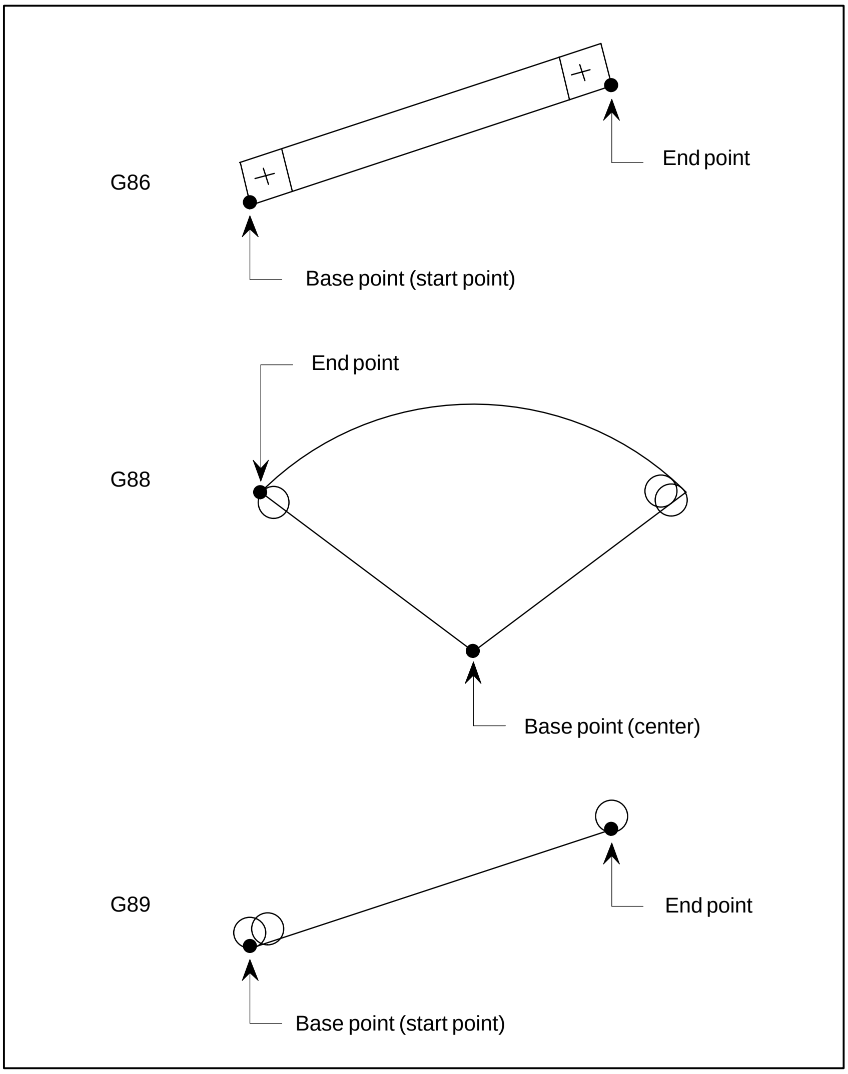
N561G72G90X100.0Y100.0 ;
N562G26I30.0J0K4 ;
N563G26I50.0J45.0K4 ;

It is not required to command G72G90X100.0Y100.0 in the block next to N562.

Machining when the centers are given as an incremental sizes



N564G00G90X48.5Y50.0 ;
N565G26I20.0J45.0K4 ;
N566G91X52.3 ;
N567G26I20.0J45.0K4 ;
N568X52.3 ;
N569G26I50.0J45.0K4 ;


G86
End point
Base point (start point)
End point
G88
Base point (center)
G89
End point
Base point (start point)

14.1.11 Notes on Pattern Functions

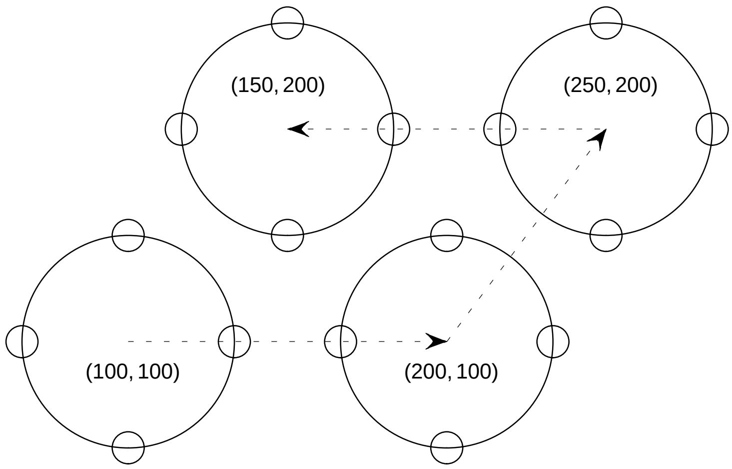
WARNING

1 Don't command M code in a block where the pattern function is commanded.
2 If a T code is commanded in a block where the pattern function is commanded, the X, Y axes are positioned to the first punch point and also a tool is selected concurrently. After positioning and tool selection, punching is made at the first punch point.
3 Even when the pattern function is commanded in G01, G02 or G03 mode, the X, Y axes are positioned at rapid traverse. However, since T code cannot be commanded in G01, G02, or G03 mode, such a command is not executable, in general.
4 If a pattern function is executed using the single block operation, the single block stop is made after punching has been finished at respective punch points.
5 If the pattern function is commanded in MDI mode, punching is not executed, but positioning only is made. Some machines may execute punching.
6 Since radius, number of punch points, and other parameters in the pattern function are not modal data, they must be commanded correctly in each block.
7 Operation for the C-axis can be specified in a block in which the pattern function is specified.

14.2 MEMORY AND CALL BY A/B MACRO

When it is desired to repeatedly use a pattern with the same figure among the pattern functions, this function can store it in memory with a given number and access it as needed. Programs other than those using the pattern functions can be stored in memory and can be called.

1) Memory
   When a one-digit numeral from 1 to 5 subsequent to address A is given before the G code of the pattern function, the pattern function according to the G-code is stored in the memory.
   (Example) A2G76I300J3000K5;
2) Call
   By giving a one-digit numeral from 1 to 5 subsequent to address B, it is possible to call the pattern function stored previously with address A.
   (Example) B2;
3) Example
   When bolt hole circles with the same figure having a center in (100, 100), (200, 100), (250, 200), and (150, 200) are present:

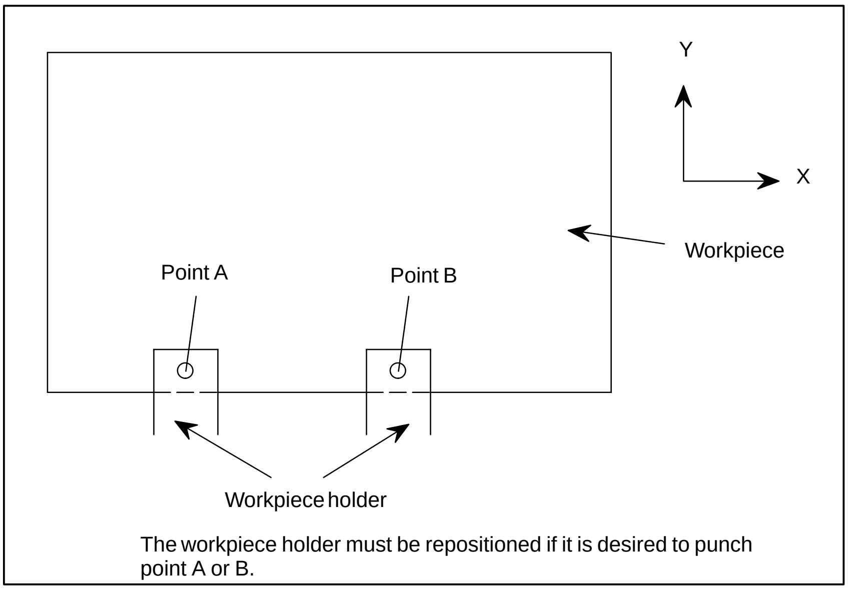


:	
G72X100.Y100. ;	
A2G26I50.J0K4 ;	← Designated for storage after execution
G72X200.Y100. ;	
B2 ;	← Recall of pattern A2
G72X250.Y200. ;	
B2 ;	
G72X150.Y200. ;	
B2 ;	
:	

WARNING
1 Do not give the same number because the pattern function differs. At this time, the previous memory is erased.
2 Only a block with up to 192 characters can be stored in memory.

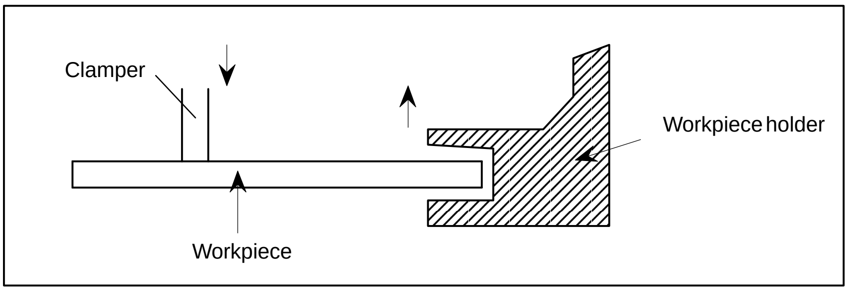
14.3 AUTOMATIC REPOSITIONING (G75)

By changing the hold position of a workpiece by the workpiece holders, a workpiece having a size larger than the stroke in X-axis direction of the machine can be machined.
If it is desired to punch a workpiece at the workpiece holder position when the workpiece was set to the machine, the hold position of the workpiece must be changed.

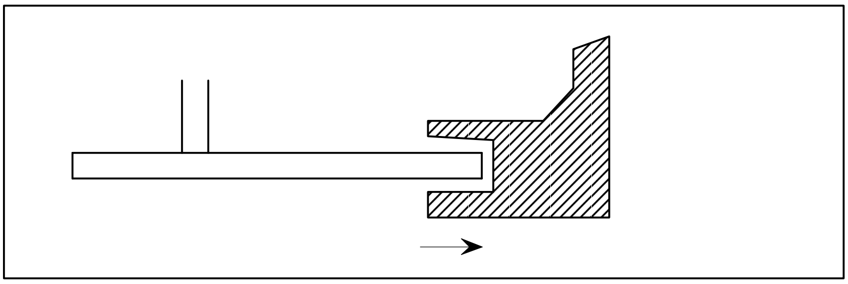


Repositioning of a workpiece is generally done according to the following procedure, assuming that the workpiece is positioned at a location where the repositioning of the workpiece is executable.

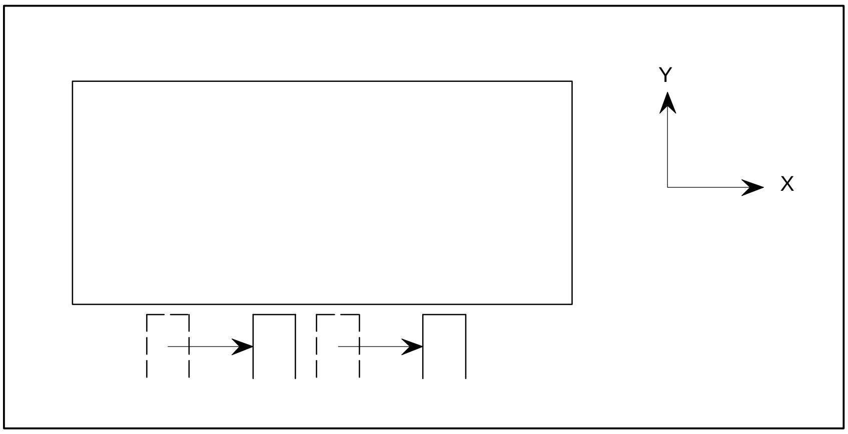
1) The claw of the workpiece holder is opened, and also the clamper depresses the workpiece concurrently to fix the workpiece as a general procedure, so that the workpiece is not deviated from the table.



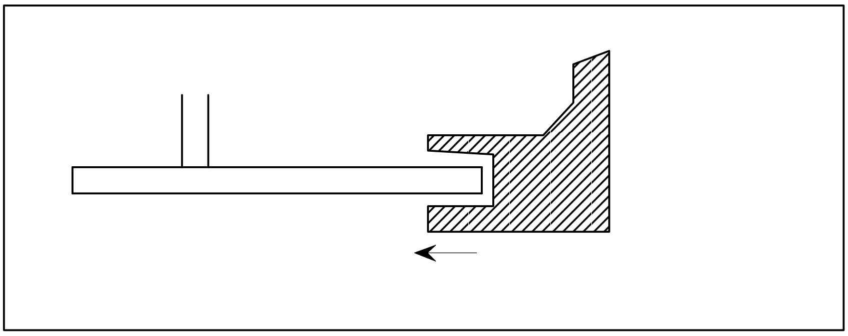
2) The workpiece holder moves in the Y-axis direction and separates from the workpiece.



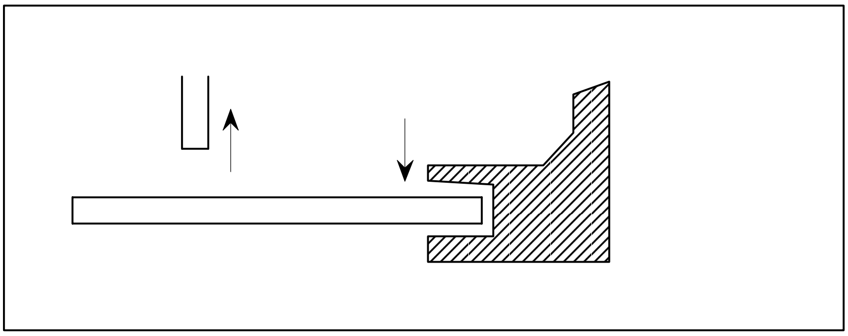
3) The workpiece holder moves in the X-axis direction to relocate the hold position.



4) The workpiece holder moves in the Y-axis direction to return to the position where it can hold the workpiece.



5) The claw of the workpiece holder is closed to hold the workpiece, and the clamper lifts and separates from the workpiece concurrently.



A series of the above operation can be done by one-block command including G function.

G75X $\underline{\ x\ }$;

The above command is executed by being divided into the following 5 blocks.

(1) M10;

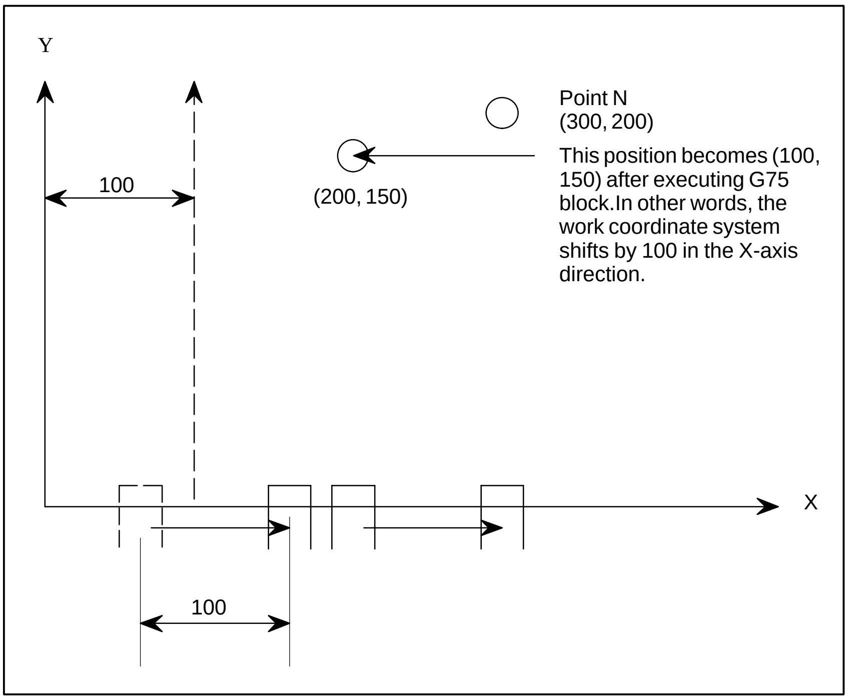
(2) G70G91 $\underline{\ y_R\ }$;

(3) G70G91X $\underline{\ -x\ }$;

(4) G70G91Y $\underline{\ -y_R\ }$;

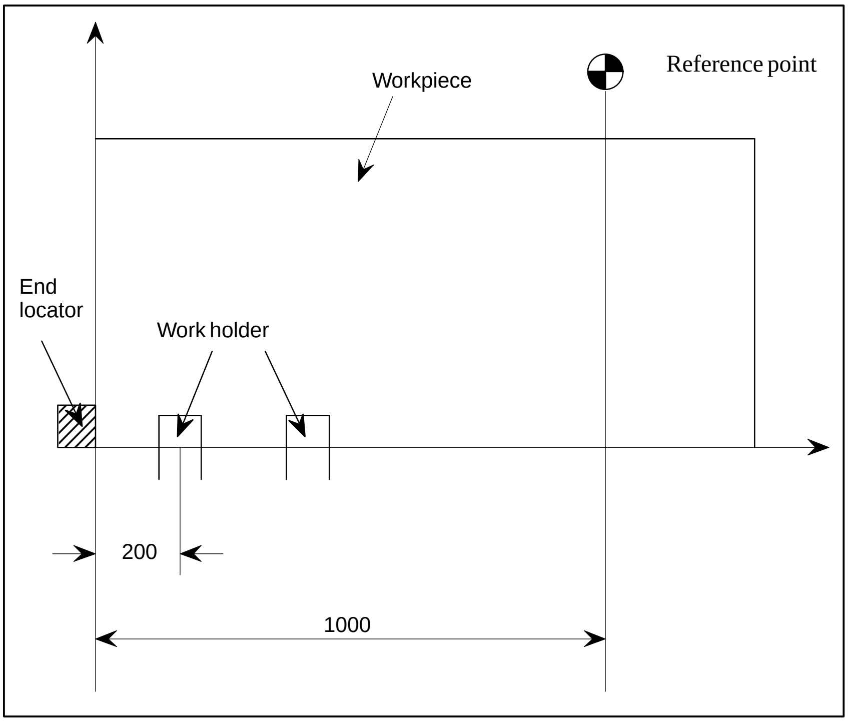
(5) M11;

Blocks (1) - (5) correspond to operation steps 1) - 5), respectively. A relief or a return mount R in the Y-axis direction is preset by parameter (No. 16209: for metric input, No. 16210: for inch input). For this amount, refer to the machine tool builder's manual.
By G75 command, the X and Y axes move.
The movement of the Y-axis is just cancelled by its relief and return. If it is assumed that the movement of the X-axis is the movement of a tool in the same manner as in ordinary block, for the X-axis command hereafter, X-axis command value must be subtracted by the value displaced by G75 or a coordinate system must be set by a G92 command with the tool at the position where the G75 was commanded in the block next to G75.



G90X200.0Y150.0 ;
G75X100.0 ;
X200.0Y200.0 ;
(Point N cannot be commanded as X300.0Y200.0; if the CNC does not process in it.)

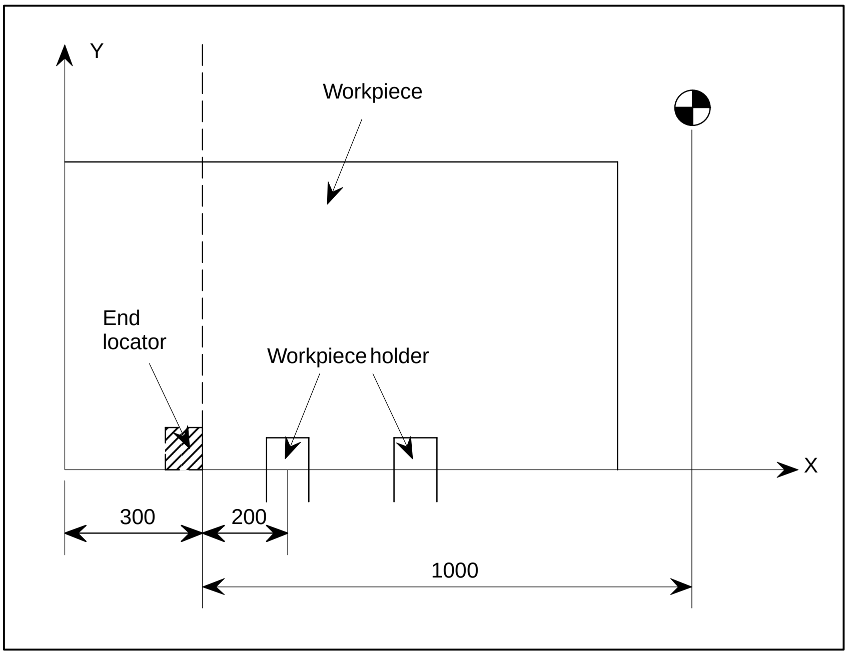
The movement of the X-axis for repositioning changes the workpiece holder position. The tool position remains unchanged in the work coordinate system, in practice.
Accordingly, internal processing is made in this control system so that the coordinate value of the tool is not changed by the axis movement performed by repositioning.
Thus, it is not necessary to take the repositioning of a workpiece into consideration in programming.
Now, the following description covers such a case that several workpieces are sequentially machined according to a program including the repositioning command.



Refer to the above figure as an example.
The reference point is assumed as the start point of the tool. Assume that the distance between the reference point and the end locator is 1000 mm, and the left workpiece holder is mounted at a place distant from the end locator by 200 mm.
In order to machine the next workpiece by the same program after machining one workpiece, the start point of the tool must also be the reference point to the next workpiece.
Accordingly, programming must be made to set the end point to the reference point.

```
G92X1000.0Y . . . ;
. . . . . . . . . .
. . . . . . . . . .
. . . . . . . . . .
. . . . . . . . . .
G75X300.0 ;
. . . . . . . . . .
. . . . . . . . . .
. . . . . . . . .
G28M30 ;
```

“The end point is the reference point.” means that the position of the end point is just the same as the start point on the machine tool, and the workpiece holder position is the start point, i.e., the position distant from the end locator by 200 mm.
However, the left lower point of the workpiece, i.e., the zero point of the work coordinate system, does not meet the end locator position, but it is distant from the end locator by 300 mm in the –X direction.



When a new workpiece is set by attaching it to the end locator after removing the workpiece, the zero point of the work coordinate system must be positioned at the end locator.
Accordingly, the coordinate system must be set by G92 command at the start of the program without fail.

WARNING

1 Neither T code nor M code should be commanded in G75 block.

2 The repositioning amount of the workpiece holder is specified by a numerical value following address X in G75 command. If repositioning is made in the +X direction of the work coordinate system, specify it by a positive numerical value.

3 Other M codes may be used for M10 and M11 depending upon machine tool builders.

4 Programming is made, assuming that the workpiece is fixed and a tool moves along the workpiece.
However, the workpiece moves, while the tool is fixed generally. Accordingly, the movement direction of the workpiece, or, the movement direction of the workpiece holder, is contrary to the movement direction of the tool.
This means that, when the workpiece holder moves in the +X and +Y directions of the work coordinate system, the tool must be regarded to move in the –X and –Y direction respectively. Accordingly, the command in (2), (3), (4) blocks is opposite to movement direction in the work coordinate system of the workpiece holder.

5 In G75 command, the relief quantity of the workpiece holder from the workpiece is equal to the return quantity of the workpiece holder to the workpiece in the absolute value of its movement amount.
However, if a workpiece is uneven, the workpiece holder depresses the workpiece, causing the workpiece to be deviated, assuming that the workpiece holder returns by the relief quantity as it is.
For such a work, change the relief quantity and return quantity by the following programming.

```
    M10;
    G91Y  y1  ;
    X  –x  ;
    Y  –y2  ;        y1 differs from y2.
M11;
```

Even in the above command, internal processing is made in NC so that the tool position remains unchanged in the work coordinate system, irrespective of the movement of the X and Y axes.
Accordingly, it is not necessary for programming to take this movement into consideration in the subsequent commands.
However, since the workpiece cannot be held if such a command is given again, it is necessary to take these circumstances into consideration when performing repositioning again.

6 It is not recommended for cycle time and machining accuracy to repeatedly execute repositioning by a program.

7 The single block stop is made after five sequential motion, if G75 is executed by a single block operation.

14.4 MACRO FUNCTION

The macro function enables commands consisting of several blocks to be stored in the NC memory as a single macro and to be called when necessary.

14.4.1 Storage of Macros

To store several blocks as a single macro, attach numerics of 2 digits (01 to 89) following address work U before commanding several blocks to be stored and attach the same numerals following address word V after them, then the several blocks sandwiched in between U and V are stored.
We call the numerics of 2 digits following U or V "macro number". If the macro number is any of 01 to 59, then blocks sandwiched in between U and V are stored with their execution. If the macro number is any of 60 to 89, only storage of blocks is performed without their execution.

Examples

```
;
G92X1830.0Y1270.0 ;
      . . . . .
      . . . . .
U02 ;
G90X100.0Y500.0T32 ;    ┐
G72X150.0Y700.0 ;       │
G87I100.0J300.0P10.0 ;  │
N100M100 ;              ├ Storing while command
G72X500.0Y300.0T26 ;    │ executing
A03G26I100.0J0K4 ;      │
G72X800.0Y300.0 ;       │
B03 ;                   ┘
V02 ;
      . . . . .
      . . . . .
U70 ;                   ┐
G90X200.0Y100.0T20  ;   │
G79I8.0J10.0P3K10 ;     ├ Storing without command
G90X1000.0Y50.0T31 ;    │ executing
G76I25.0J60.0K6 ;       │
V70 ;                   ┘
      . . . . .
      . . . . .
```

In the blocks to be sandwiched in between U and V, all commands other than ones for storage of macros according to other macro numbers (see example) may be specified. <u>But controller resetting commands, such as M02, M30, etc., can not be specified.</u> In addition, the part for custom macros cannot be specified.
Address words U and V must be specified in independent blocks except for the sequence number (a numeric string of 5 digits following N).
Details of macro numbers 90 to 99 will be described later.

Examples

```
        . . . . .
        . . . . .
U10 ;
G90X10000Y50000T32 ;
G72X15000Y70000 ;
G87I10000J30000P1000 ;
N100M100 ;
U20 ;
G72X50000Y30000T26 ;
A03G26I10000J0K4 ;
G72X80000Y30000 ;
B03 ;
V20 ;
G90X20000Y10000T20 ;
V10 ;
```

As shown in the above example, another macro cannot be stored while a certain macro is being stored.

14.4.2 Macro Call

Several blocks stored previously as a single macro can be called by specifying numerics of 2 digits attached following address word W.

Examples

```
        . . . . .
        . . . . .
U05 ;
G90X100.0Y500.0T32 ;
G72150.0Y700.0 ;
G87I100.0J300.0P10.0 ;
M00 ;
V05 ;
        . . . . .
        . . . . .
G93G90X5000.0Y0 ;
W05 ;
        . . . . .
```

Address word W must be specified in a independent block except for the sequence number. But as regards the commands G73 and G74 for the multi-piece machining function, W can be specified together with their G codes.

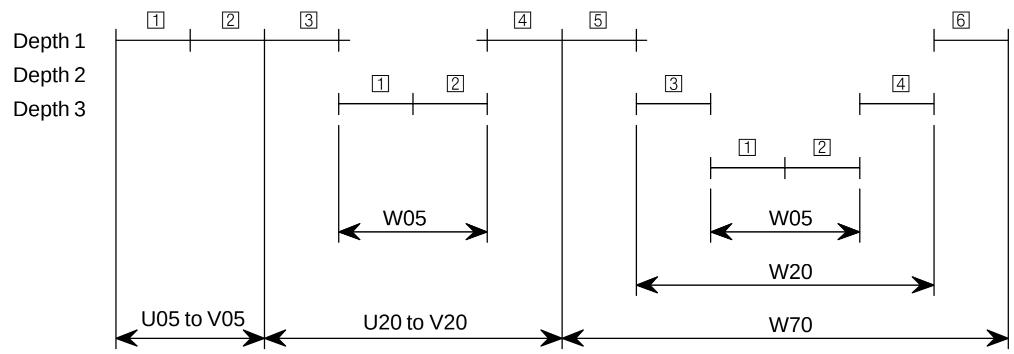
14.4.3 Nesting Call of Macros

A certain macro can call another macro and then further the latter macro can call any other macro. The depth of nesting call is up to 3.

Examples

```
U05 ;                  [1] to [6] : Signifies a block number
 . . . .  ; [1]
 . . . .  ; [2]
V05 ;
U20 ;
 . . . .  ; [3]
   W05 ;
 . . . .  ; [4]
V20 ;
U70 ;
 . . . .  ; [5]  }
   W20 ;         }  These are not executed during storage because
 . . . .  ; [6]  }  the macro number is 70.
V70 ;
W70 ;
```

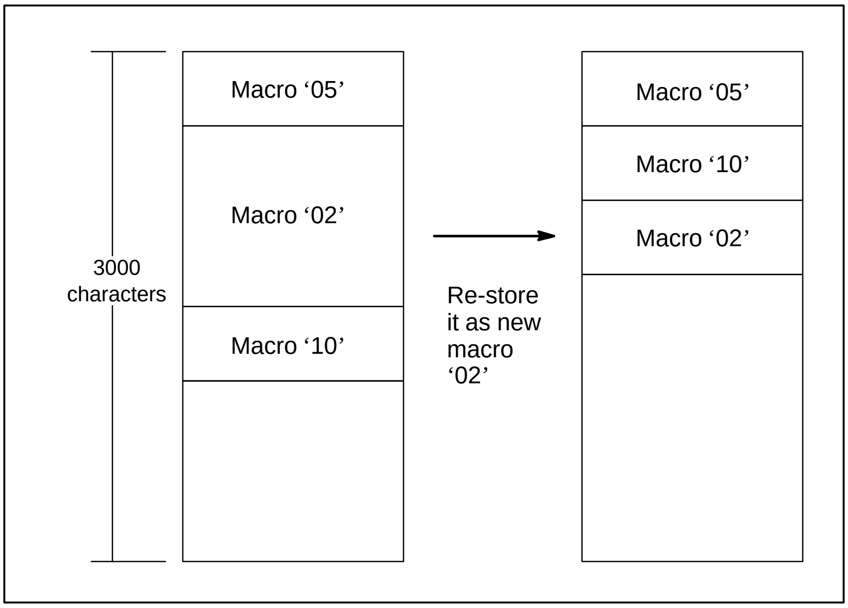
The execution sequence of this example is as follows:



Multiplexed calls can be nested up to three deep.

14.4.4 Macro Storage Capacity

The storage capacity of each macro 01 to 89 is variable. However the entire storage capacity is limited to 3200 characters.
Effective use of the storage area for storing macros is guaranteed since previously stored macros are erased if they are changed to new macros with the same macro numbers as before when they are out of use.
The maximum number of characters that can be stored can be increased up to 27072, by setting parameter No. 16228 accordingly
This feature is illustrated as follows:



All characters between address word U and the End of Block (EOB) code of a block which address word V is specified are stored in the macros storage area, provided that characters to be ignored, such as the space code, etc., and characters from the control-out code to the control-in code are not stored.
Characters of any block in which the slash (‘ / ’) code for the optional block skip has been specified are not stored if the OPTIONAL BLOCK SKIP switch is on when storing. They are stored in the macro storage area if the OPTIONAL BLOCK SKIP switch is off when storing, the on/off of the OPTIONAL BLOCK SKIP switch is valid at call time.

Examples

```
 . . . . .
N100  (U10 ;
G90X10000Y5000T20 ;
N101A01G26I4000J9000K10 ;
G91X10000 ;
N102B01 ;
N105V10 ; )
 . . . . . .
```

Characters in brackets (   ) are stored in memory.

14.4.5 Storage and Call of Multiple Macros (Macro Numbers 90 to 99)

With macro numbers 90 to 99, several macros can be stored and called as a single macro, though the item 14.4.1 “Storage of macros” describes that another macro cannot be stored while a certain macro is being stored. Macro numbers 90 to 99, however, only define several macros as a single macro, but they cannot store any executable commands by themselves. So, no memory prepared for macros is used. Macro numbers 90 to 99 are prepared mainly for simplifying programs in the multi-piece machining function.

Examples

```
U90 ;
   U01 ;
     . . .
     . . .
   V01 ;
   U10 ;
     . . .
     . . .
   V10 ;
   U15 ;
     . . .
     . . .
   V15 ;
   G90X10000Y20000T15 ; ← This block is not stored at all.
   U02 ;                  (such a use is no good)
     . . .
     . . .
   V02 ;
V90 ;
     . . .
     . . .
W90 ; ← This command is equivalent to the following set of several
        commands.
          W01 ; ⎫
          W10 ; ⎬ =W90
          W15 ; ⎪
          W02 ; ⎭
```

The number of macros storable by each of macro numbers 90 to 99 is up to 15.

14.4.6 Deletion of Stored Macros

Macros stored therefore are all deleted in the following cases:

(1) Reset (including reset due to M02, M30, etc.)

(2) Controller power off

14.5 MULTI–PIECE MACHINING FUNCTION

The multi-piece machining function enables several sheets of product with the same punching shape to be produced from a single sheet of material at a time by simple commands.
This function allows so called “trial machining” that performs punching only on a sheet of product from the machining command tape for “multi-piece machining” by a simple setup method, therefore the machining command tape can be easily checked before full machining.

14.5.1 Base Point Command of Multi-Piece Machining (G98)

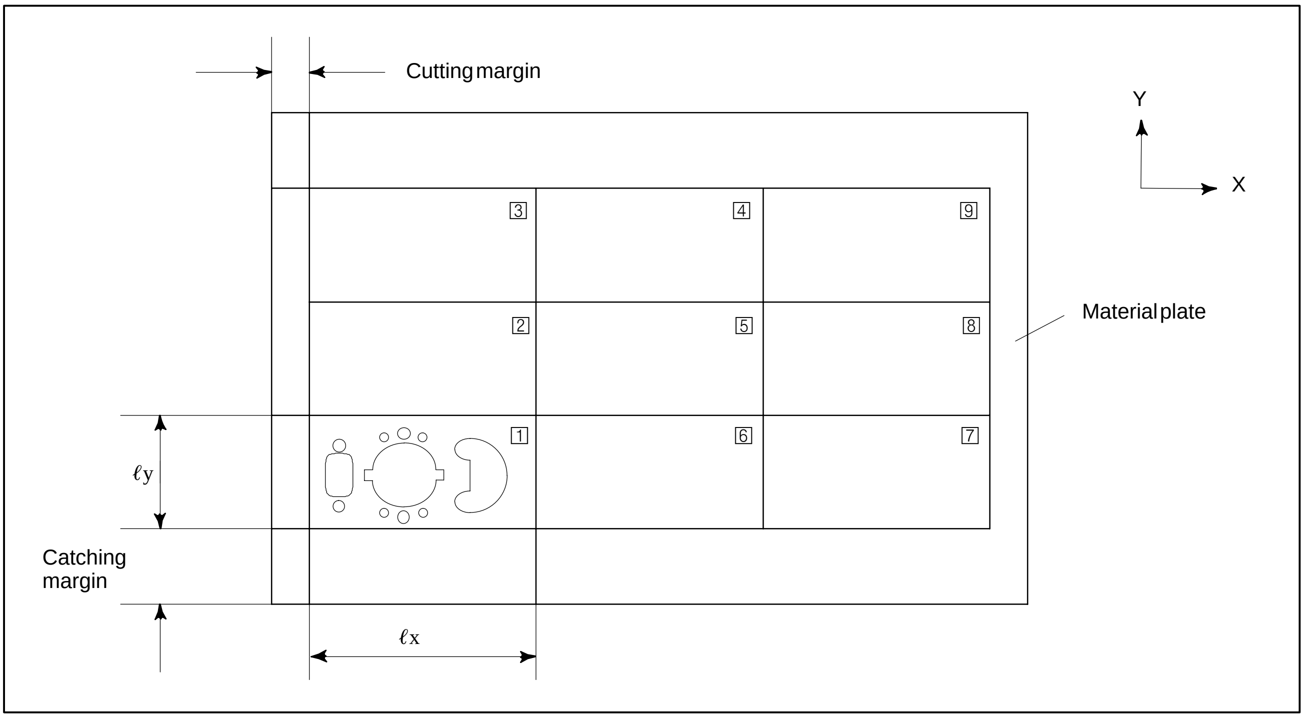


Fig. 14.5.1 (a)

Parts $\boxed{2}$ to $\boxed{9}$ as shown above have the same punching shape as part $\boxed{1}$. Machining commands to punch on a sheet of material must be specified on the product part at the lower left ($\boxed{1}$).
The point at the lower left of the set of multi-products (point B shown in Fig. 14.5.1 (a); called as “Base point of multi-piece machining” hereafter) must be specified prior to the machining commands to punch on a product part by G98 as absolute co-ordinates under the system of co-ordinates specified using G92 command. In the G98 command, the X-axial and Y-axial lengths of one product and the numbers of products in the directions of X-axis and Y-axis must be specified.

G98X $\underline{\ xb\ }$ Y $\underline{\ yb\ }$ I $\underline{\ \ell x\ }$ J $\underline{\ \ell y\ }$ P $\underline{\ nx\ }$ K $\underline{\ ny\ }$;

xb: X-axis coordinate value of the base point of multi-piece machining
yb: Y-axis coordinate value of the base point of multi-piece machining
ℓx: X axial length of one product part (a positive number)

ℓy: Y axial length of one product part (a positive number)
nx: The number of products in the X axial direction (Note)
ny: The number of products in the Y axial direction (Note)

> **NOTE**
> Product part $\boxed{1}$ is not counted.

After command of G98, specify machining commands on the lower left product part ($\boxed{1}$ in Fig. 14.5.1 (a)) using the macro function (specify the machining commands between address word U and V).
In the multi-piece machining function, if a set of machining commands corresponding to a single tool is executed continuously on the all product parts, the tool change count is decreased so that the time required for full machining on a material sheet can be considerably decreased. The following illustrates this.

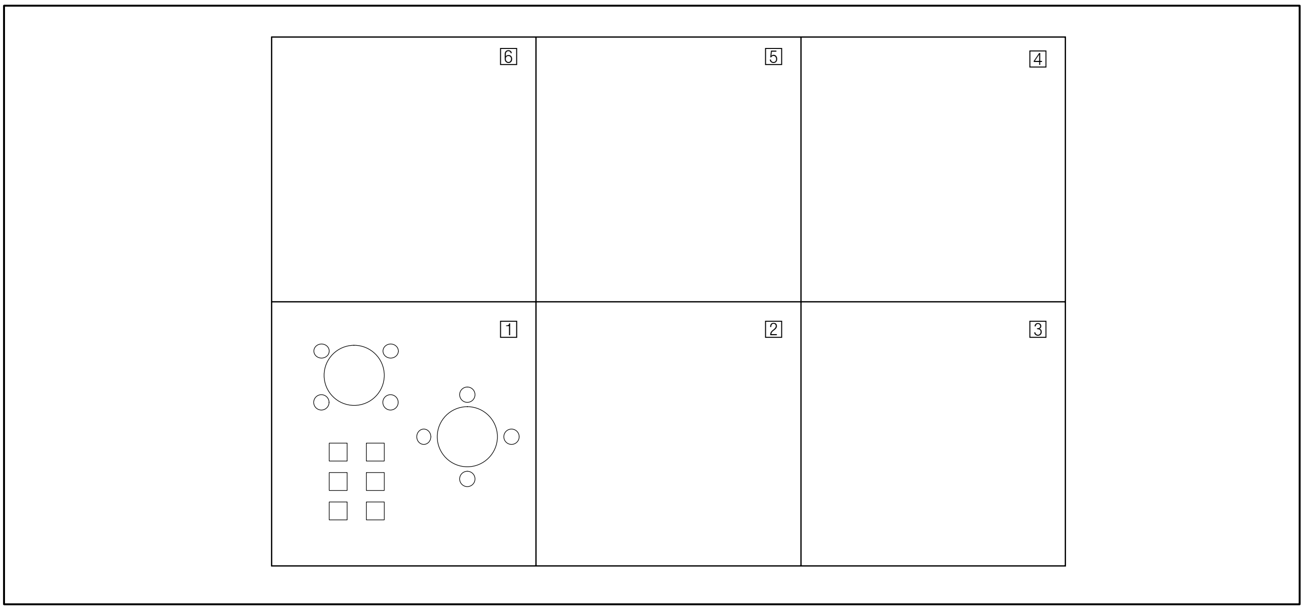


Fig. 14.5.1 (b)

Let the following four types of tools be used to punch products for one sheet, as shown in Fig. 14.5.1 (b).

□ T31
∘ T11
○ T22
◯ (with T33 in nibbling mode)

Method 1

In this method, the machining commands to punch on one product part are stored as a single macro.

```
U01 ;
  . . . .   T31 ;
  . . . . . . . . .
  . . . . . . . . .
  . . . .   T22 ;
  . . . . . . . . .
  . . . . . . . . .
  . . . .   T33 ;
  . . . . . . . . .
  . . . . . . . . .
  . . . .   T11 ;
  . . . . . . . . .
  . . . . . . . . .
V01 ;
```

This method will run the risk of taking longer time for machining on a sheet of material because tool change is performed in the order, T31 → T22 → T33 → T11, while the respective product parts from 1 to 6 are machined if macro number 01 is called by the multi-piece machining command. (Tool change counts: $4 \times 6=24$)

Method 2

In this method, each set of machining commands to punch with a single tool is stored as a single macro.

```
U01 ;
  . . . .   T31 ;
  . . . . . . . . .
  . . . . . . . . .
V01 ;
U02 ;
  . . . .   T22 ;
  . . . . . . . . .
  . . . . . . . . .
V02 ;
U03 ;
  . . . .   T33 ;
  . . . . . . . . .
  . . . . . . . . .
V03 ;
U04 ;
  . . . .   T11 ;
  . . . . . . . . .
```

. . . .
V04 ;

If macro numbers 01, 02, 03 and 04 are called sequentially by the multi-piece machining command in this case, machining proceeds as follows.

1) Tool T31 performs full machining on all product parts.
2) Then, T22 performs full machining on all product parts.
3) Then, T33 performs full machining on all product parts.
4) Then, T11 performs full machining on all product parts.

Thus, its tool change is completed by only four steps so that tool change time is considerably decreased compared with method 1.

CAUTION

1 The G98 coordinate system is canceled by the G92 command.
2 If the parameter CLR (No. 3402#6) is set, the G98 coordinate system is canceled by a reset.
3 When the UV macro contains the G91 command, the tool moves according to the incremental value from the current position rather than the incremental position based on the G98 coordinate system.

14.5.2 Multi–Piece Machining Commands (G73, G74)

Specify any of the following commands, and then multi-piece machining is performed by calling machining commands stored using the macro function.

G73 W ωn Q q ; or
G74 W ωn Q q ;

where

ωn: A macro number

q: Machining start area specification

- q=1 Machining starts from the lower left area (1 in Fig. 14.5.1 (a))
- q=2 Machining starts from the lower right area (7 in Fig. 14.5.1 (a))
- q=3 Machining starts from the upper left area (3 in Fig. 14.5.1 (a))
- q=4 Machining starts from the upper right area (9 in Fig. 14.5.1 (a))

G73 goes on punching in the X axial direction, whereas G74 goes on punching in the Y axial direction in grid parts-line.
In the example shown in Figure 14.5.1 (b) to start machining from area 1 and to punch in the X axial direction, specify the following commands for the respective methods:

Method 1: G73 W01 Q1;

Method 2: G73 W01 Q1;
G73 W02 Q1;
G73 W03 Q1;
G73 W04 Q1;

Let the number of tool required to machine products for one sheet be n, and method 2 requires n times of commands G73/G74 to be specified. In such a case, several macro numbers can be represented by a single macro number if machining start area (specified by Q) and machining direction (G73/G74) are both the same with respect to the corresponding tools. Macro numbers to be provided for this purpose are 90 to 99. (These are called multiple macro storage numbers hereafter.)
To store macro numbers 01, 02 and 04 as a single macro number 90 in the example shown in Fig. 14.5.1 (b), specify as follows.

```
U90 ;
U01 ;
  . . . .   T31 ;
  . . . . . . . . .
V01 ;
U02 ;
  . . . .   T22 ;
  . . . . . . . . .
V02 ;
U04 ;
  . . . .   T11 ;
  . . . . . . . . .
V04 ;
V90 ;
```

In this case, the command of “G73 W90 Q1” becomes identical with the following series of commands.

```
G73 W01 Q1 ;
G73 W02 Q1 ;
G73 W04 Q1 ;
```

Though multi-piece machining commands (G73/G74) are generally used to punch products of the same shape arranged at intervals of grid, they can be also performed to punch exclusively in the X or Y axial direction.
In such a case, specify the number of products in the X or Y axial direction (to be specified by address word P or K) in G98 to “0”.

14.5.3 Setting of Machining Method for Multi–Piece Machining

When products are machined using a NC tape for multi-piece machining, any desired machining method can be selected according to a set-up from MDI.
Input a setting value into the setting data number 16206 in MDI mode according to the desired machining method.

Setting value for No. 16206

0: The NC tape for multi-piece machining is not used

1: Trial punching for multi-piece machining

2: Machining on the reset of material punched for trial

3: Full machining on a material for multi-piece machining

If trial punching is selected, only the lower-left product part of material (1 in Fig. 14.5.1 (a) and (b)). As a result, macros except 60 to 89 are executed while storing, and blocks specified in G73/G74 are all ignored. Machining on the reset of material punched for trial signifies that after punching on only the lower-left product part of material, the remaining product parts of material are machined entirely. For this purpose, no machining is performed during the storage of macros, and the machining on the lower-left product part is skipped when commands of G73/G74 are executed.
For full machining on a material, no machining is performed during the storage of macros, but machining over the entire product parts is performed by G73/G74.
Set to “0” if NC tape for the multi-piece machining is not used.

14.5.4 Command for Restarting Machining Multiple Products

Specifying the position from which machining multiple products restarts with address P in a block in which the G73 or G74 command for machining multiple products is specified enables machining multiple products to restart from the specified position.

Command format:

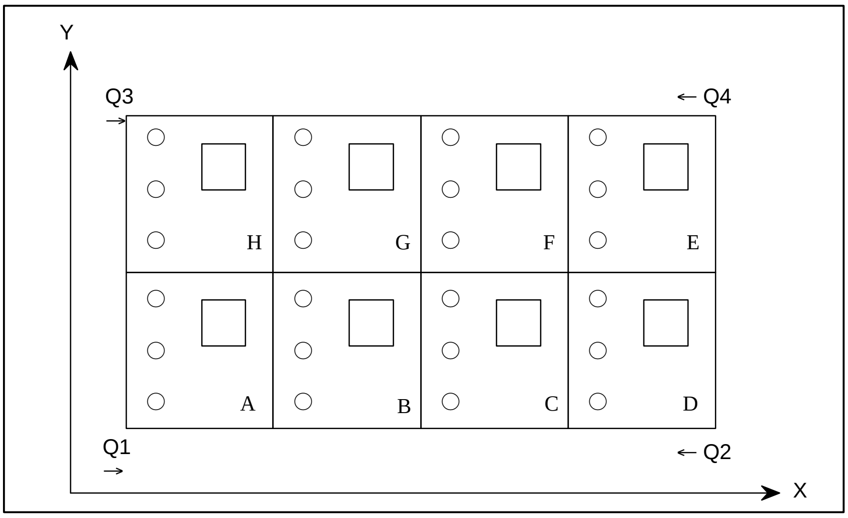
G73 (G74) W_w_ Q_q_ P_p_ ;

w : Macro number

q : Machining sequence

p : Position from which machining restarts

A P code specifies the position from which machining multiple products restarts in a block where a G73 or G74 command is specified and a Q code is used to specify a machining area.
Valid setting: 1≦p≦total number of products to be machined



Examples

G98X–Y–I–J–P3K1 ;

U90 ;

|

V90 ;

G75W90Q–P5 ;

In the above program, products are machined in the order specified by the following Q command.

(1) When machining restarts

(a) Q1 command: Products are machined in the order of E, F, G, and H.

(b) Q2 command: Products are machined in the order of H, G, F, and E.

(c) Q3 command: Products are machined in the order of D, C, and B.

(d) Q4 command: Products are machined in the order of B, C, and D.

(2) When machining starts

(a) Q1 command: Products are machined in the order of E, F, G, and H.

(b) Q2 command: Products are machined in the order of H, G, F, and E.

(c) Q3 command: Products are machined in the order of D, C, B, and A.

(d) Q4 command: Products are machined in the order of A, B, C, and D.

WARNING

When restarting machining multiple products is specified, the reference position specified by the G98 command for setting the reference position used in machining multiple products must be used. Do not change the reference position.

NOTE

1 A restart command is invalid for trial machining.

2 When machining multiple products is restarted, specify the position from which machining restarts using address P so that the number of products includes the number of products to be used for the trial, though they are not machined.

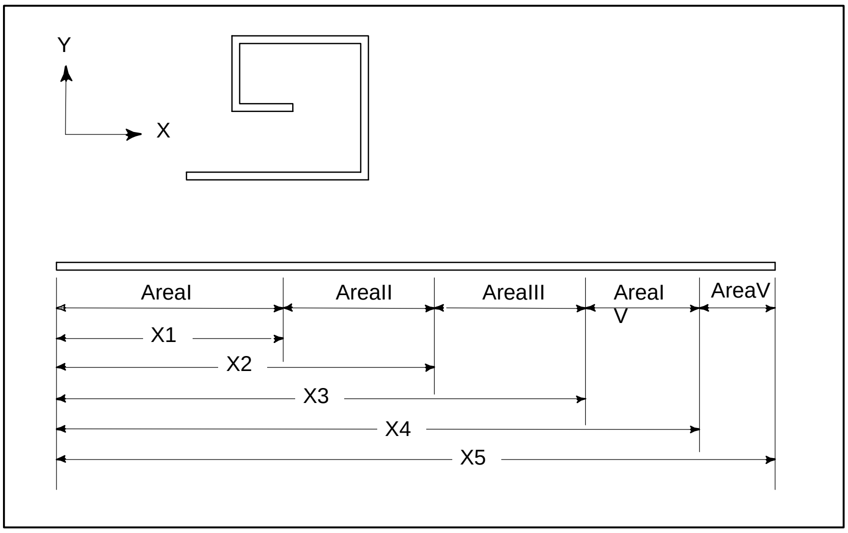
3 If a setting specified with address P is not within the range from 1 to the total number of products inclusive, alarm No. 4544 is issued.

14.6 BENDING COMPENSATION (G38, G39)

The hole position gap accompanied bending is compensated and the drilling is performed.

Program format

- Bending compensation for X axis direction
  G38I X1 J X2 K X3 P X4 Q X5 R α ;
- Bending compensation for Y axis direction
  G39I Y1 J Y2 K Y3 P Y4 Q Y5 R β ;



α, β : These are called the bending coefficient. The compensation amount attendant on a single bending is specified.
Unit: input unit

X1 to X5 : The distance of X and Y axes direction from the standard point is specified.

(Y1 to Y5) : Bending can be performed up to 4 times.

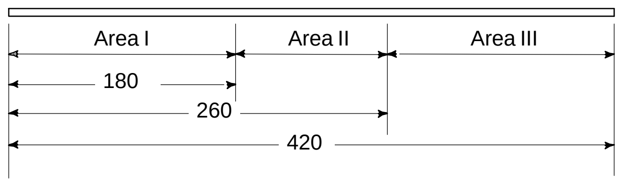
Reference position is specified at local coordinate system.
G52X_ Y_ ;

In a block after a bending command is sent, it is judged to which area automatically designated the command value of the move command belongs, thus allowing compensation in accordance with the bending frequency.
When bending compensation cancel is required.

1) Specify the cancel of bending compensation.
   G38R0 ; ⋯ (Cancel of X axis direction)
   G39R0 ; ⋯ (Cancel of Y axis direction)
2) Specify M02, M30.
   It is possible to cancel even in the reset state according to reset, clear, and emergency stop.

Examples

When the bending compensation of only X axis direction is performed.



Program

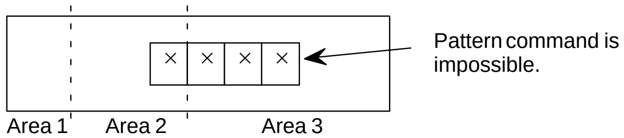
:

G52X100.Y0 ;	Specifications of standard point
G38I180.J260.K420.R-1. ;	Bending compensation command
G90X-50. ;	Absolute coordinate (X50.)
X270. ;	Absolute coordinate (X269.)
X300. ;	Absolute coordinate (X298.)
M02 ;	Cancel of bending compensation execution

In the above example, when the negative direction from the standard point, for example, X50. direction (X-50. since the coordinates system is offset in the program in the coordinates form the origin) is designated, bending compensation is not performed. At this time, it is regarded as area 1, that is, an area with no compensation by bending. As described above, when the number of bending is less than 4, it is possible to omit what is not required among X1-X5 and Y1-Y5 representing the distance from the standard point which indicates the area boundary.

WARNING

In one block to which a pattern command, for example, is given during bending compensation, it is regarded as a move within the area, and no command over the area can be given.



15 COMPENSATION FUNCTION

This chapter describes the following compensation functions:

15.1 OVERVIEW OF CUTTER COMPENSATION C (G40 TO G42)

When the tool is moved, the tool path can be shifted by the radius of the tool (Fig. 15.1 (a)).
To make an offset as large as the radius of the tool, CNC first creates an offset vector with a length equal to the radius of the tool (start–up). The offset vector is perpendicular to the tool path. The tail of the vector is on the workpiece side and the head positions to the center of the tool.
If a linear interpolation or circular interpolation command is specified after start–up, the tool path can be shifted by the length of the offset vector during machining.
To return the tool to the start position at the end of machining, cancel the cutter compensation mode.

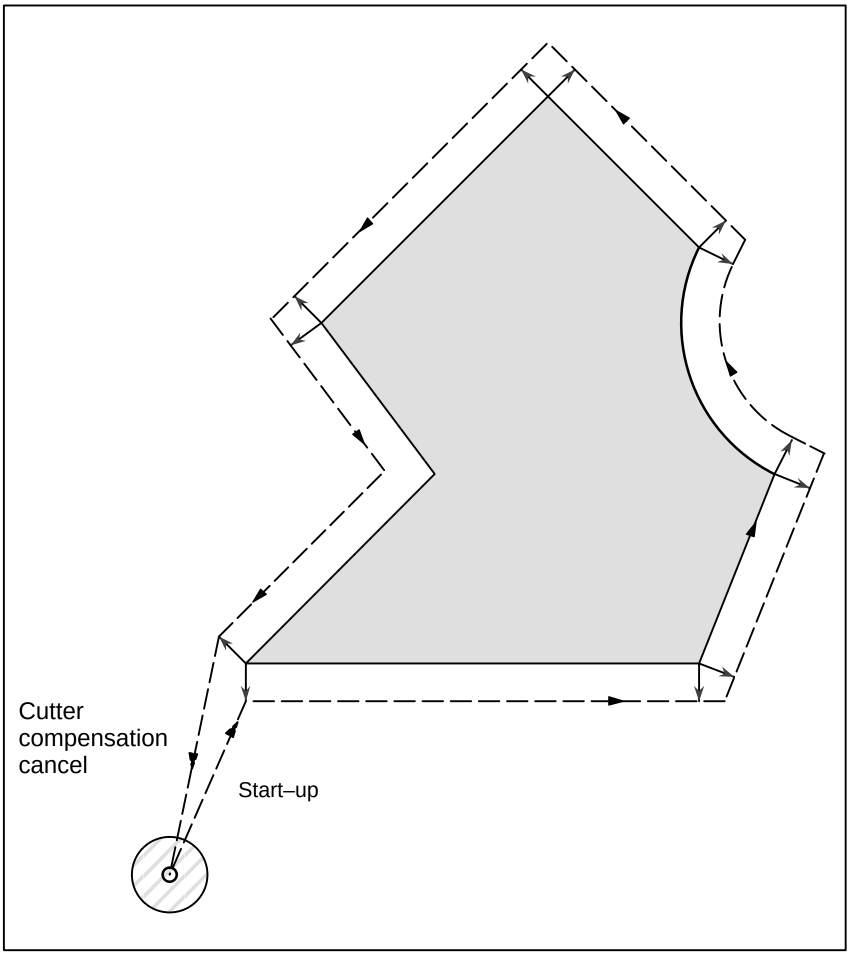


Fig. 15.1 (a) Outline of Cutter Compensation C

Format

- **Start up (Tool compensation start)**

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

G41 : Cutter compensation left (Group07)
G42 : Cutter compensation right (Group07)
IP_ : Command for axis movement
D_ : Code for specifying as the cutter compensation value (1 to 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 tool center path coincides with the programmed path.

- **Start Up**

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

- **Offset mode**

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

- **Offset mode cancel**

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

1. G40 has been commanded.

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

When performing offset cancel, circular arc commands (G02 and G03) are not available. If a circular arc is commanded, an alarm (No. 034) is generated and the tool stops.

In the offset cancel, the control executes the instructions in that block and the block in the cutter compensation buffer. In the meantime, in the case of a single block mode, after reading one block, the control executes it and stops. By pushing the cycle start button once more, one block is executed without reading the next block.

Then the control is in the cancel mode, and normally, the block to be executed next will be stored in the buffer register and the next block is not read into the buffer for cutter compensation.

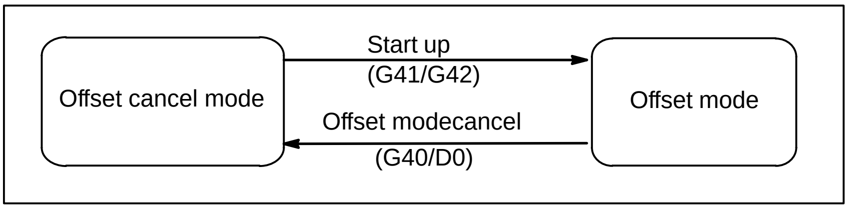


Fig. 15.1 (b) Changing the offset mode

- **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.

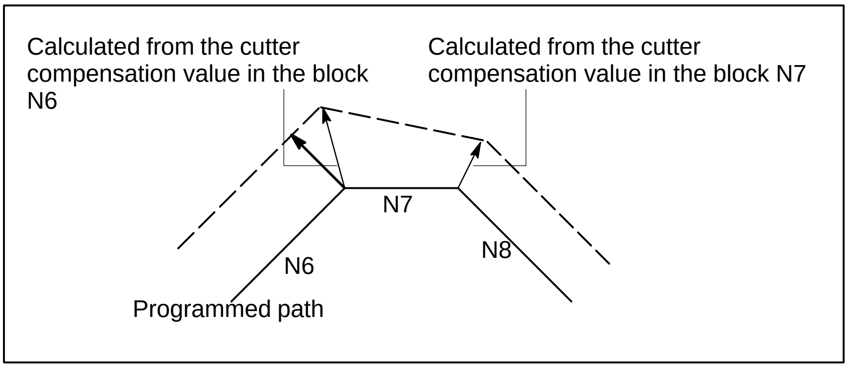


Fig. 15.1 (c) Changing the Cutter Compensation Value

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

If the offset amount is negative (–), distribution is made for a figure in which G41's and G42's are all replaced with each other on the program. Consequently, if the tool center is passing around the outside of the workpiece, it will pass around the inside, and vice versa.
The figure below shows one example. Generally, the offset amount is programmed to be positive (+).
When a tool path is programmed as in ((1)), if the offset amount is made negative (–), the tool center moves as in ((2)), and vice versa. Consequently, the same tape permits cutting both male and female shapes, and any gap between them can be adjusted by the selection of the offset amount. Applicable if start–up and cancel is A type. (See subsec. 15.2.2 and 15.2.4)

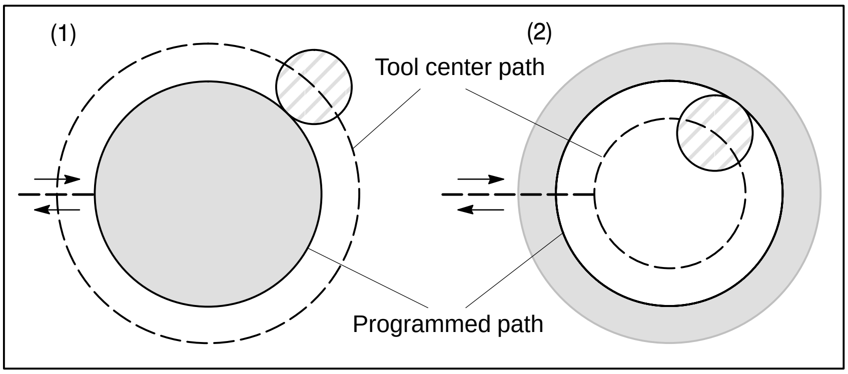


Fig. 15.1 (d) Tool Center Paths when Positive and Negative Cutter Compensation Values are Specified

- **Cutter compensation value setting**

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

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

> **NOTE**
> 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.

- **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.

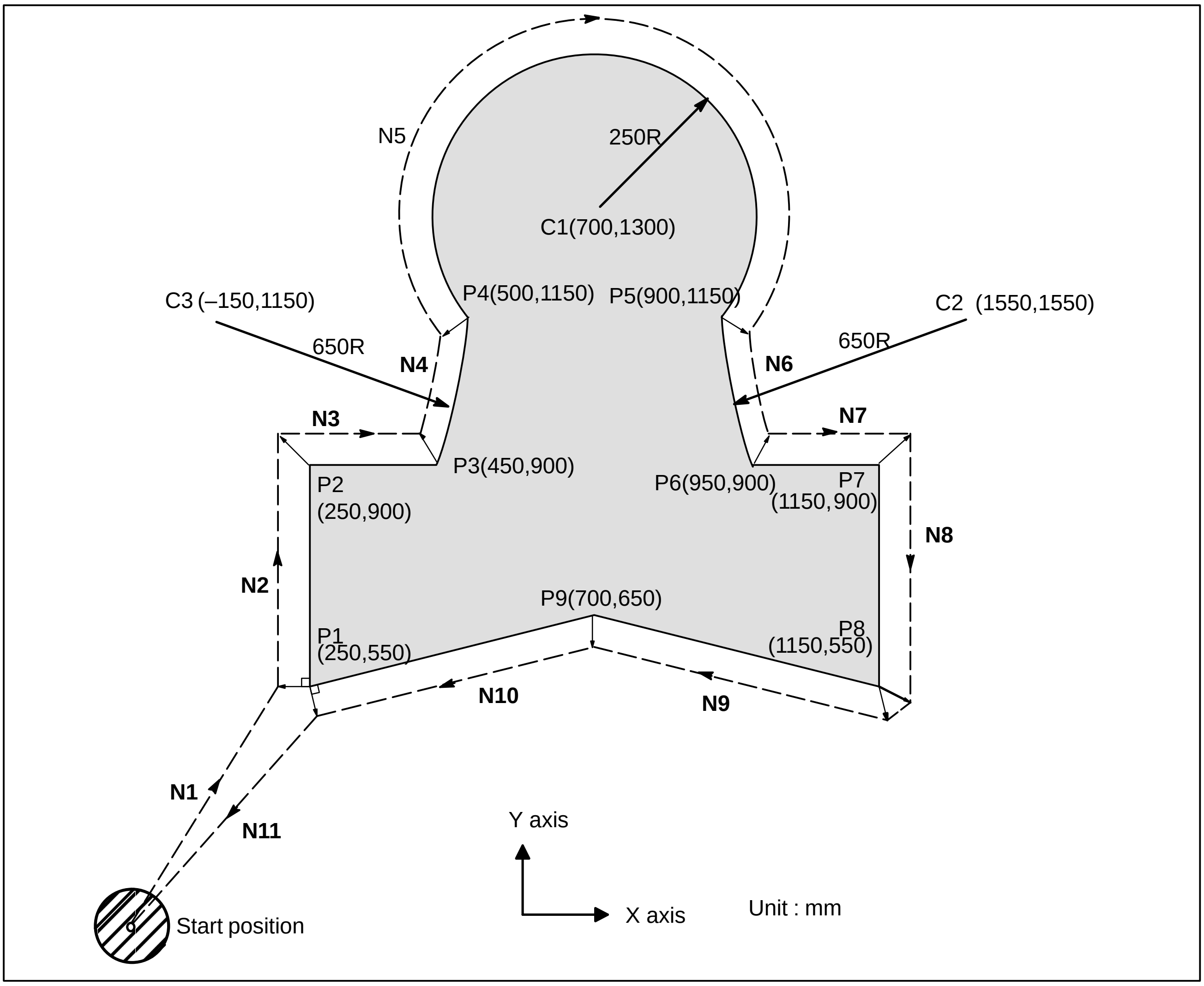
- **Specifying a cutter compensation value**

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 tool path projected on the offset plane is compensated.
The offset plane is changed during the offset cancel mode. If it is performed during the offset mode, an alarm (No. 37) is displayed and the machine is stopped.

Examples



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

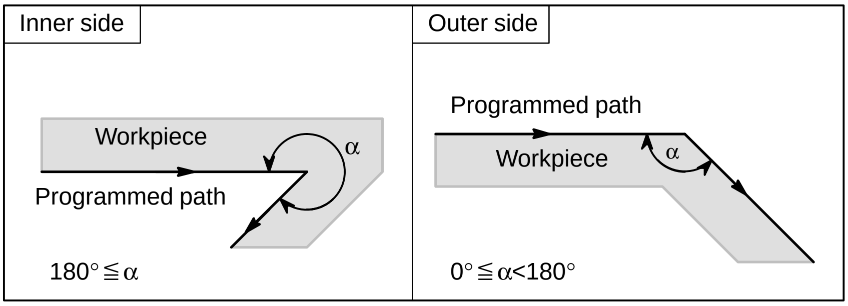
15.2 DETAILS OF CUTTER COMPENSATION C

This section provides a detailed explanation of the movement of the tool for cutter compensation C outlined in Section 15.1.
This section consists of the following subsections:

15.2.1 General

- **Inner side and outer side**

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



- **Meaning of symbols**

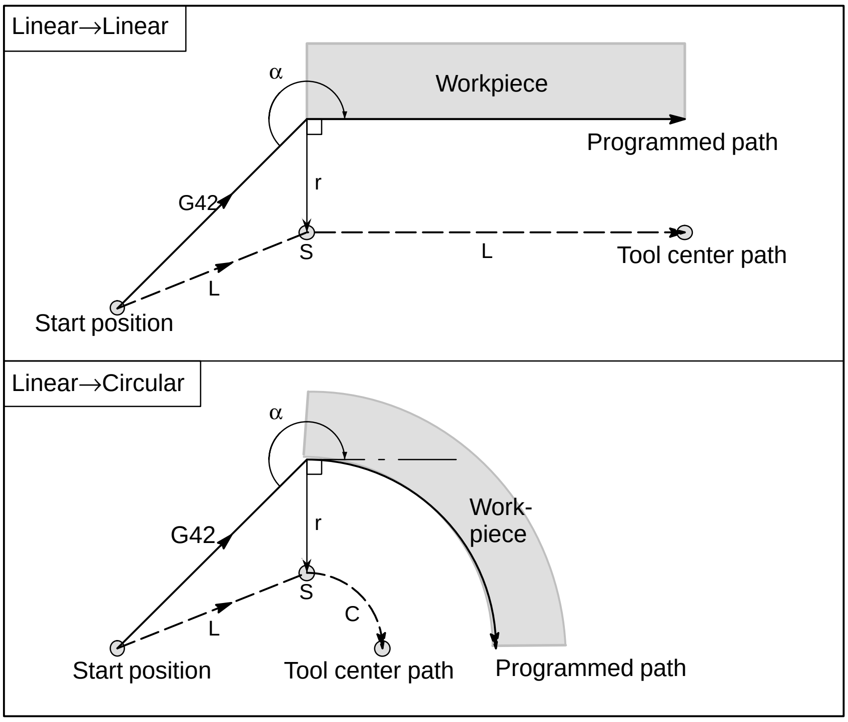
The following symbols are used in subsequent figures:
– ***S*** indicates a position at which a single block is executed once.
– ***SS*** indicates a position at which a single block is executed twice.
– ***SSS*** indicates a position at which a single block is executed three times.
– ***L*** indicates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

15.2.2 Tool Movement in Start–up

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

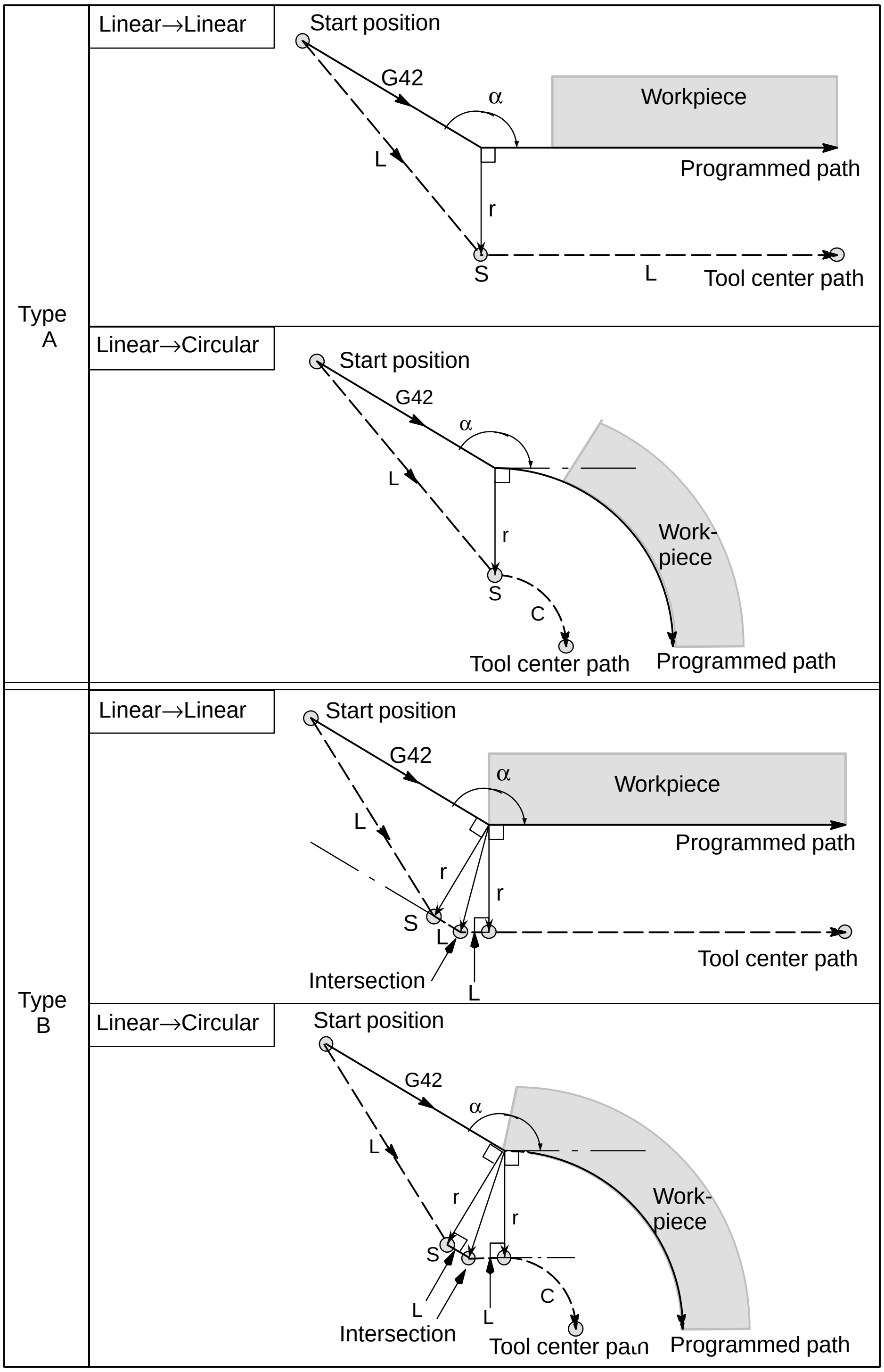
Explanations

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



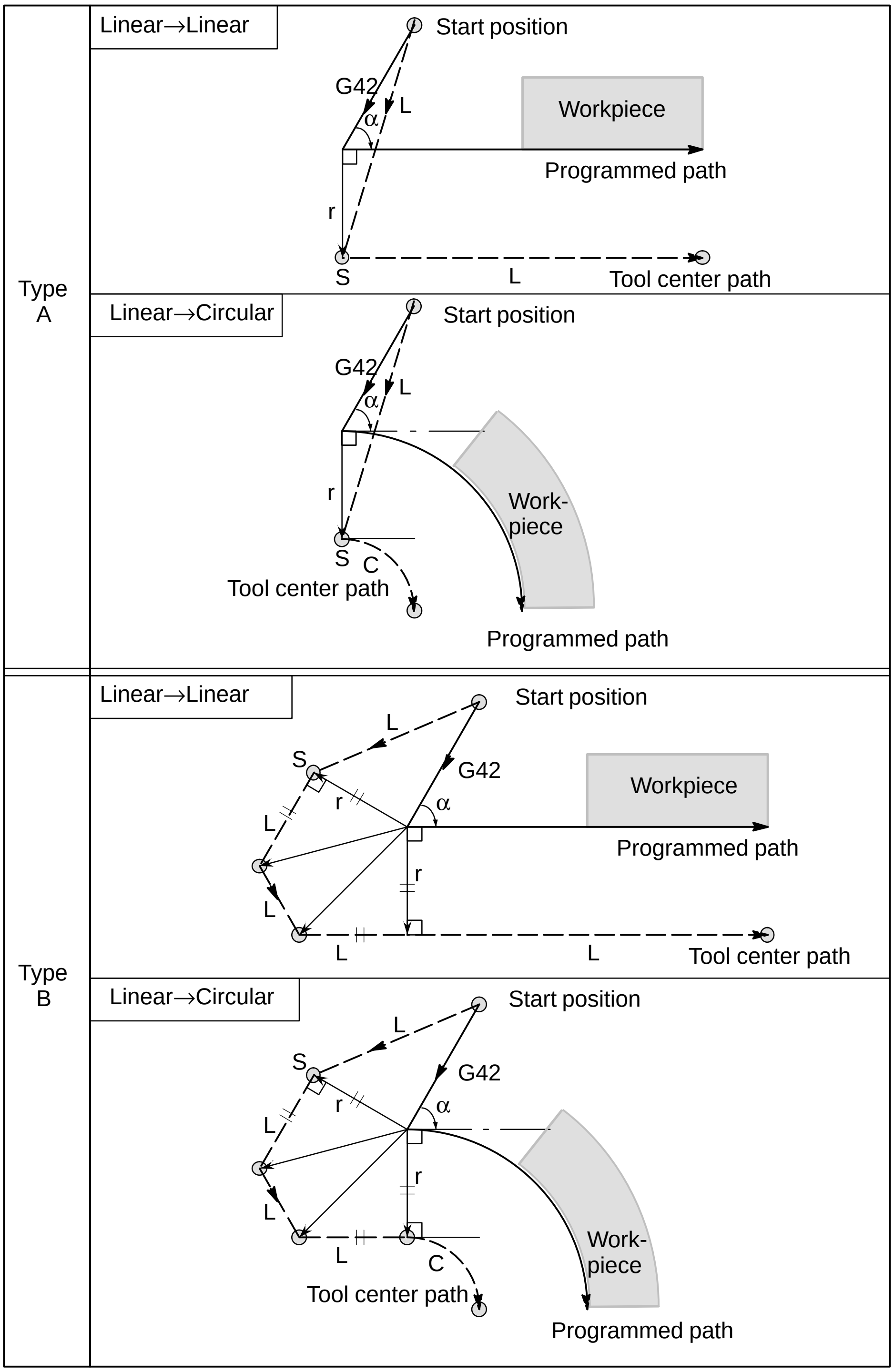
- **Tool movement around the outside of a corner at an obtuse angle (90°≦α<180°)**

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

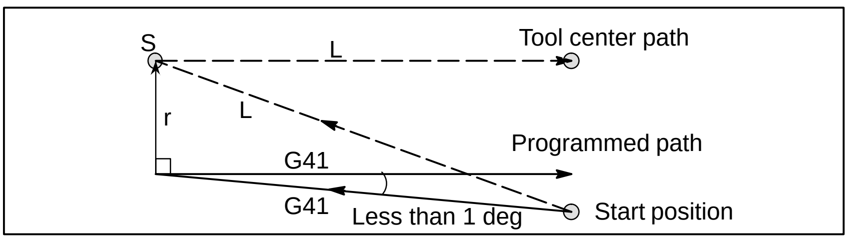


- **Tool movement around the outside of an acute angle (α<90°)**

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



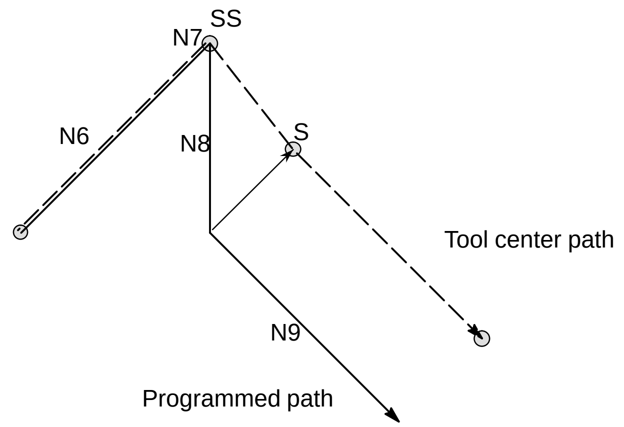
- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



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

If the command is specified at start–up, the offset vector is not created.

```
G91 G40 … ;
      :
N6 X1000.0 Y1000.0 ;
N7 G41 X0 ;
N8 Y–1000.0 ;
N9 Y–1000.0 X1000.0 ;
```



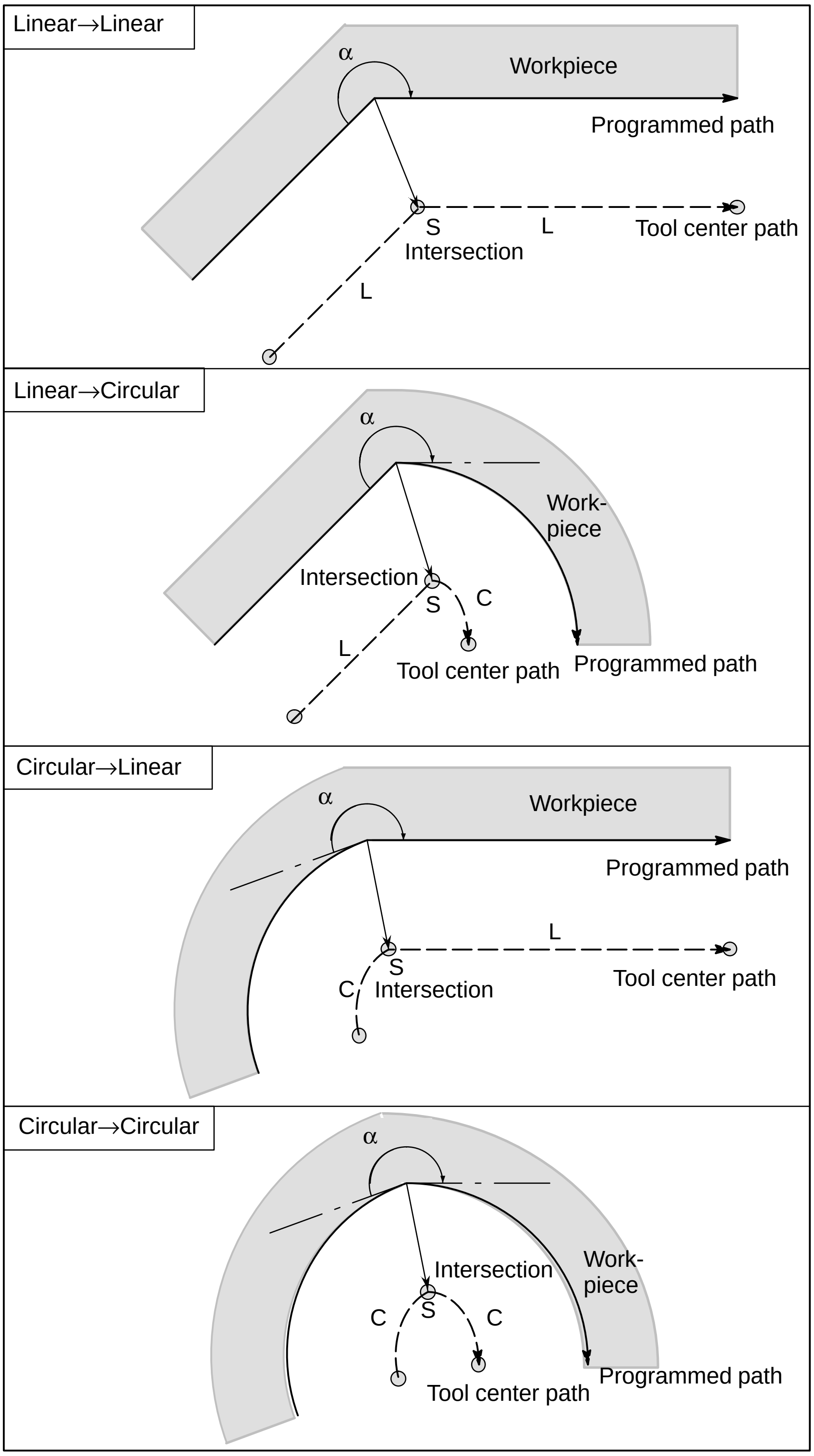
NOTE
For the definition of blocks that do not move the tool, see Subsection 15.2.3.

15.2.3 Tool Movement in Offset Mode

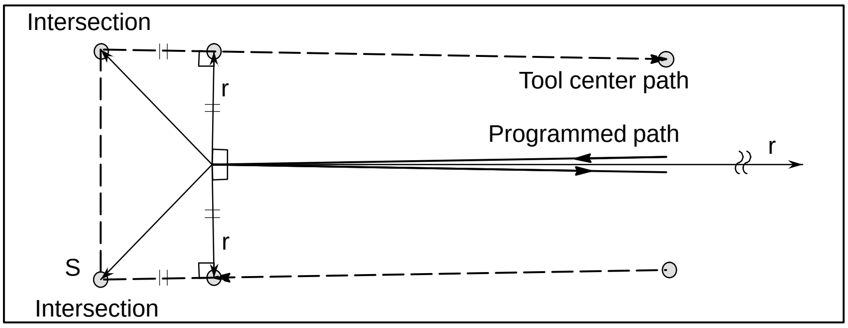
In the offset mode, the tool moves as illustrated below:

Explanations

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

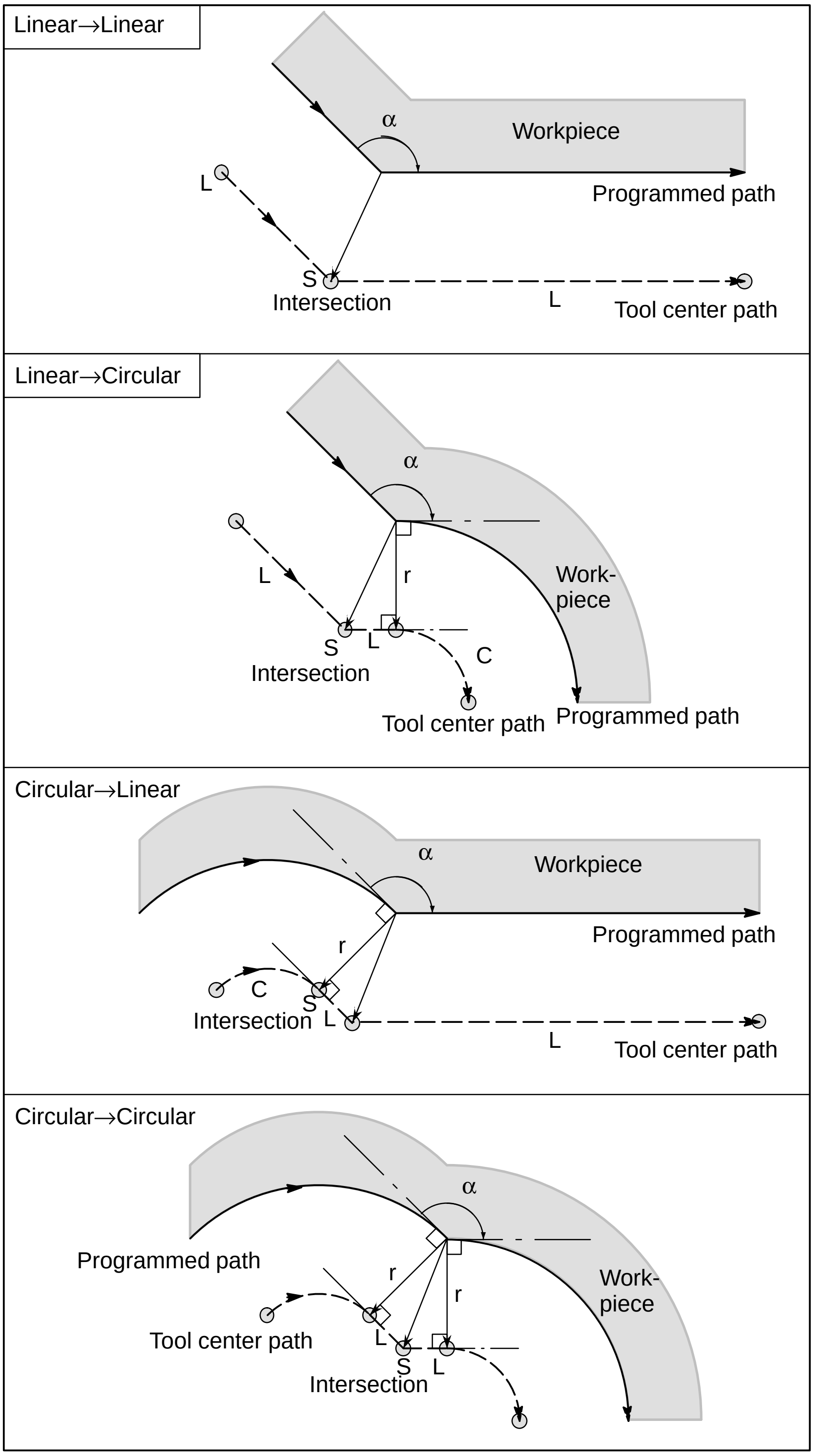


- **Tool movement around the inside (α<1°) with an abnormally long vector, linear → linear**

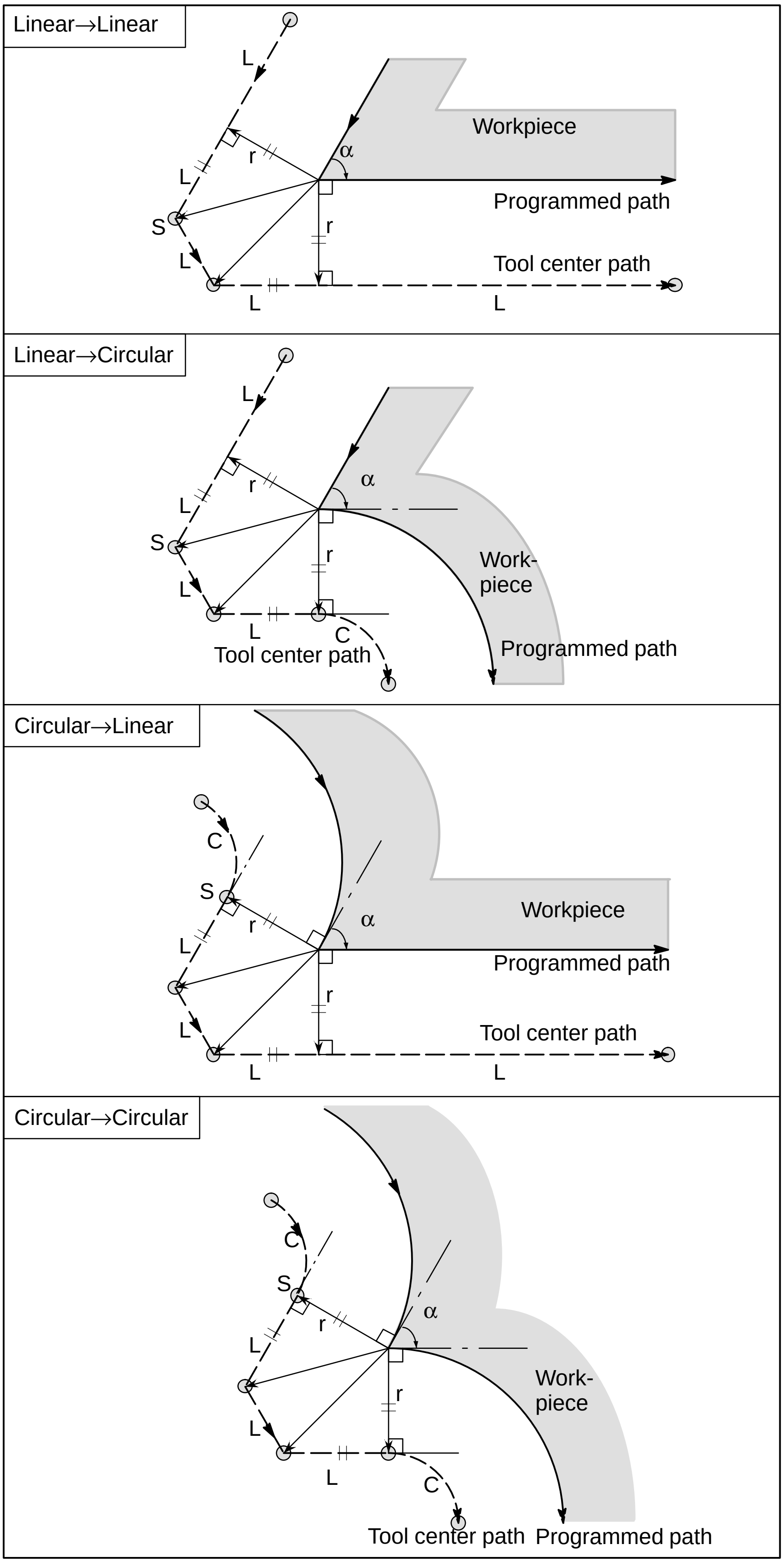


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

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



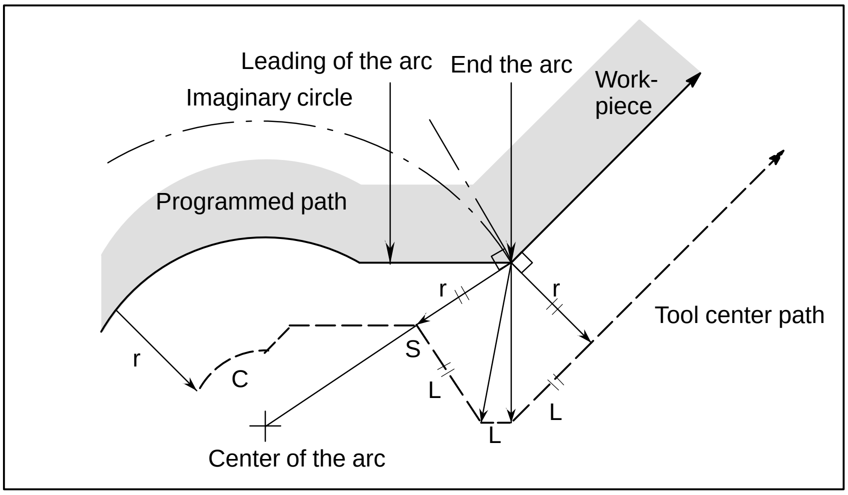
- **Tool movement around the outside corner at an acute angle (α<90°)**



- **When it is exceptional**

End position for the arc is not on the arc

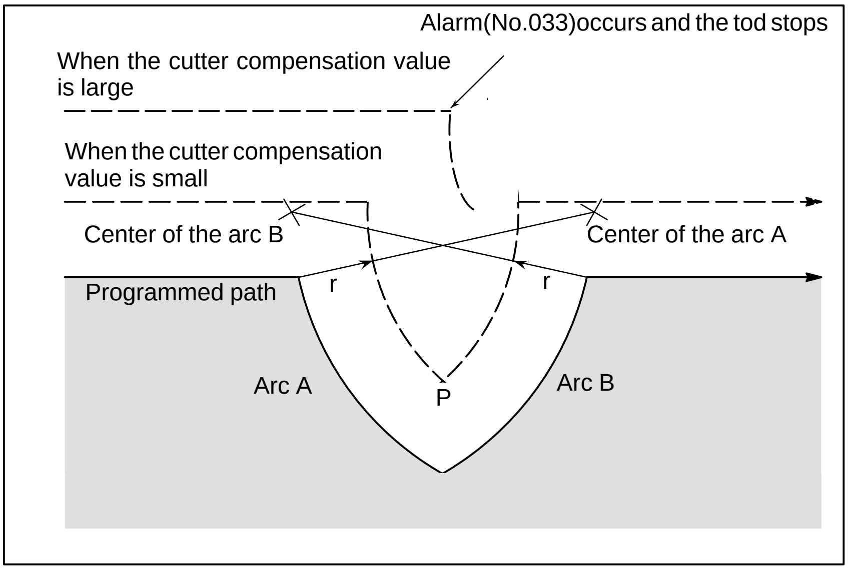
If the end of a line leading to an arc is programmed as the end of the arc by mistake as illustrated below, the system assumes that cutter compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption,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



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

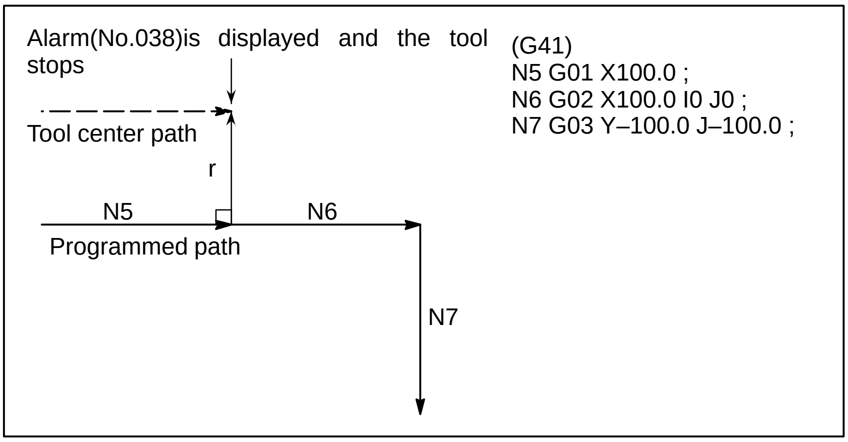
There is no inner intersection

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



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

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



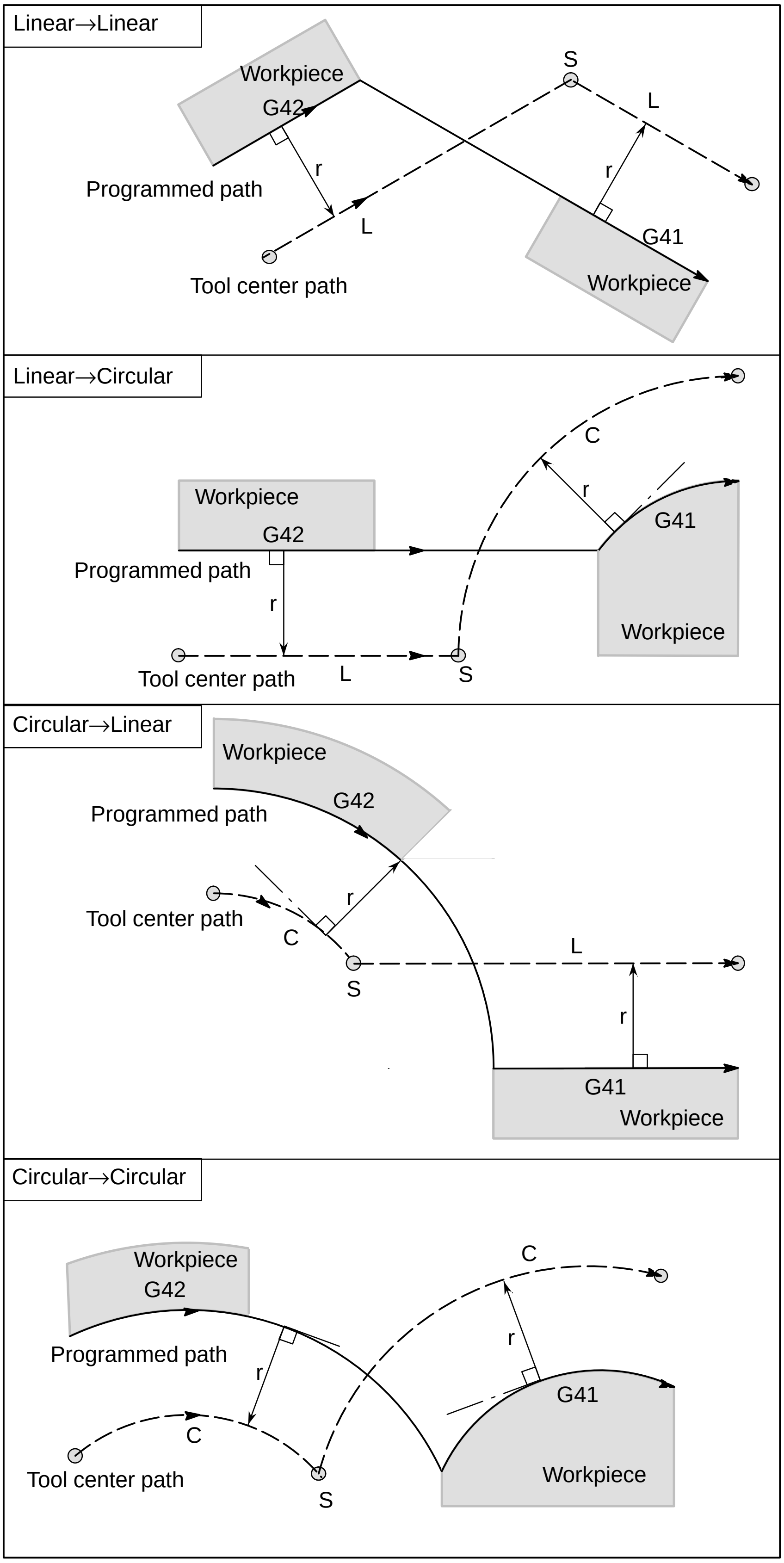
- **Change in the offset direction in the offset mode**

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

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

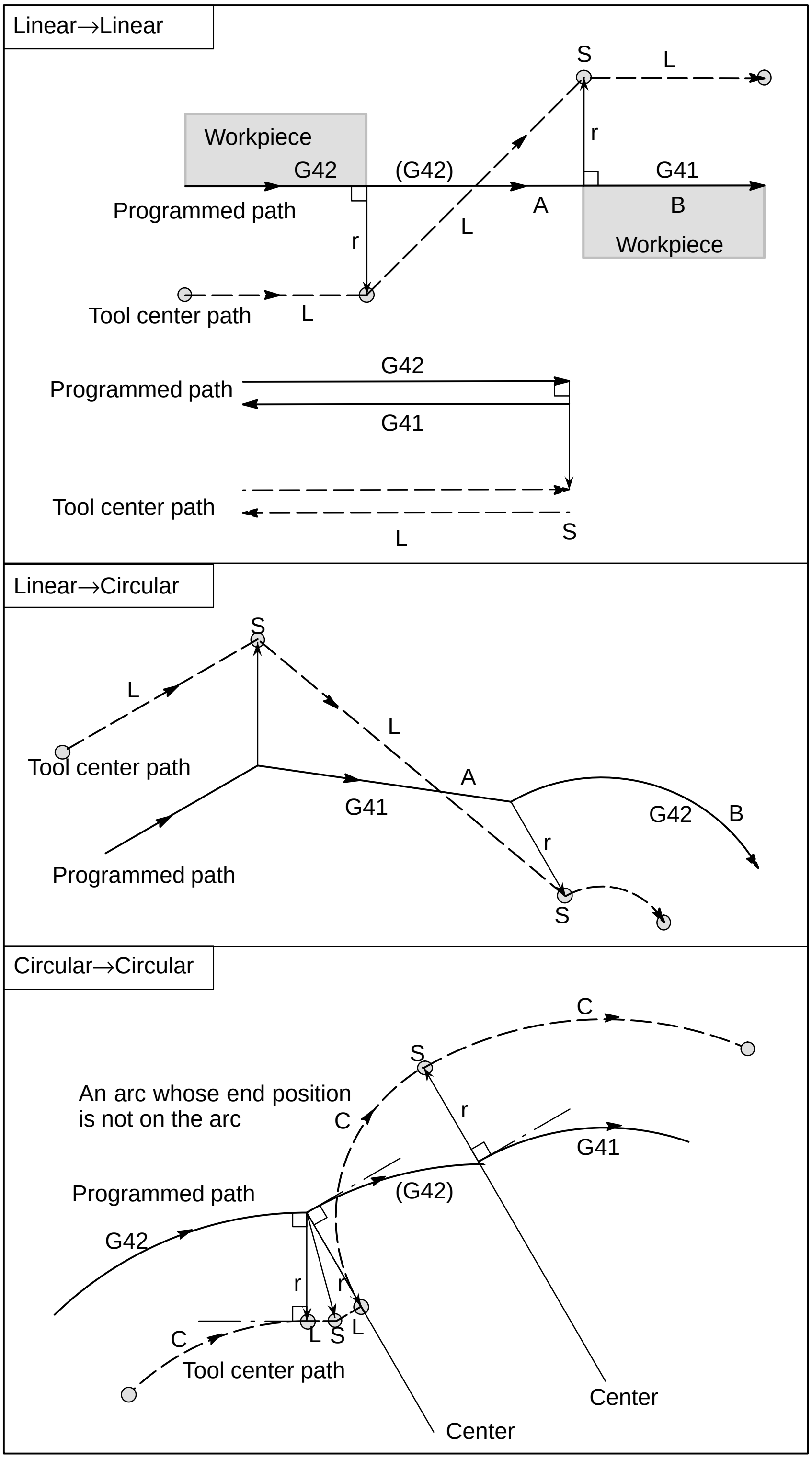
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool center path of that block and the tool center path of a preceding block. However, the change is not available in the start–up block and the block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

Tool center path with an intersection



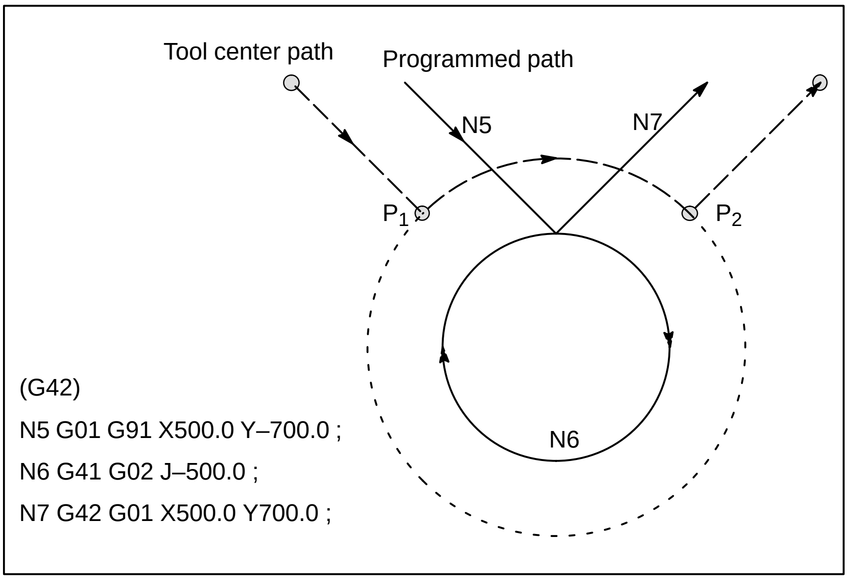
Tool center path without an intersection

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



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

Normally there is almost no possibility of generating this situation. However, when G41 and G42 are changed, or when a G40 was commanded with address I, J, and K this situation can occur.
In this case of the figure, the cutter compensation is not performed with more than one circle circumference: an arc is formed from P_1 to P_2 as shown. Depending on the circumstances, an alarm may be displayed due to the ”Interference Check” described later. To execute a circle with more than one circumference, the circle must be specified in segments.

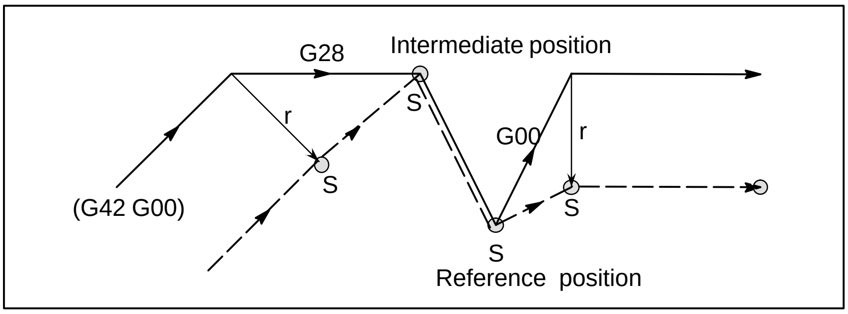


- **Temporary cutter compensation cancel**

If the following command is specified in the offset mode, the offset mode is temporarily canceled then automatically restored. The offset mode can be canceled and started as described in Subsections 15.2.2 and 15.2.4.

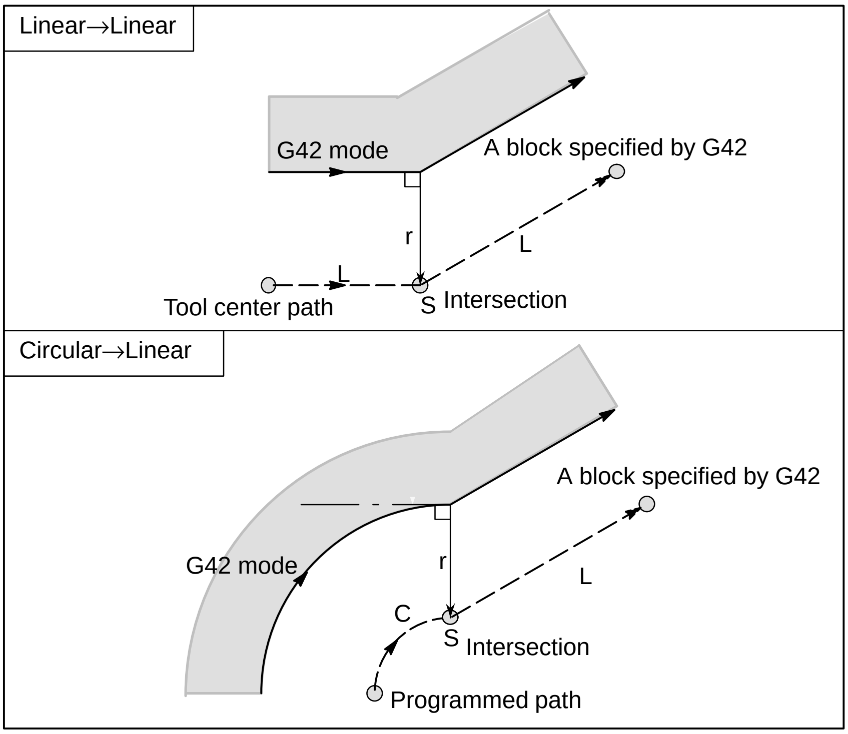
Specifying G28 (automatic return to the reference position) in the offset mode

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



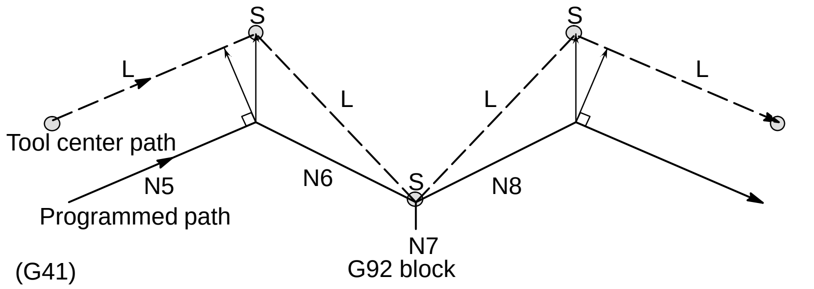
- **Cutter compensation G code in the offset mode**

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.15.2.3.



- **Command cancelling the offset vector temporarily**

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



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

- **A block without tool movement**

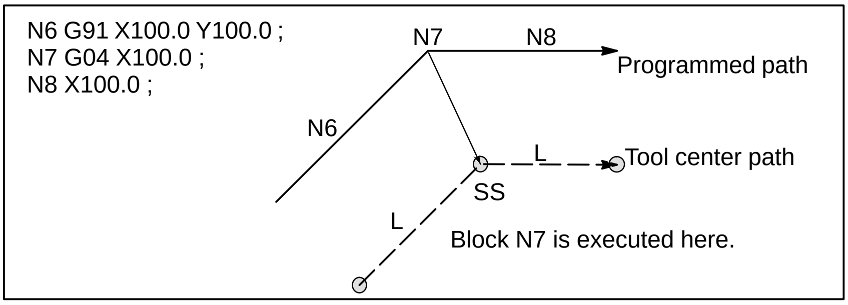
The following blocks have no tool movement. In these blocks, the tool will not move even if cutter compensation is effected.

M05 ;. . . . . M code output
S21 ; . . . . .S code output
G04 X100.0 ; . . . . . Dwell
G10 L11 P01 R10.0 ; Cutter compensation value setting
(G17) Z200.0 ; . . . . Move command not included in the offset plane.
G90 ; . . . . G code only
G91 X0 ; . . . . Move distance is zero.

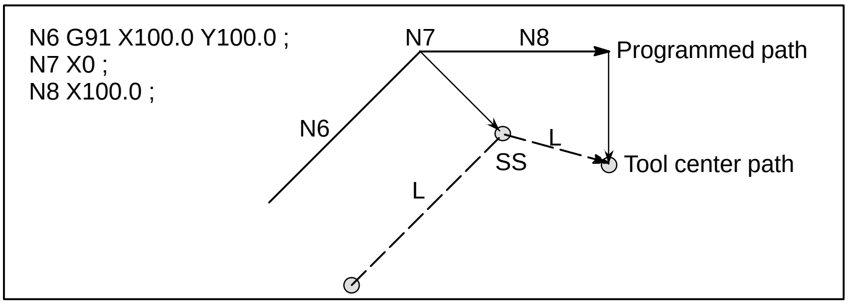
Commands (1) to (6) are of no movement.

A block without tool movement specified in offset mode

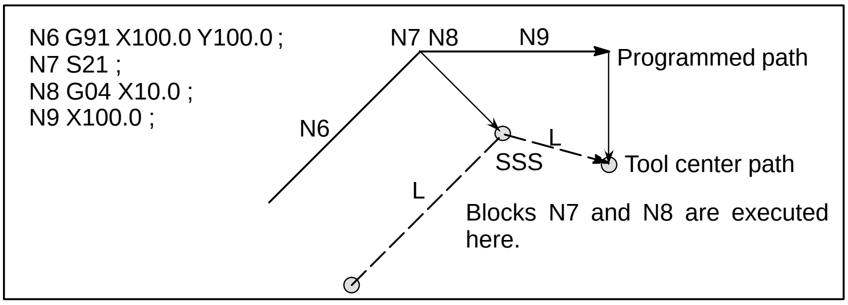
When a single block without tool movement is commanded in the offset mode, the vector and tool center path are the same as those when the block is not commanded. This block is executed at the single block stop point.



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

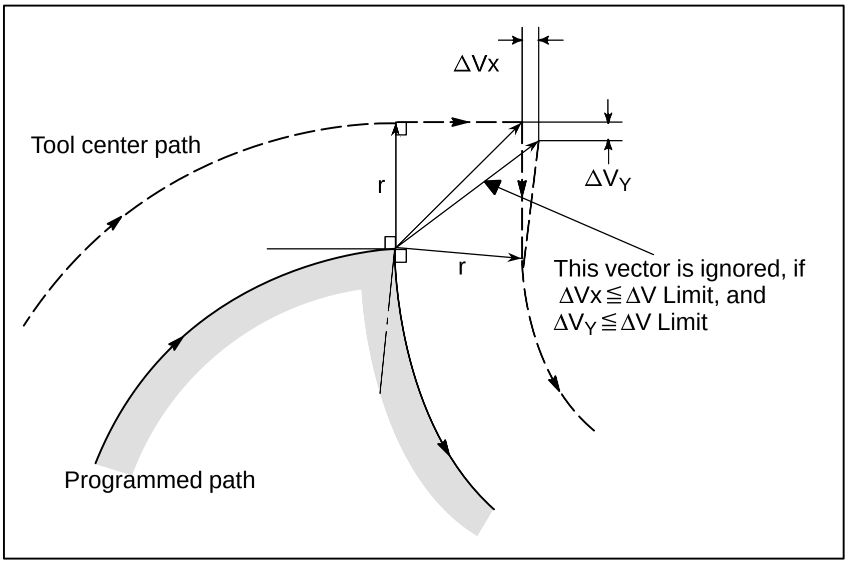


Two blocks without tool movement should not be commanded consecutively. If commanded, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in earlier block, so overcutting may result.

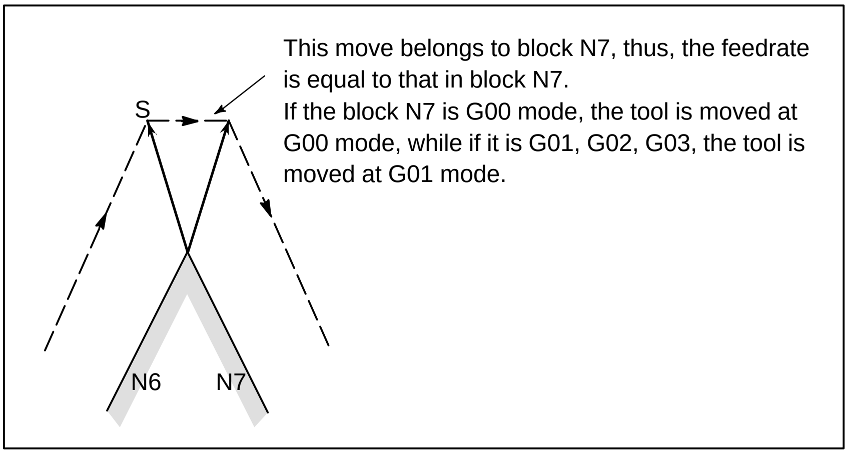


• Corner movement

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

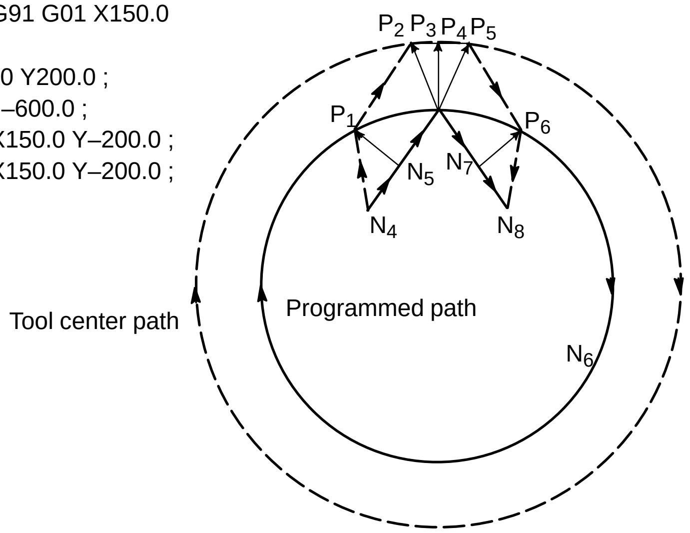


If $\Delta Vx \leqq \Delta V$limit and $\Delta Vy \leqq \Delta V$limit, the latter vector is ignored. The ΔVlimit is set in advance by parameter (No. 5010).
If these vectors do not coincide, a move is generated to turn around the corner. This move belongs to the latter block.



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

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



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

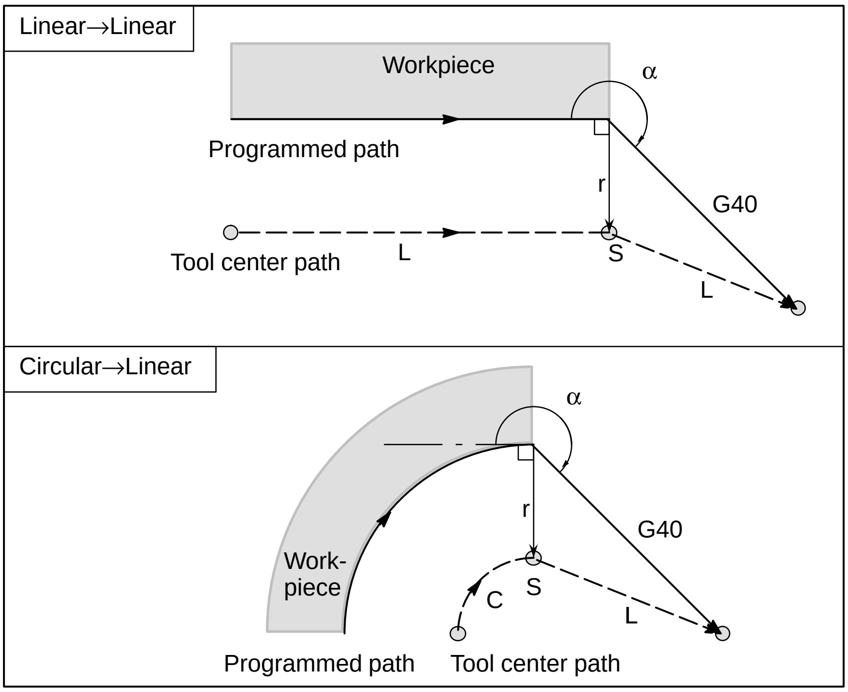
- **Interruption of manual operation**

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

15.2.4 Tool Movement in Offset Mode Cancel

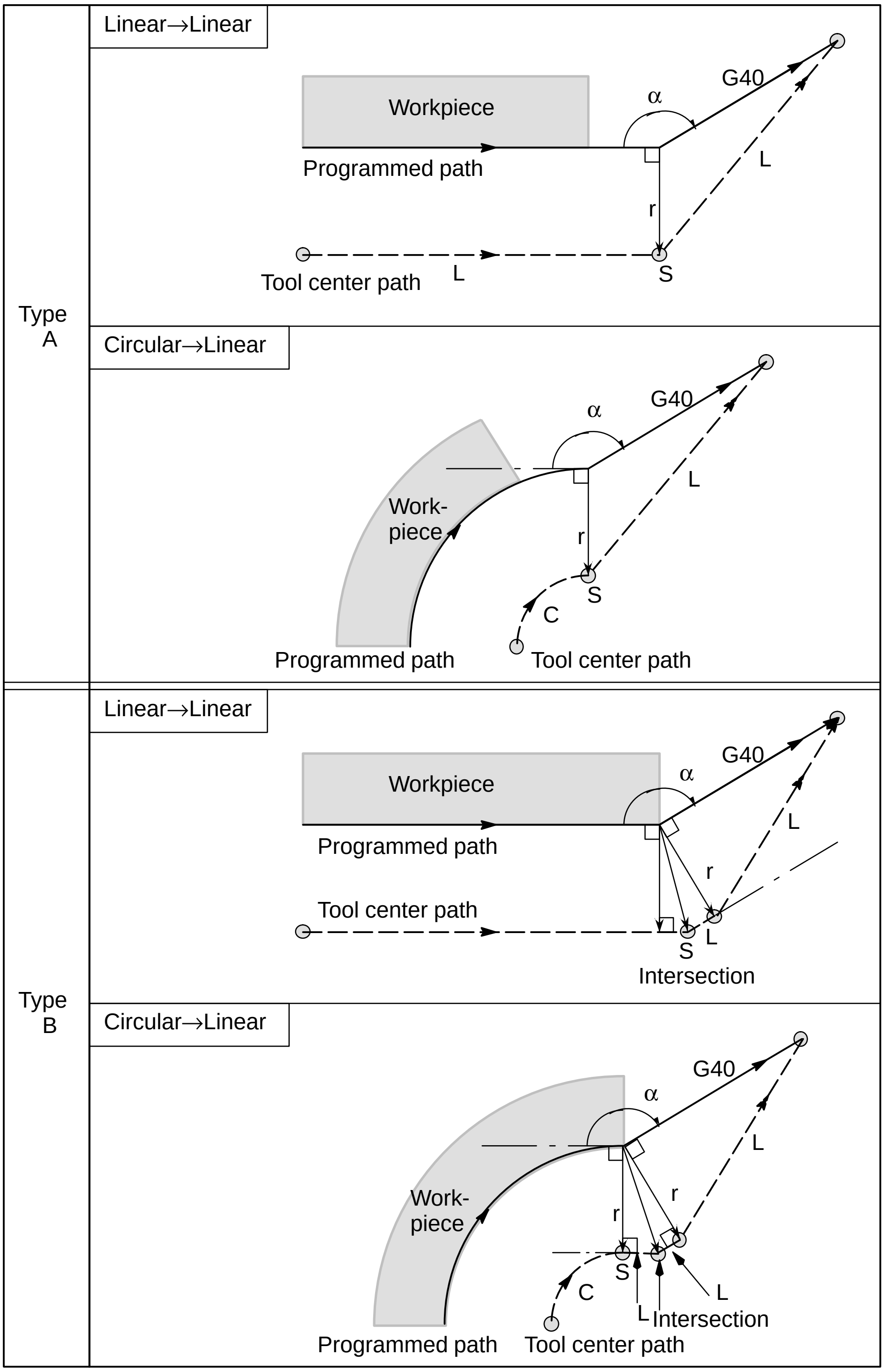
Explanations

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



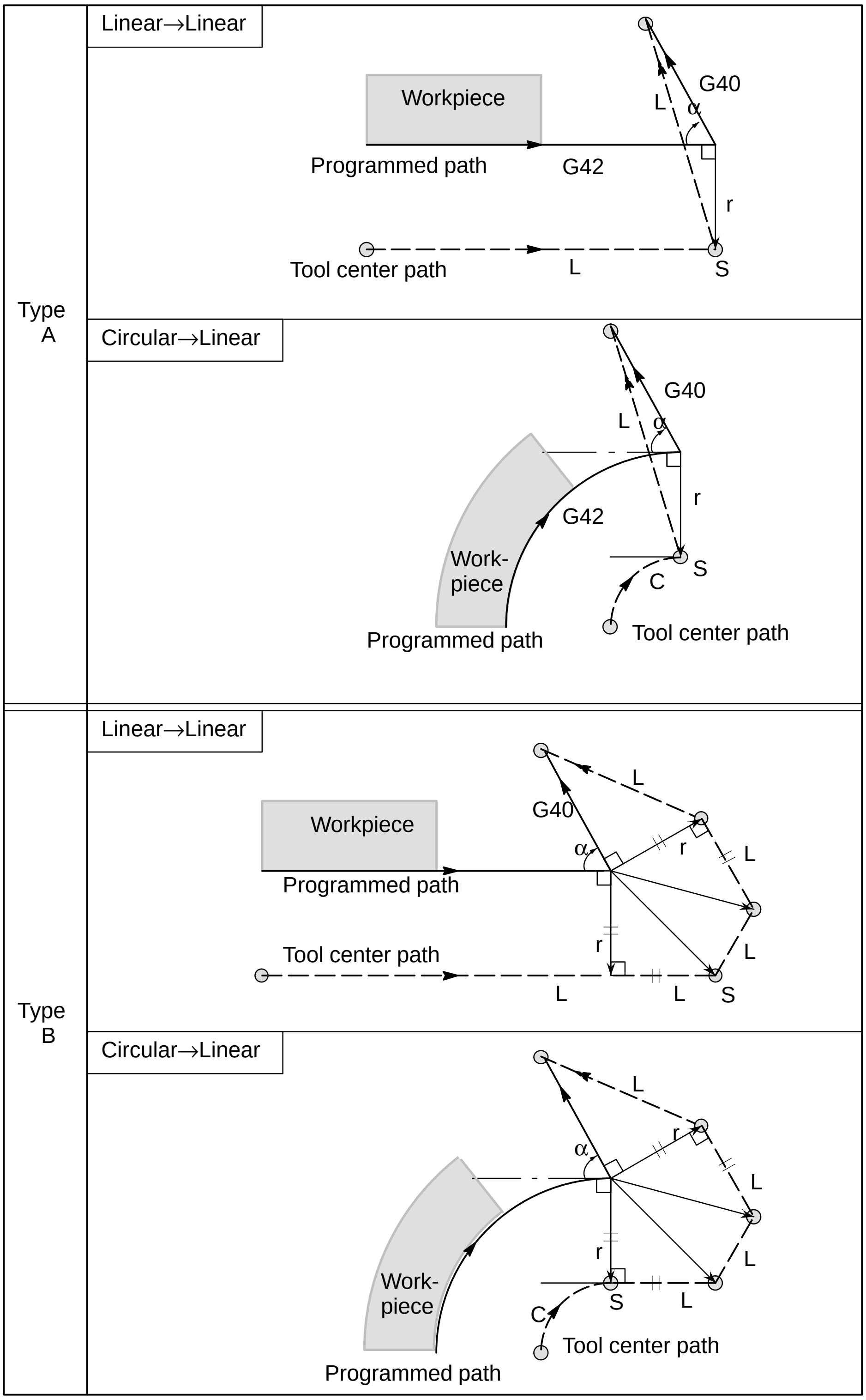
- **Tool movement around an outside corner at an obtuse angle (90°≦α<180°)**

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

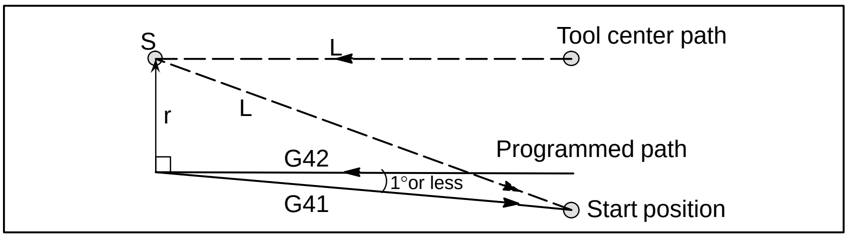


- **Tool movement around an outside corner at an acute angle (α<90°)**

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

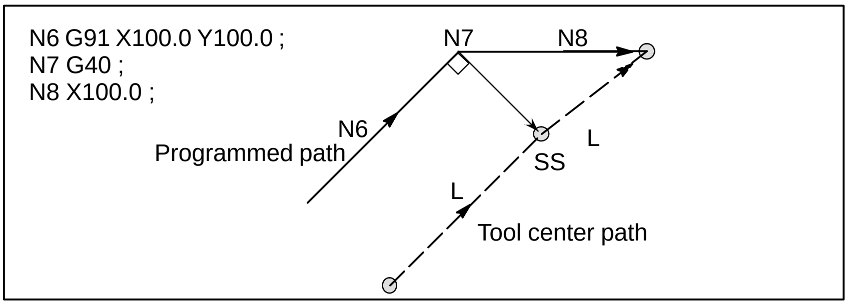


- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



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

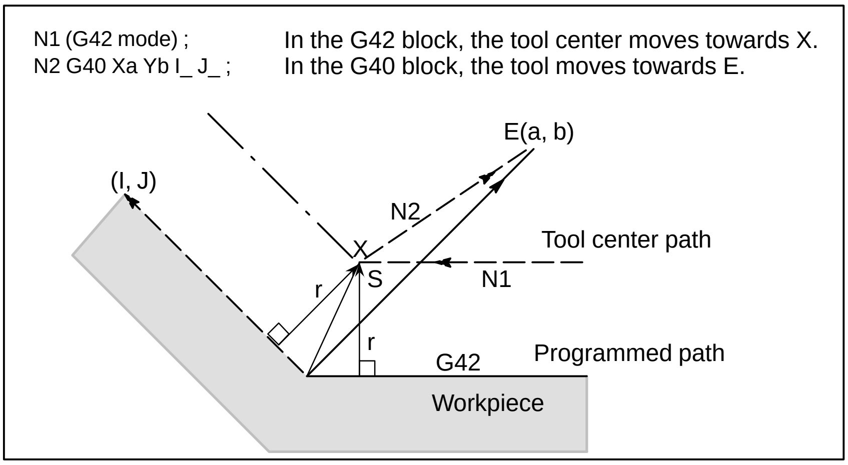
When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in the earlier block, the vector is cancelled in the next move command.



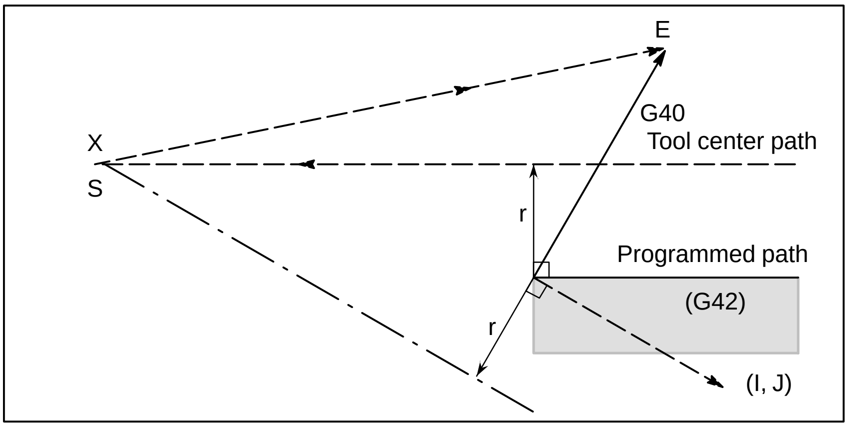
- **Block containing G40 and I_J_K_**

The previous block contains G41 or G42

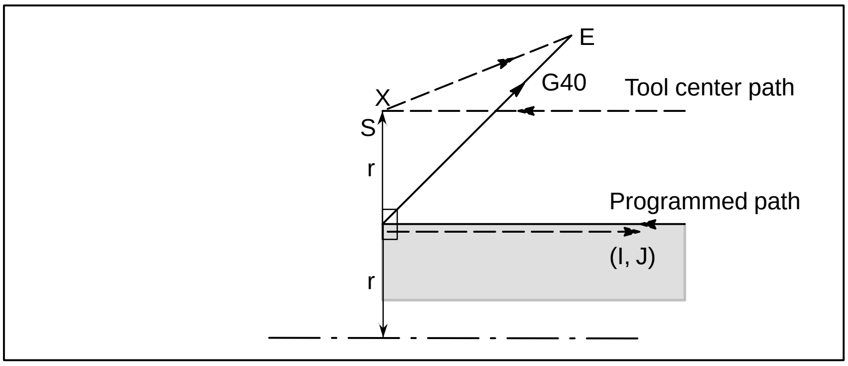
If a G41 or G42 block precedes a block in which G40 and I_, J_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I,J), (I,K), or (J,K). The direction of compensation in the former block is inherited.



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

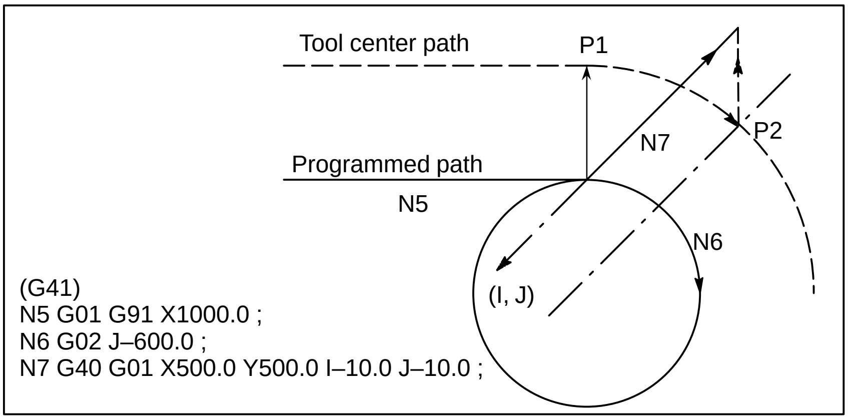


When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



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

In the example shown below, the tool does not trace the circle more than once. It moves along the arc from P1 to P2. The interference check function described in Subsection 15.2.5 may raise an alarm.



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

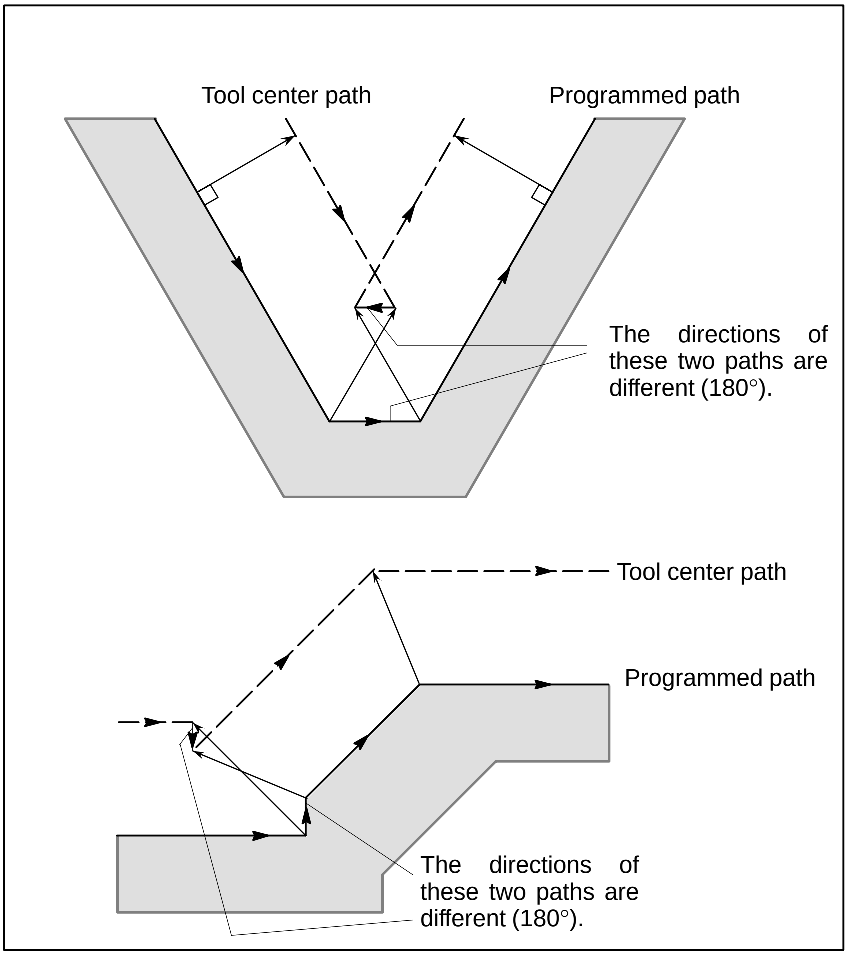
15.2.5 Interference Check

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

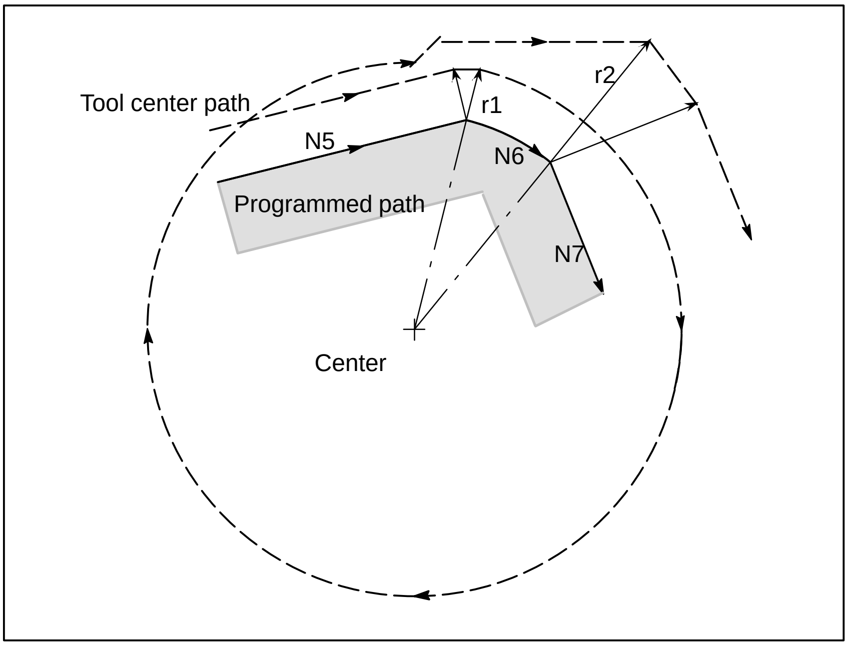
Explanations

- **Criteria for detecting interference**

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



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



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

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

- **Correction of interference in advance**

(1) Removal of the vector causing the interference

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

Check between vectors V_4 and V_5

Interference —– V_4 and V_5 are ignored.

Check between V_3 and V_6

Interference —– V_3 and V_6 are ignored

Check between V_2 and V_7

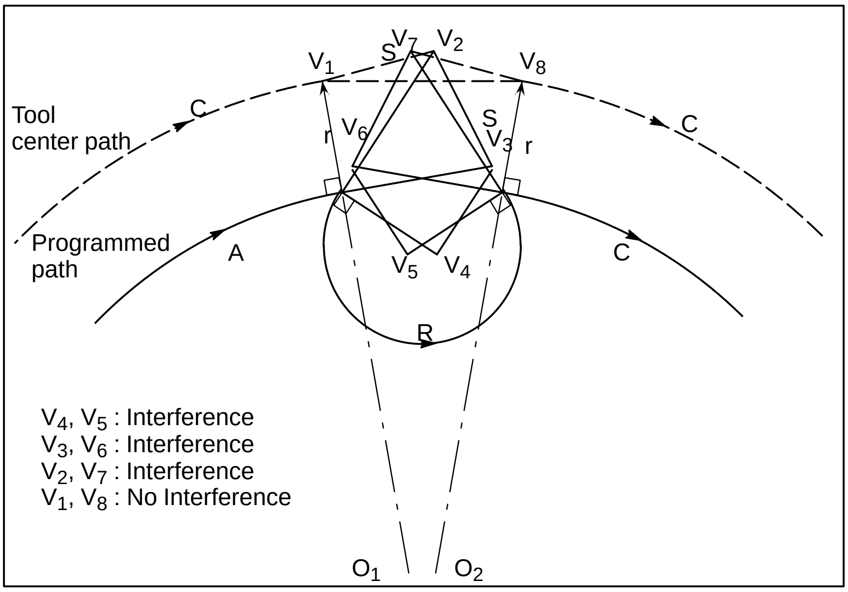
Interference —– V_2 and V_7 are Ignored

Check between V_1 and V_8

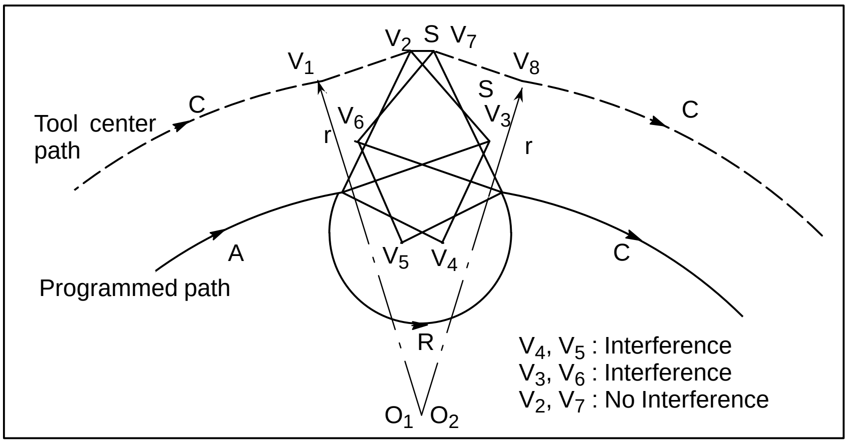
Interference ——– V_1 and V_8 are cannot be ignored

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

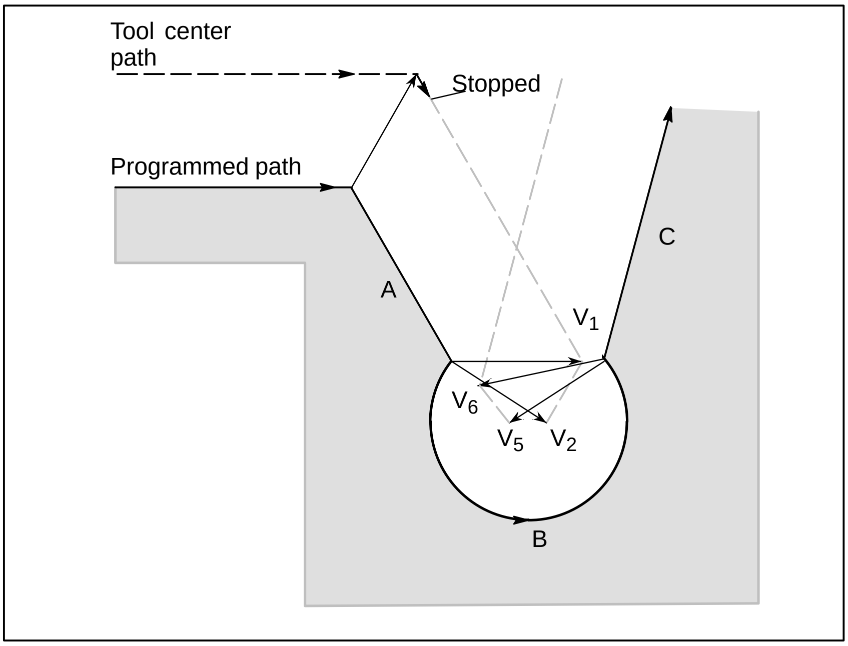
(Example 1) The tool moves linearly from V_1 to V_8



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



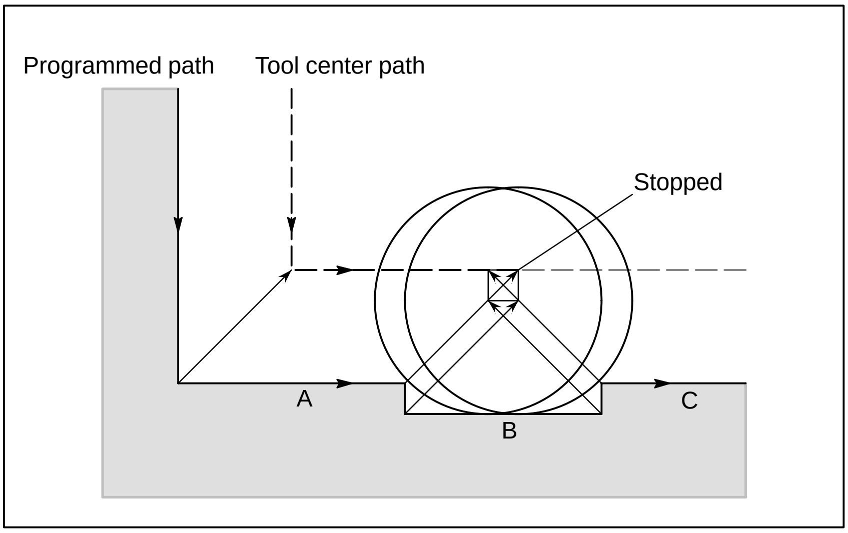
(2) If the interference occurs after correction (1), the tool is stopped with an alarm.
If the interference occurs after correction (1) or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the alarm (No.41) is displayed and the tool is stopped immediately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



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

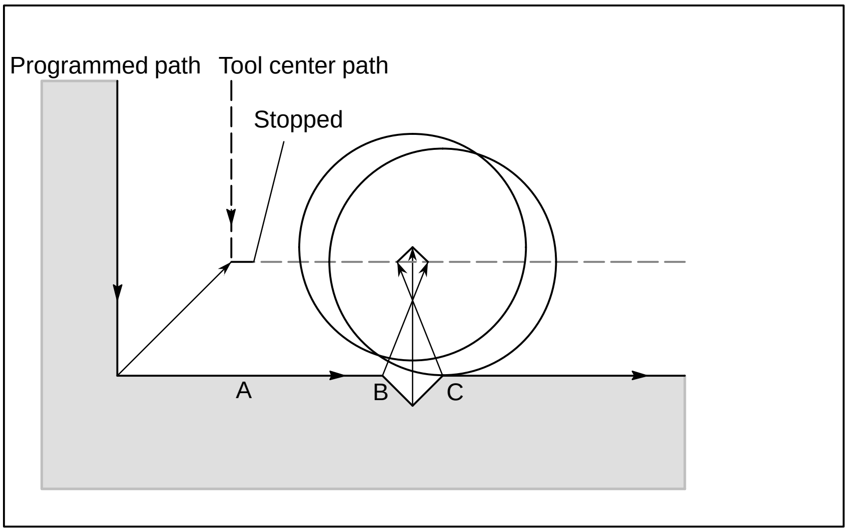
- **When interference is assumed although actual interference does not occur**

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



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

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



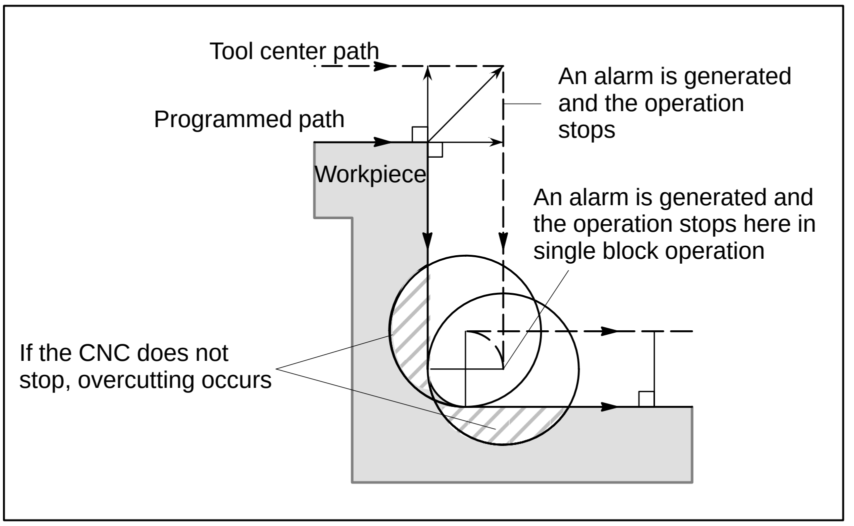
Like (1) , the direction is reverse in block B.

15.2.6 Overcutting by Cutter Compensation

Explanations

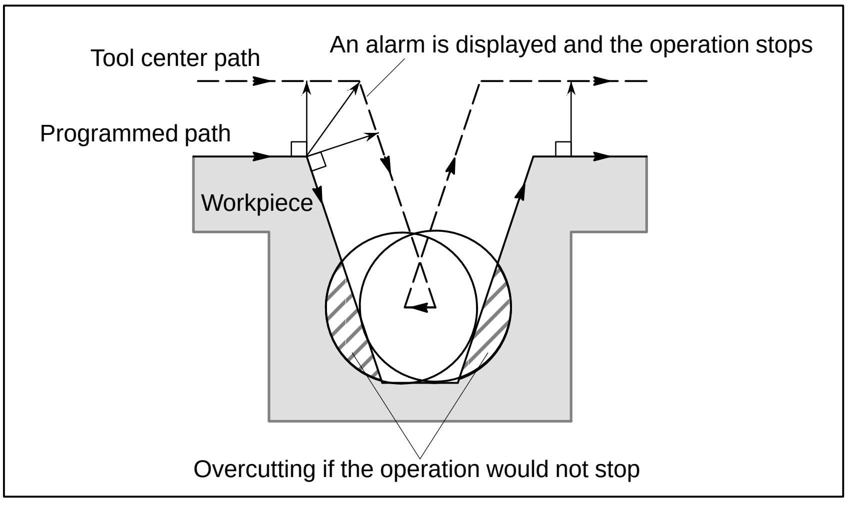
- **Machining an inside corner at a radius smaller than the cutter radius**

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



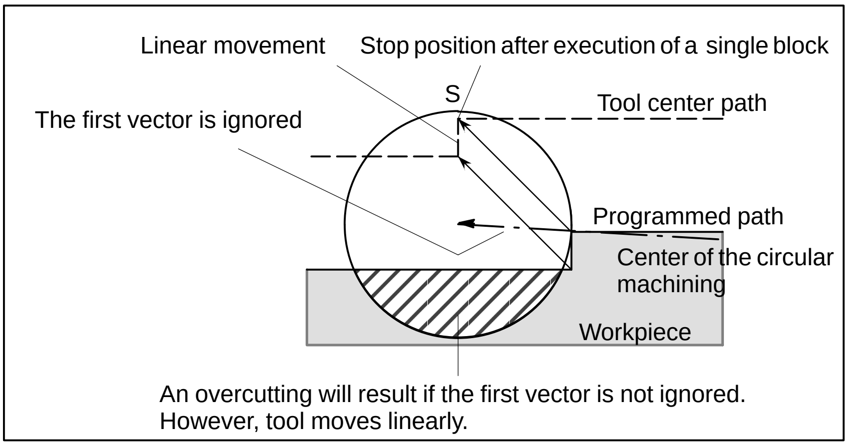
- **Machining a groove smaller than the tool radius**

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



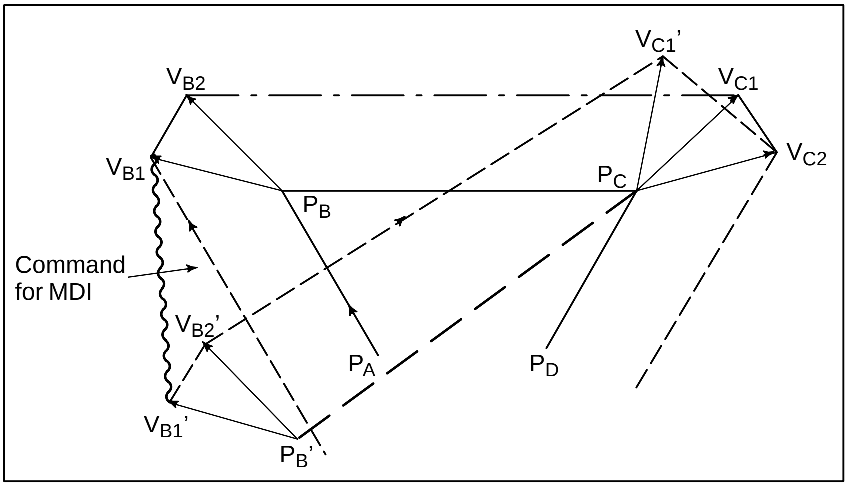
- **Machining a step smaller than the tool radius**

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



15.2.7 Input Command from MDI

Cutter compensation C is not performed for commands input from the MDI. However, when automatic operation using the CNC tape composed of absolute commands is temporarily stopped by the single block function, MDI operation is performed, then automatic operation starts again, the tool path is as follows :
In this case, the vectors at the start position of the next block are translated and the other vectors are produced by the next two blocks. Therefore, from next block but one, cutter compensation C is accurately performed.



When position P_A, P_B, and P_C are programmed in an absolute command, tool is stopped by the single block function after executing the block from P_A to P_B and the tool is moved by MDI operation. Vectors V_{B1} and V_{B2} are translated to V_{B1}' and V_{B2}' and offset vectors are recalculated for the vectors V_{C1} and V_{C2} between block P_B–P_C and P_C–P_D.
However, since vector V_{B2} is not calculated again, compensation is accurately performed from position P_C.

15.2.8 G53 and G28 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 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.

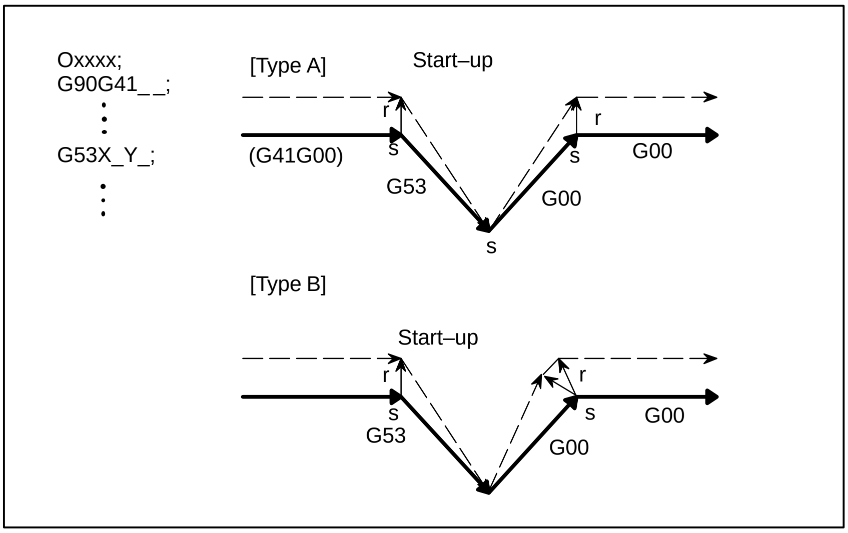
Explanations

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

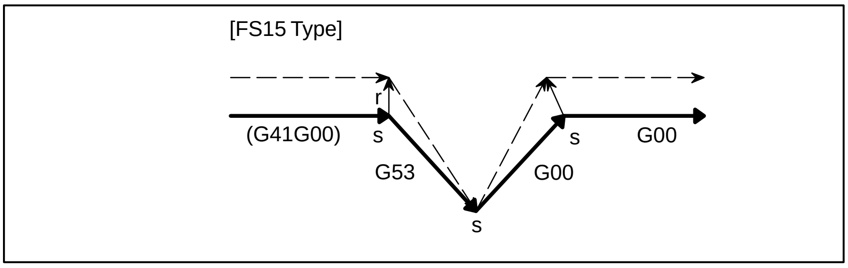
When G53 is specified in cutter compensation C mode, the previous block generates a vector that is perpendicular to the move direction and which has the same magnitude as the offset value. Then, the offset vector is canceled when movement to a specified position is performed in the machine coordinate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

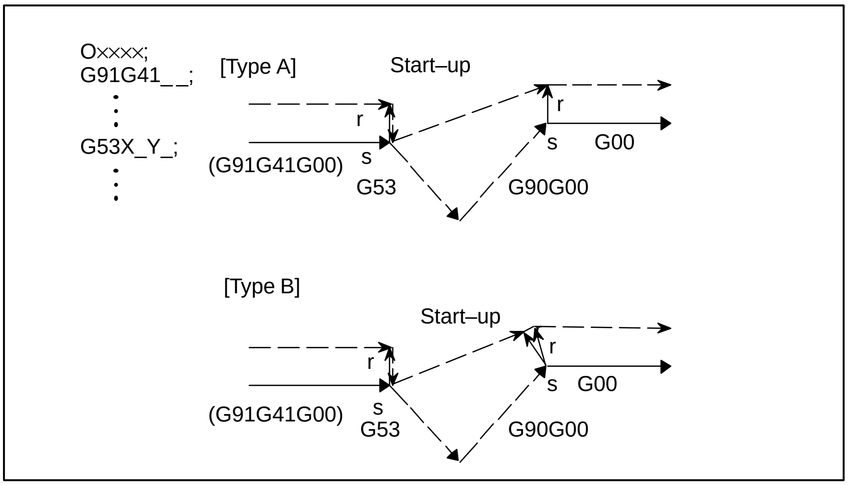
(1) G53 specified in offset mode
When CCN (bit 2 of parameter No.5003)=0



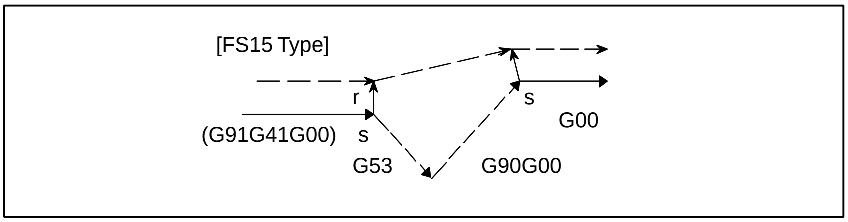
When CCN (bit 2 of parameter No.5003)=1



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



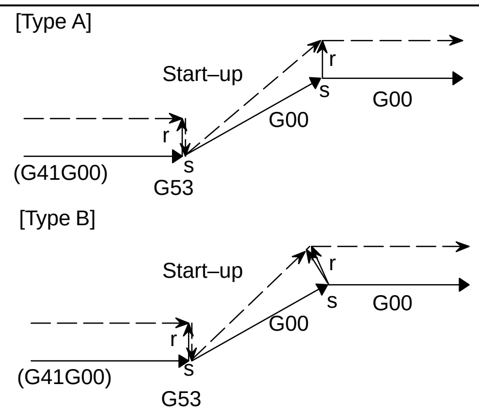
When CCN (bit2 of parameter No.5003)=1



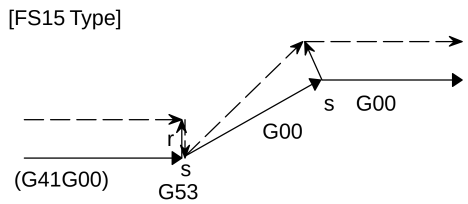
(3) G53 specified in offset mode with no movement specified

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

```
Oxxxx;
G90G41_ _;
  :
G00X20.Y20. ;
G53X20.Y20. ;
  :
```



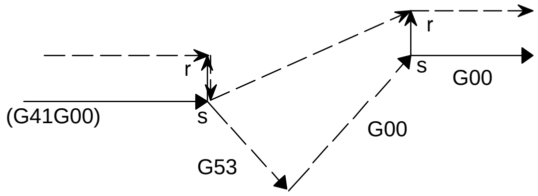
When CCN (bit2 of parameter No.5003)=1



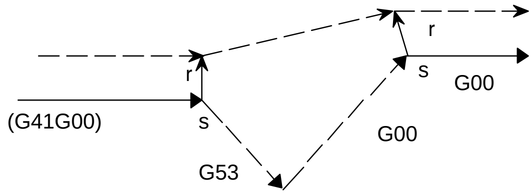
WARNING

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.)

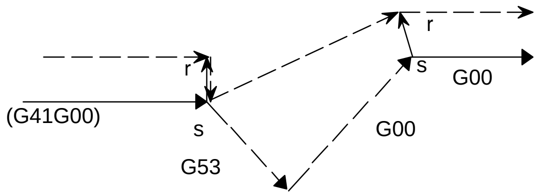
Example 1: When CCN (bit 2 of parameter No. 5003) = 0, type A is used, and all–axis machine lock is applied



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

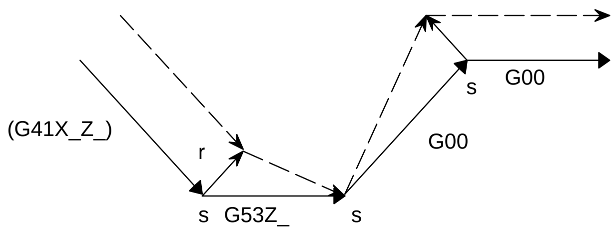


Example 3: When CCN (bit 2 of parameter No. 5003) = 1 and specified–axis machine lock is applied [FS15 type]



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.)

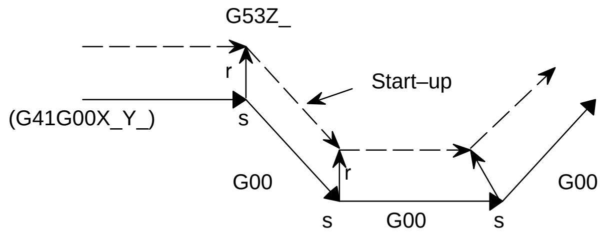
Example: When CCN (bit 2 of parameter No.5003)=1[FS 15 type]



NOTE

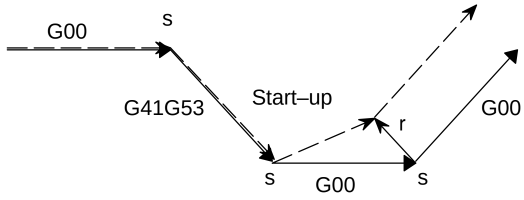
1 When a G53 command specifies an axis that is not in the cutter compensation C plane, a perpendicular vector is generated at the end point of the previous block, and the tool does not move. In the next block, offset mode is automatically resumed (in the same way as when two or more continuous blocks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

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

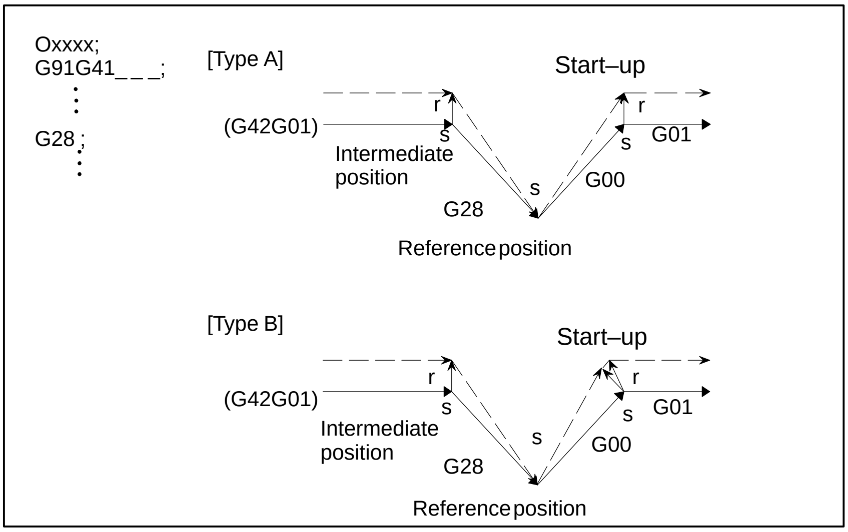


- **G28 command in cutter compensation C mode**

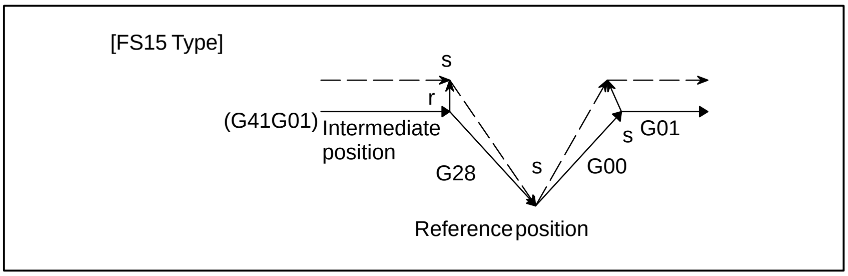
When G28 is specified in cutter compensation C mode, an operation of FS15 type is performed if CCN (bit 2 of parameter No. 5003) is set to 1.

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.

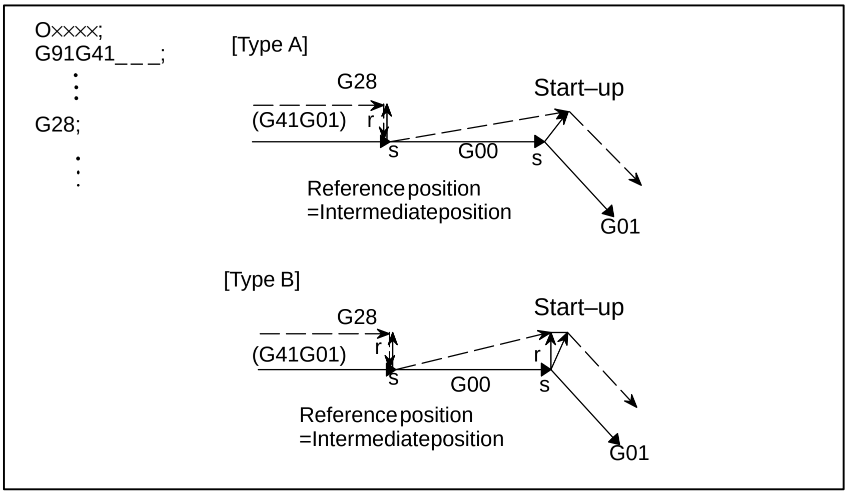
(1) G28 specified in offset mode (with movement to an intermediate position not performed)
When CCN (bit 2 of parameter No.5503)=0



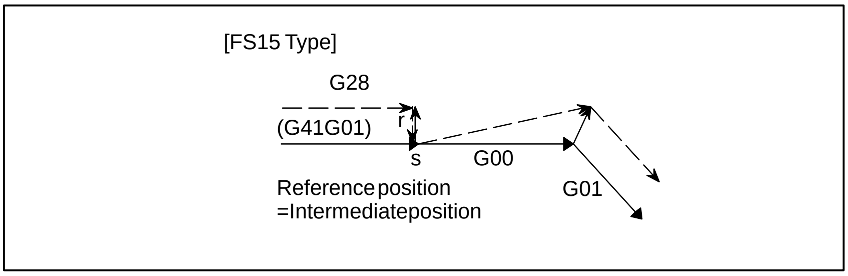
When CCN (bit 2 of parameter No.5503)=1



(2) G28 specified in offset mode (with no movement performed)
When CCN (bit 2 of parameter No.5503)=0



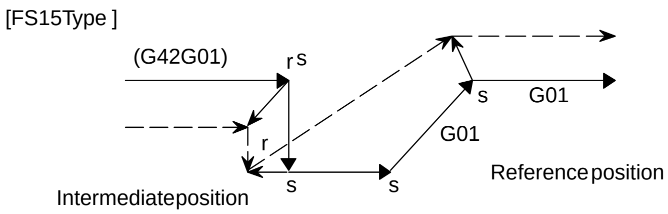
When CCN (bit 2 of parameter No.5503)=1



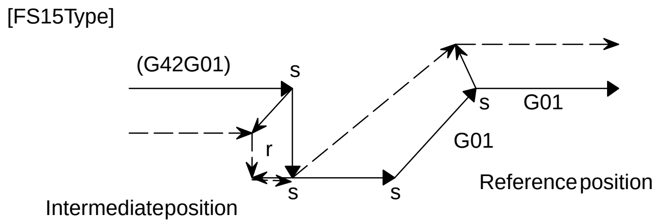
WARNING

1 When a G28 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

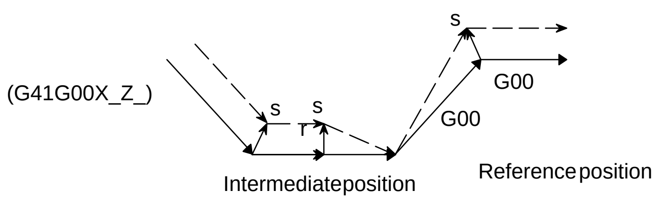


Example2: When CCN (bit 2 of parameter No.5003)=1 and each–axis machine lock is applied



2 When G28 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 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).
2 When a G28 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.

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

Tool compensation values can be entered into CNC memory from the screen panel (see section III–9.1) or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address D in a program.
The value is used for cutter compensation.

Explanations

- **Valid range of tool compensation values**

Table 15.3 shows the valid input range of tool compensation values.

Table15.3 The valid input range of tool compensation value

Increment system	Geometric compensation value	
	Metric input	Inch input
IS–A	±9999.99 mm	±999.999inch
IS–B	±999.999 mm	±99.9999inch

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

The memory can hold 32 tool compensation values.
Address D is used in the program.
The range of the number that comes after the address D depens on the number of tool compensation values : 0 to 32.

Format

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

G10L11P_R_ ;

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

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

15.4 SCALING (G50, G51)

A programmed figure can be magnified or reduced (scaling).
The dimensions specified with X_, and Y_, 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.

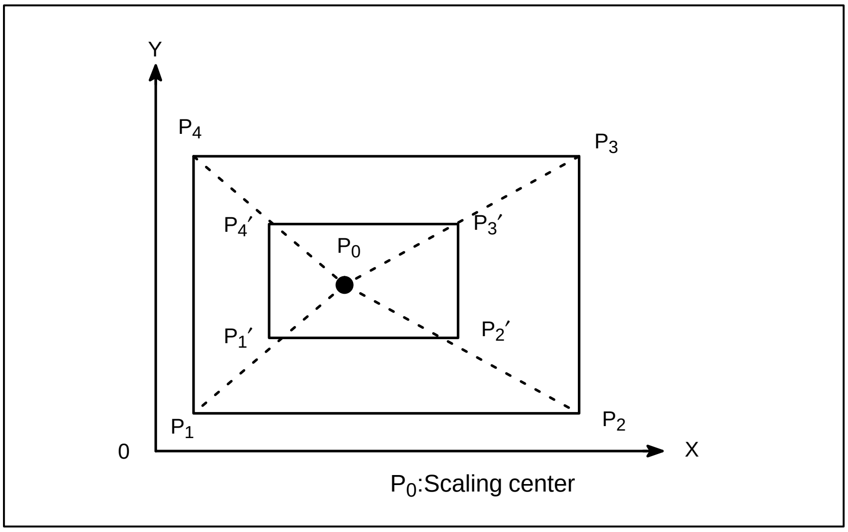


Fig. 15.4 (a) Scaling (P_1 P_2 P_3 P_4→$P_1'P_2'P_3'P_4'$)

Format

SCALING UP OR DOWN ALONG ALL AXES AT THE SAME RATE OF MAGNIFICATION

Format	Meaning of command
G51X_Y_P_ ; Scaling start ⋮ } Scaling is effective. (Scaling mode) **G50 ;** Scaling cancel	**X_Y_** : 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_I_J_K_;Scaling start ⋮ } Scaling is effective. (Scaling mode) **G50** Scaling cancel	**X_Y_** Absolute command for center coordinate value of scaling **I_J_K_** Scaling magnification for X axis and Y 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 (No. 5400#07) which value is selected. If scaling P is not specified on the block of scaling (G51X_Y_P_ ;), the scaling magnification set to parameter (No. 5411) is applicable. If X,Y, are omitted, the tool position where the G51 command was specified serves as the scaling center.

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

Each axis can be scaled by different magnifications. Also when a negative magnification is specified, a mirror image is applied. First of all, set a parameter XSC (No. 5400#6) which validates each axis scaling (mirror image).
Then, set parameter SCLx (No. 5401#0) to enable scaling along each axis.
Least input increment of scaling magnification of each axis (I, J, K) is 0.001 or 0.00001(set parameter SCR (No. 5400#7)).
Magnification is set to parameter 5421 within the range $\pm$0.00001 to $\pm$9.99999 or $\pm$0.001 to $\pm$9.999
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).

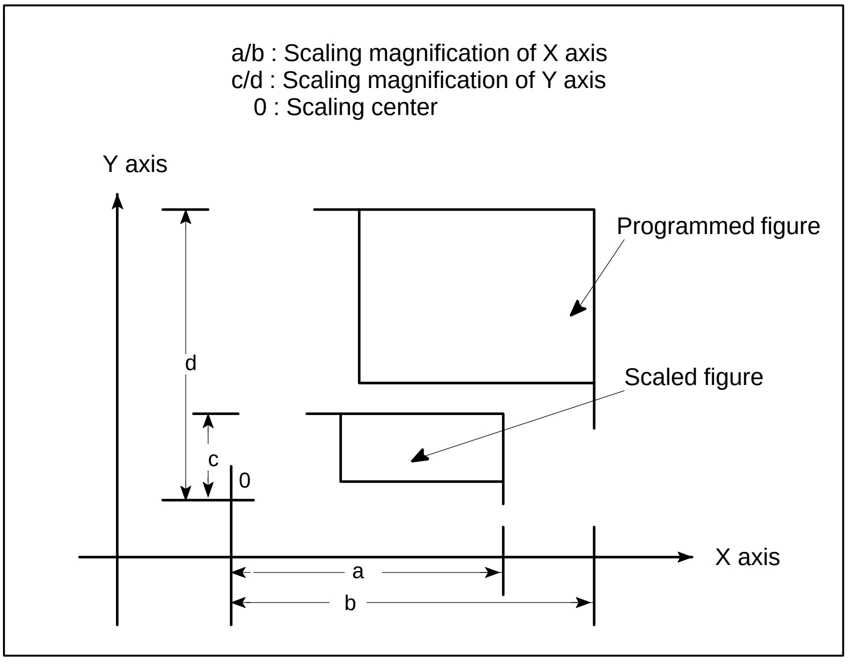


Fig. 15.4 (b) Scaling of each axis

- **Scaling of circular interpolation**

Even if different magnifications are applie to each axis in circular interpolation, the tool will not trace an ellipse.
When different magnifications are applied to axes and a circular interpolation is specified with radius R, it becomes as following figure 15.4 (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 I2000 J1000;
G02 X100.0 Y0.0 R100.0 F500 ;
```

Above commands are equivalent to the following command:

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

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

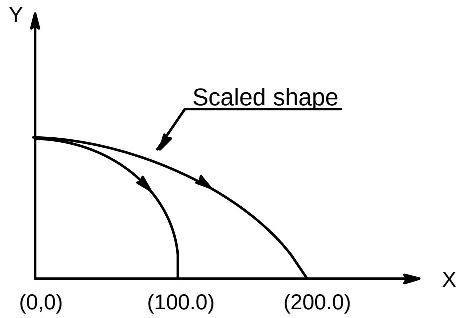


Fig. 15.4 (c) Scaling for circular interpolation1

When different magnifications are applied to axes and a circular interpolation is specified with I, J and K, it becomes as following figure 15.4 (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 Y100.0 ;
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.

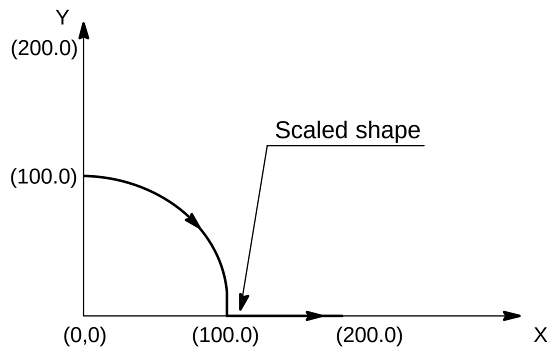


Fig. 15.4 (d) Scaling for circular interpolation 2

- **Invalid scaling**

This scaling is not applicable to cutter compensation values and tool offset values (Fig. 15.4 (e)).

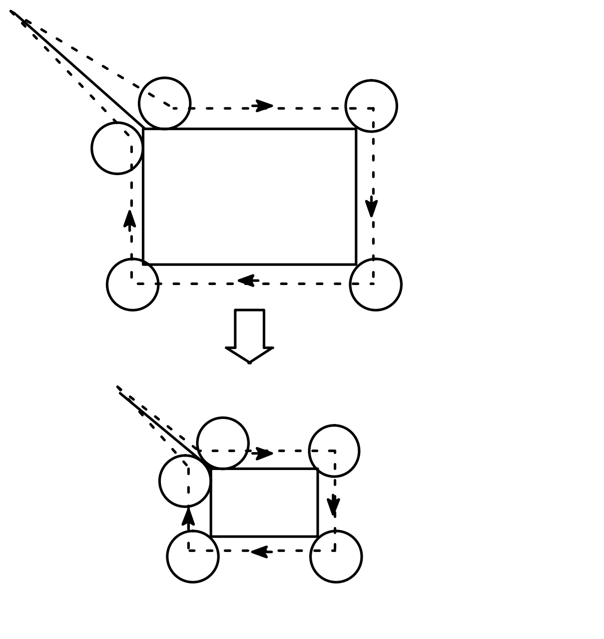


Fig. 15.4 (e) Scaling during cutter compensation

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

WARNING

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

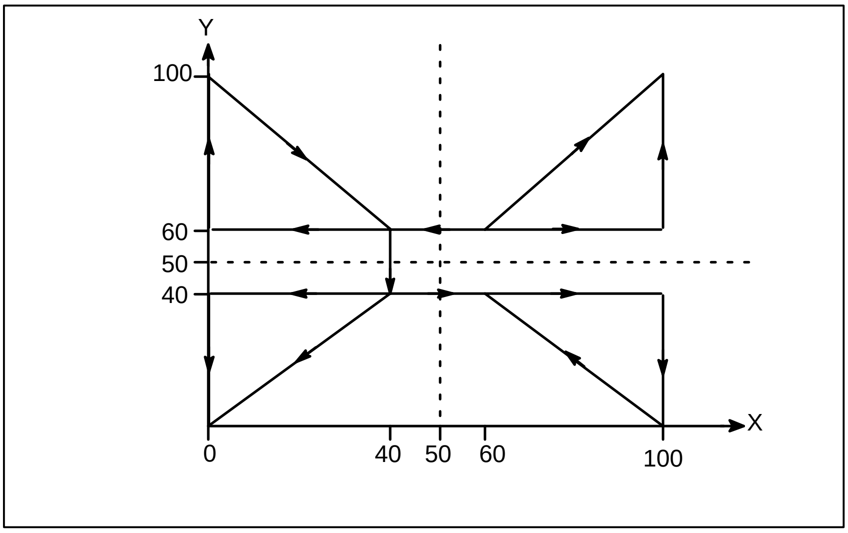


Fig. 15.4 (f) Example of a mirror image program

15.5 COORDINATE SYSTEM ROTATION (G84, G85)

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.

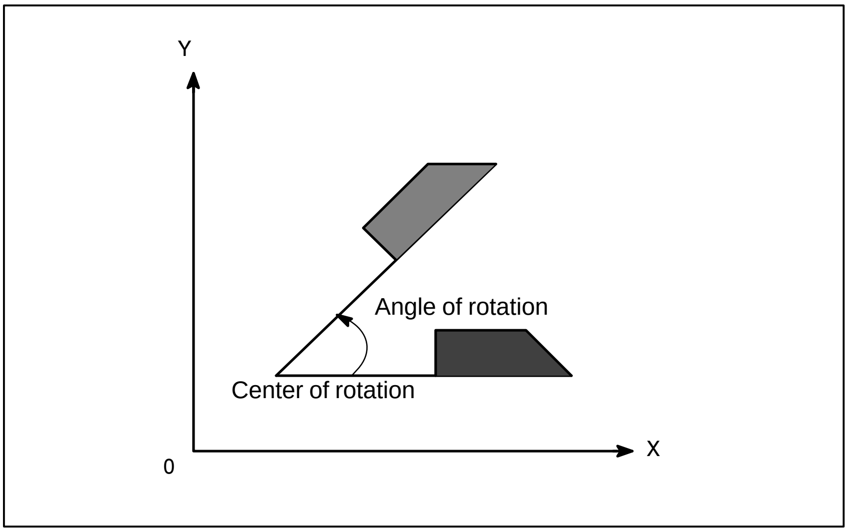


Fig. 15.5 (a) Coordinate system rotation

Format

Format

$\left\{\begin{matrix}\textbf{G17}\\ \textbf{G18}\\ \textbf{G19}\end{matrix}\right\}$ **G84** α_β_**R**_ ; Start rotation of a coordinate system.

Coordinate system rotation mode
(The coordinate system is rotated.)

G85 ; 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 G84.

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 to 360,000

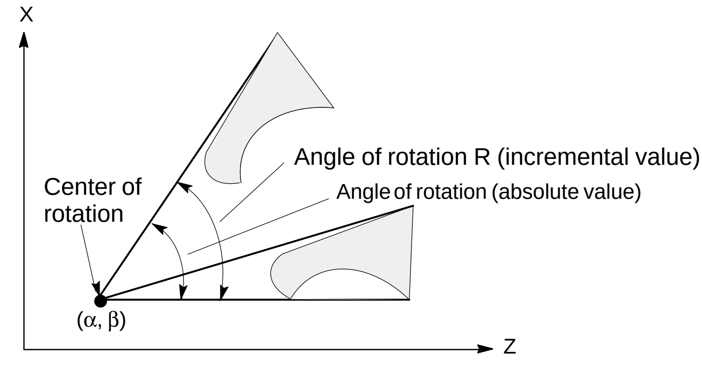


Fig. 15.5 (b) Coordinate system rotation

> **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 (G84). G17, G18 or G19 must not be designated in the mode of coordinate system rotation.
As for the incremental position commands designated between the G84 block and a block with an absolute command; it is regarded that the position where G84 was designated is the center of rotation (Fig. 15.5 (c)).
When α_ and β_ are omitted, the position where G84 is commanded is set as the center of rotation.
When angle of rotation is omitted, the value set to parameter (No. 5410) is regarded as the rotation angle. The coordinate system rotation is cancelled by G85;
G69 may be designated in the same block as the other commands. Tool offset, such as cutter compensation, or tool offset, is performed after the coordinate system is rotated for the command program.

> **WARNING**
> Be sure to command absolute command for the first movement command after G85.

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.

- **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.

```
N1 G92 X – 5000 Y – 5000 G85 G17 ;
N2 G84 X7000 Y3000 R60000 ;
N3 G90 G01 X0 Y0 F200 ;
   (G91X5000Y5000)
N4 G91 X10000 ;
N5 G02 Y10000 R10000 ;
N6 G03 X – 10000 I – 5000 J – 5000 ;
N7 G01 Y – 10000 ;
N8 G85 G90 X – 5000 Y – 5000 M02 ;
```

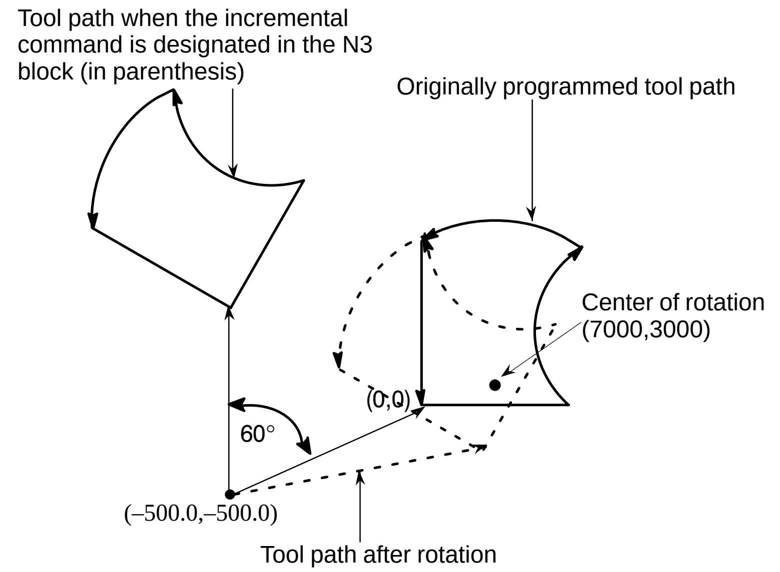


Fig. 15.5 (c) Absolute/incremental command during coordinate system rotation

Examples

- **Cutter compensation C and coordinate system rotation**

It is possible to specify G84 and G85 in cutter compensation C mode. The rotation plane must coincide with the plane of cutter compensation C.

N1 G92 X0 Y0 G85 G01 ;
N2 G42 G90 X1000 Y1000 F1000 D01 ;
N3 G84 R – 30000 ;
N4 G91 X2000 ;
N5 G03 Y1000 R1000 J500 ;
N6 G01 X – 2000 ;
N7 Y – 1000 ;
N8 G85 G40 G90 X0 Y0 M30 ;

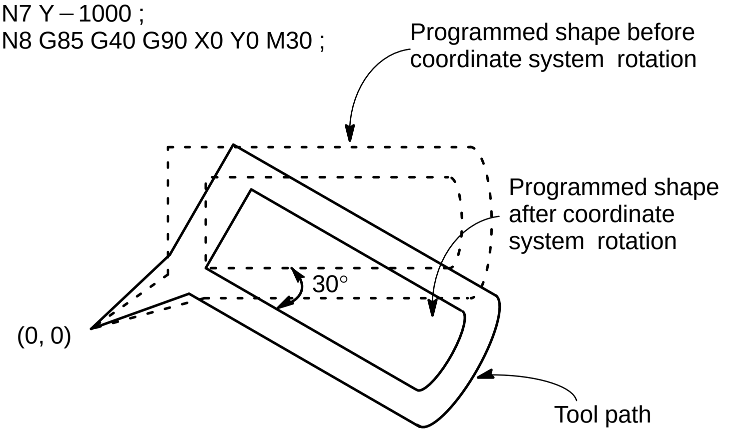


Fig. 15.5 (d) Cutter compensation C and coordinate system rotation

- **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 (G84) 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
G84 ; coordinate system rotation mode start

⋮

G85 ; 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.15.5 (e)) :
(cutter compensation C cancel)

G51 ; scaling mode start
G84 ; coordinate system rotation start
:
G41 ; cutter compensation C mode start
:

```
G92 X0 Y0 ;
G51 X3000 Y1500 P500 ;
G84 X2000 Y1000 R45000 ;
G01 X4000 Y1000 ;
Y1000 ;
X–2000 ;
Y–1000 ;
X2000 ;
```

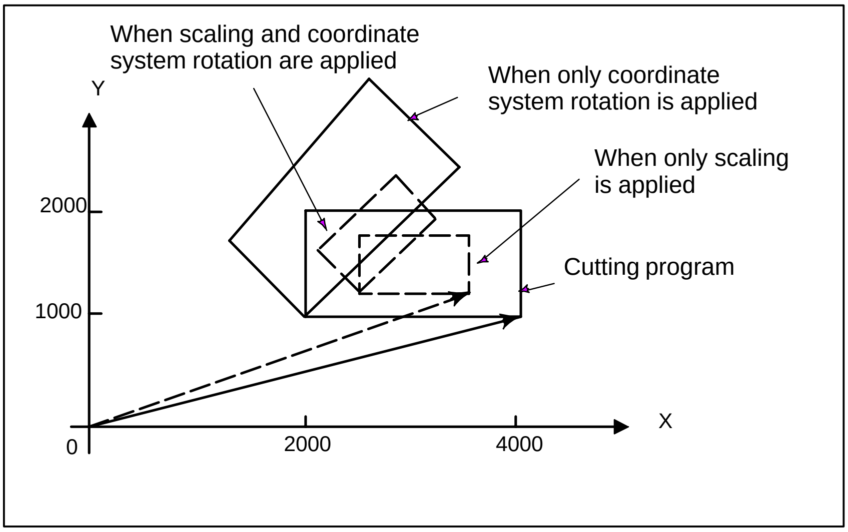


Fig. 15.5 (e) Scaling and coordinate system rotation in cutter compensation C mode

- **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 displacement is treated as an absolute or incremental value depending on the specified G code (G90 or G91).

```
  G92 X0 Y0 G85 G17;
  G01 F200 D01 ;
  M98 P2100 ;
  M98 P072200 ;
  G00 G90 X0 Y0 M30 ;

O 2200 G84 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 ;
```

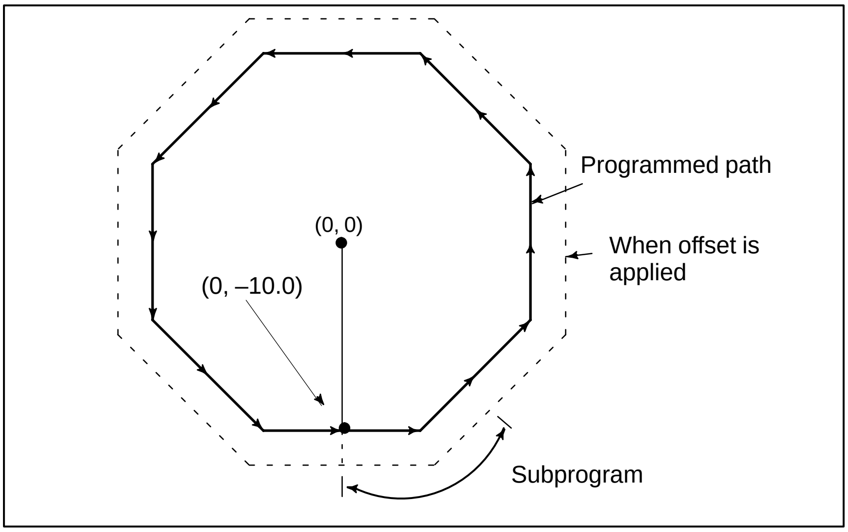


Fig. 15.5 (f) Coordinate system rotation command

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

When a tool with a rotation axis (C–axis) is moved in the XY plane during cutting, the normal direction control function can control the tool so that the C–axis is always perpendicular to the tool path (Fig. 15.6 (a)).

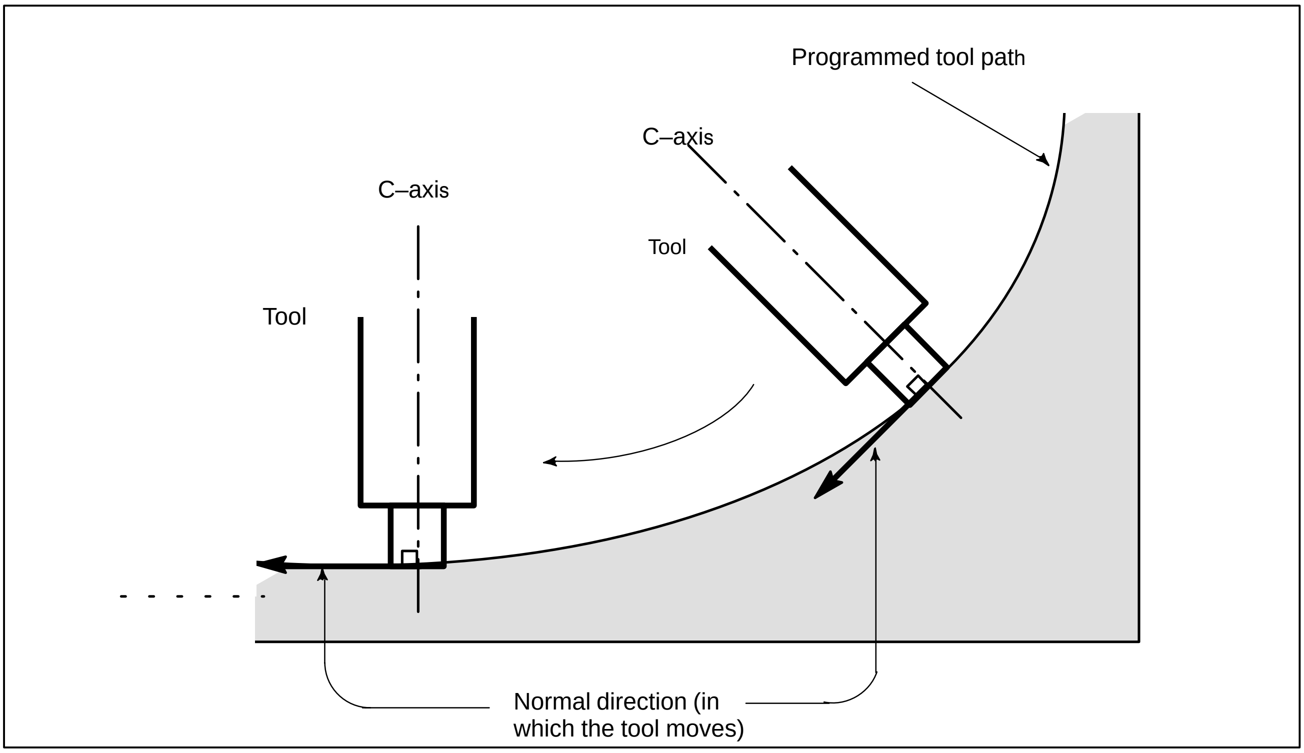
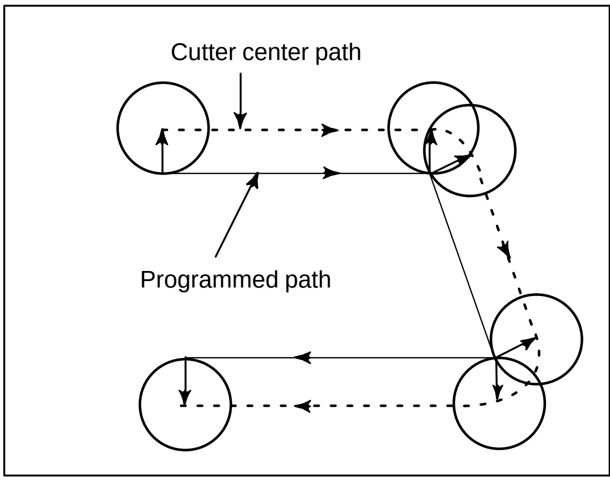


Fig. 15.6 (a) Sample Movement of the tool

Format

G code	Function	Explanation
G41.1 or **G151**	Normal direction control left	If the workpiece is to the right of the tool path looking toward the direction in which the tool 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	



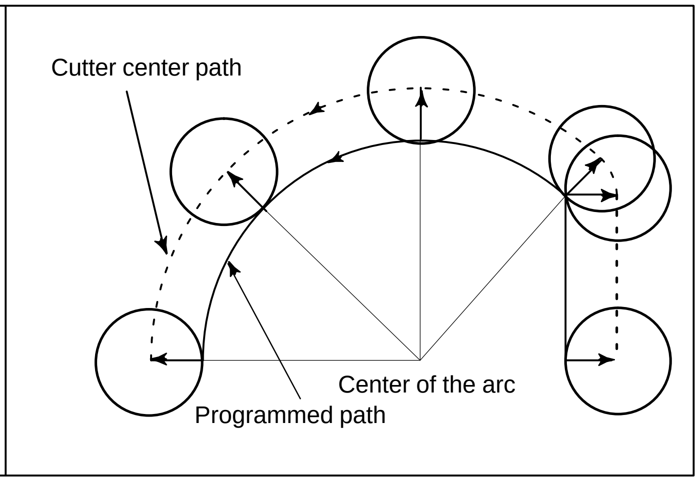


Fig15.6 (b) Normal direction control left (G41.1)

Fig15.6 (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. 15.6 (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°.

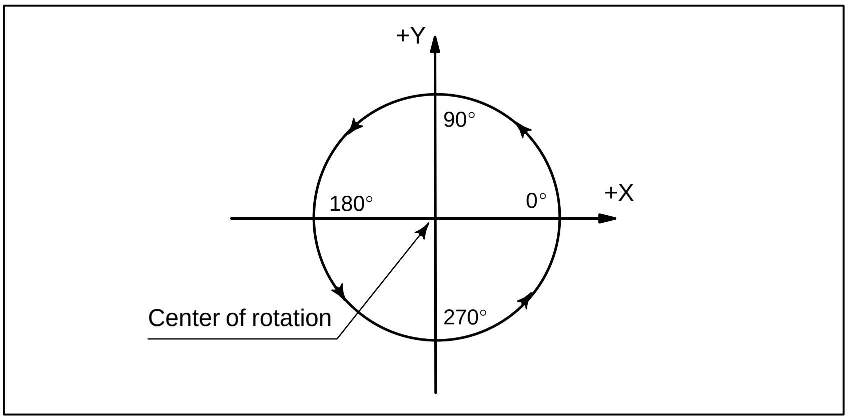


Fig. 15.6 (d) Angle of the C axis

- **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 tool 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 tool is automatically inserted so that the C–axis becomes perpendicular to the tool path at the beginning of each block. The tool is first oriented so that the C–axis becomes perpendicular to the tool path specified by the move command, then it is moved along the X– and Y axes.
In the cutter compensation mode, the tool is oriented so that the C–axis becomes perpendicular to the tool path created after compensation.
In single–block operation, the tool is not stopped between a command for rotation of the tool and a command for movement along the X– and Y–axes. A single–block stop always occurs after the tool is moved along the X– and Y–axes.

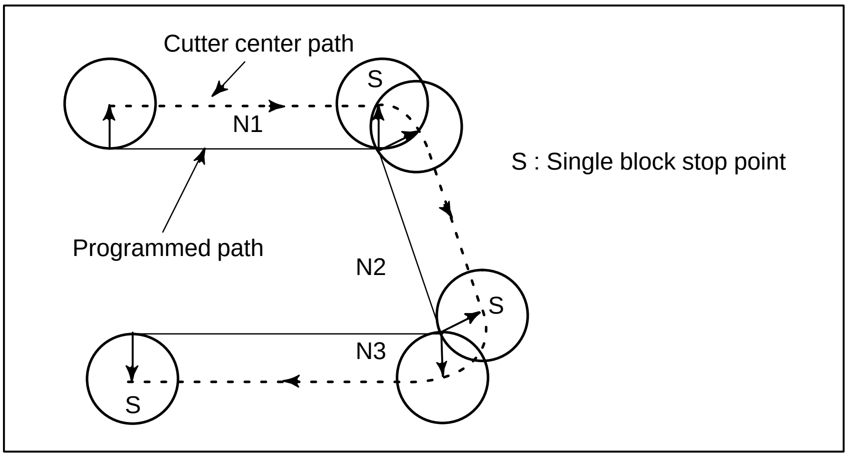


Fig. 15.6 (e) Point at which a Single–Block Stop Occurs in the Normal Direction Control Mode

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 tool is controlled so that the C–axis is always perpendicular to the tool path determined by circular interpolation.

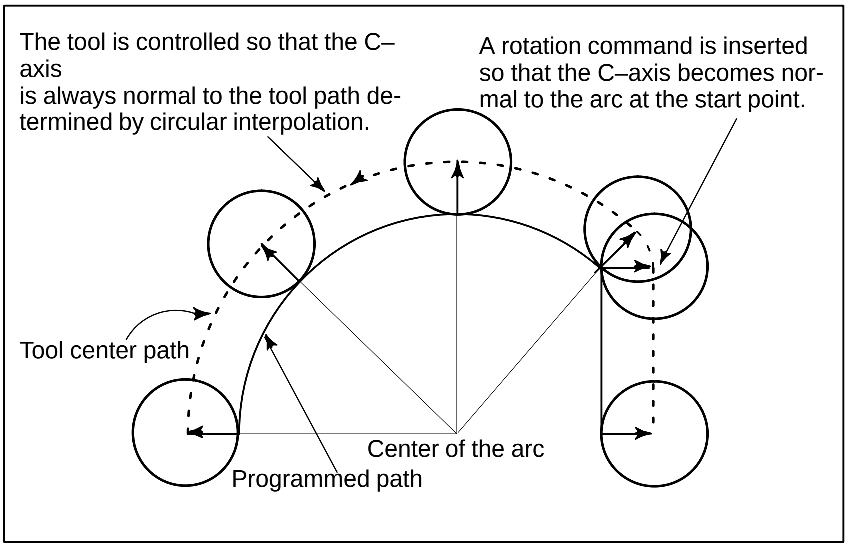


Fig. 15.6 (f) Normal direction control of the circular interpolation

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**

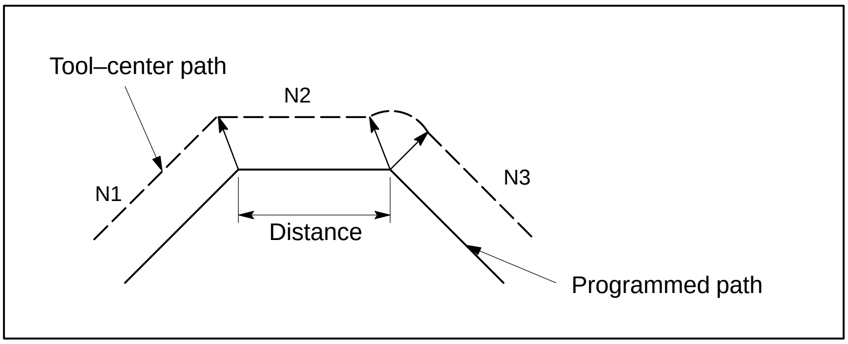
When the rotation angle to be inserted, calculated by normal–direction control, is smaller than the value set with parameter No. 5482, the corresponding rotation block is not inserted for the axis to which normal–direction control is applied. This ignored rotation angle is added to the next rotation 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.

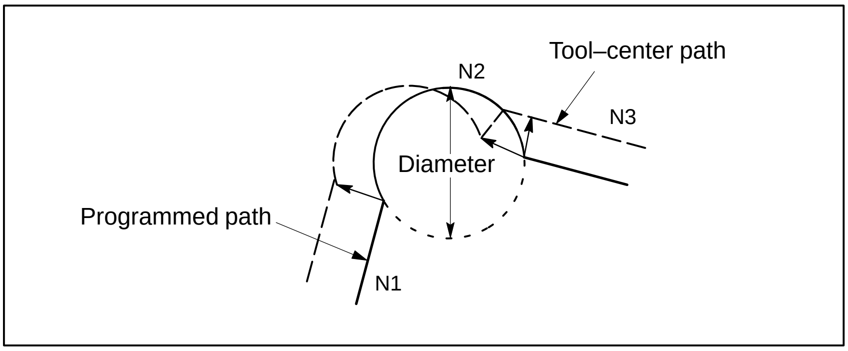
- **Linear movement**

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 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.

- **T and C commands during normal–line direction control**

1) During normal–line direction control, the T command results in an alarm (No. 4606) except when the TANDC parameter (bit 7 of parameter No. 16263) is 1, in which case the T command is valid.
2) During normal–line direction control, the C command is ignored. A C–axis offset command is also ignored if a C–axis index tool is specified.
3) A C–axis offset command specified before a normal–line direction control mode is entered remains valid after the normal–line direction control mode is entered.
4) Usually, a check is made on T commands during normal–line direction control. If a T command is already specified when a normal–line direction control mode is entered, a check is not made on the T command. Normal–line direction control is carried out no matter what tool type is specified.

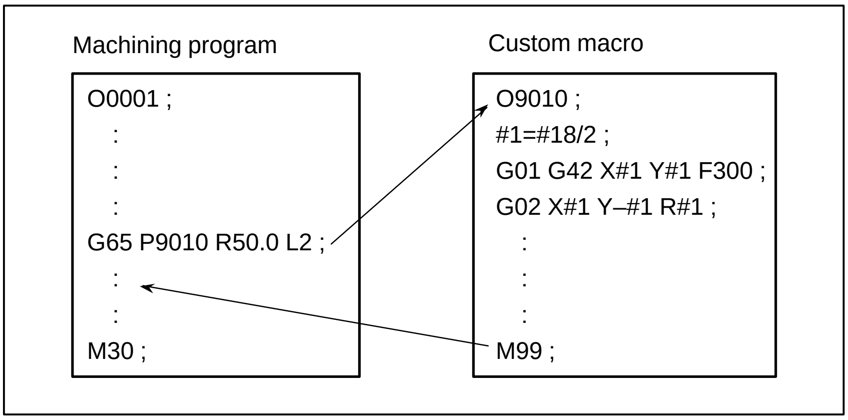
Under normal direction control, a punch press shears a sheet while holding the sheet and moving the tool with interpolation. The cutting of an arc should be programmed as follows: Linear positioning is performed with the C–axis set in the direction normal to the tangential direction at the start point of the arc; The sheet is held by the PF signal; A C–axis normal–direction movement is made with circular interpolation. The program should be created so that an arc and a straight line are connected by a tangent even in the normal direction control mode.

[Example]

```
G92 X1525 Y1525
G90
T201
G70 X500 Y200   ← Positioning for creating a tangent at the start point
                  of the arc
G41.1                       (G70: Positioning and press off command)
G00 X500 Y500   ← Positioning to the arc start point with the C–axis set
                  in the direction normal to the direction of the arc
                  tangent
G03 X421.14 Y459.171–50F5000
G40.1
```

16 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.



16.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 and X100.0.
With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI panel.

```
#1=#2+100 ;
G01 X#1 F300 ;
```

Explanation

- **Variable representation**

When specifying a variable, specify a number sign (#) followed by a variable number. Personal computers allow a name to be assigned to a variable, but this capability is not available for custom macros.

Example: #1

An expression can be used to specify a variable number. In such a case, the expression must be enclosed in brackets.

Example: #[#1+#2–12]

- **Types of variables**

Variables are classified into four types by variable number.

Table 16.1 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 to #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 to #199 #500 to #999	Common variables	Common variables can be shared among different macro programs. When the power is turned off, variables #100 to #199 are initialized to null. Variables #500 to #999 hold data even when the power is turned off.
#1000 to	System variables	System variables are used to read and write a variety of NC data items such as the current position and tool compensation values.

- **Range of variable values**

Local and common variables can have value 0 or a value in the following ranges :
-10^{47} to -10^{-29}
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

- **Displaying variable values**

Procedure for displaying variable values

Procedure

1 Press the OFFSET SETTING key to display the tool compensation screen.

2 Press the continuous menu key ▷.

3 Press the soft key **[MACRO]** to display the macro variable screen.

4 Enter a variable number, then press soft key **[NO.SRH]**.
The cursor moves to the position of the entered number.

```
VARIABLE                                O1234 N12345
  NO.        DATA         NO.         DATA
  100        123.456      108
  101          0.000      109
  102                     110
  103                     111
  104                     112
  105                     113
  106                     114
  107                     115

ACTUAL POSITION (RELATIVE)
     X        0.000           Y       0.000
     Z        0.000           B       0.000

MEM **** *** ***         18:42:15
[ MACRO ] [ 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;

16.2 SYSTEM VARIABLES

System variables can be used to read and write internal NC data such as tool compensation values and current position data. Note, however, that some system variables can only be read. System variables are essential for automation and general–purpose program development.

Explanations

• Interface signals

Signals can be exchanged between the programmable machine controller (PMC) and custom macros.

Table 16.2 (a) System variables for interface signals

Variable number	Function
#1000 to #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 to #1115 #1132	A 16–bit signal can be sent from a custom macro to the PMC. Variables #1100 to #1115 are used to write a signal bit by bit. Variable #1132 is used to write all 16 bits of a signal at one time.
#1133	Variable #1133 is used to write all 32 bits of a signal at one time from a custom macro to the PMC. Note, that values from –99999999 to +99999999 can be used for #1133.

For detailed information, refer to the connection manual (B–63833EN–1).

• Tool compensation values

Tool compensation values can be read and written using system variables.

Table 16.2 (b)

Compensation number	System variable
1 : 32	#10001 (#2001) : #10032 (#2032)

• Macro alarms

Table 16.2 (c) System variable for macro alarms

Variable number	Function
#3000	When a value from 0 to 200 is assigned to variable #3000, the NC 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 16.2 (d) System variables for time information

Variable number	Function
#3001	This variable functions as a timer that counts in 1–millisecond increments at all times. When the power is turned on, the value of this variable is reset to 0. When 65535 milliseconds is reached, the value of this timer returns to 0.
#3002	This variable functions as a timer that counts in 1–hour increments when the cycle start lamp is on. This timer preserves its value even when the power is turned off. When 1145324.612 hours is reached, the value of this timer returns to 0.
#3011	This variable can be used to read the current date (year/month/day). Year/month/day information is converted to an apparent decimal number. For example, March 28, 2001 is represented as 20010328.
#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 16.2 (e) System variable (#3003) for automatic operation control

#3003	Single block	Completion of an auxiliary function
0	Enabled	To be awaited
1	Disabled	To be awaited
2	Enabled	Not to be awaited
3	Disabled	Not to be awaited

- When the power is turned on, the value of this variable is 0.
- When single block stop is disabled, single block stop operation is not performed even if the single block switch is set to ON.

- When a wait for the completion of auxiliary functions (M, S, and T functions) is not specified, program execution proceeds to the next block before completion of auxiliary functions. Also, distribution completion signal DEN is not output.

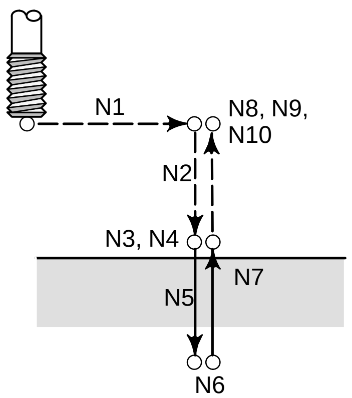
Table 16.2 (f) System variable (#3004) for automatic operation control

#3004	Feed hold	Feedrate Override	Exact stop
0	Enabled	Enabled	Enabled
1	Disabled	Enabled	Enabled
2	Enabled	Disabled	Enabled
3	Disabled	Disabled	Enabled
4	Enabled	Enabled	Disabled
5	Disabled	Enabled	Disabled
6	Enabled	Disabled	Disabled
7	Disabled	Disabled	Disabled

- When the power is turned on, the value of this variable is 0.
- When feed hold is disabled:

  <1> When the feed hold button is held down, the machine stops in the single block stop mode. However, single block stop operation is not performed when the single block mode is disabled with variable #3003.

  <2> When the feed hold button is pressed then released, the feed hold lamp comes on, but the machine does not stop; program execution continues and the machine stops at the first block where feed hold is enabled.
- When feedrate override is disabled, an override of 100% is always applied regardless of the setting of the feedrate override switch on the machine operator's panel.
- When exact stop check is disabled, no exact stop check (position check) is made even in blocks including those which do not perform cutting.




```
O0001 ;
N1  G00 G91 X#24 Y#25 ;
N2  Z#18 ;
    G04 ;
N3  #3003=3 ;
N4  #3004=7 ;
N5  G01 Z#26 F#9 ;
N6  M04 ;
N7  G01
Z–[ROUND[#18]+ROUND[#26]] ;
    G04 ;
N8  #3004=0 ;
N9  #3003=0 ;
N10 M03 ;
    M99 ;
```

Fig. 16.2 Example of using variable #3004 in a tapping cycle

• Settings

Settings can be read and written. Binary values are converted to decimals.

#3005

	#15	#14	#13	#12	#11	#10	#9	#8
Setting								
	#7	**#6**	**#5**	**#4**	**#3**	**#2**	**#1**	**#0**
Setting			SEQ			INI	ISO	TVC

#5 (SEQ) : Whether to automatically insert sequence numbers
#2 (INI) : Millimeter input or inch input
#1 (ISO) : Whether to use EIA or ISO as the output code
#0 (TVC) : Whether to make a TV check

• Mirror image

The mirror–image status for each axis set using an external switch or setting operation can be read through the output signal (mirror–image check signal). The mirror–image status present at that time can be checked. (See Section 4.7 in III.)
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 16.2 (g) System variables for the number of parts required and the number of machined parts

Variable number	Function
#3901	Number of machined parts (completion number)
#3902	Number of required parts (target number)

WARNING
Do not substitute a negative value.

- **Modal information**

Modal information specified in blocks up to the immediately preceding block can be read.

Table 16.2 (h) 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)
#4014	G54–G59	(Group 14)
#4015	G61–G64	(Group 15)
#4016	G84, G85	(Group 16)
:	:	:
#4022		(Group 22)
#4102	B 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 0, 1, 2, 3, or 33.

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 16.2 (i) System variables for position information

Variable number	Position information	Coordinate system	Tool compensation value	Read operation during movement
#5001 to #5004	Block end point	Workpiece coordinate system	Not included	Enabled
#5021 to #5024	Current position	Machine coordinate system	Included	Disabled
#5041 to #5044	Current position	Workpiece coordinate system		
#5061 to #5064	Skip signal position			Enabled
#5101 to #5104	Deviated servo position			Disabled
#6251 to #6254	Pattern base position	Workpiece coordinate system	Not included	Enabled
#6261 to #6264	Multi–piece machining coordinate system	Workpiece coordinate system	Not included	Enabled
#6271 to #6274	Local coordinate system	Workpiece coordinate system	Not included	Enabled

- The first digit (from 1 to 4) represents an axis number.
- The tool offset value currently used for execution rather than the immediately preceding tool offset value is held in variables #5081 to 5088.
- The tool position where the skip signal is turned on in a G33 (skip function) block is held in variables #5061 to #5068. When the skip signal is not turned on in a G33 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 16.2 (j) System variables for workpiece zero point offset values

Variable number	Function
#5201 : #5204	First–axis external workpiece zero point offset value : Fourth–axis external workpiece zero point offset value
#5221 : #5224	First–axis G54 workpiece zero point offset value : Fourth–axis G54 workpiece zero point offset value
#5241 : #5244	First–axis G55 workpiece zero point offset value : Fourth–axis G55 workpiece zero point offset value
#5261 : #5264	First–axis G56 workpiece zero point offset value : Fourth–axis G56 workpiece zero point offset value
#5281 : #5284	First–axis G57 workpiece zero point offset value : Fourth–axis G57 workpiece zero point offset value
#5301 : #5304	First–axis G58 workpiece zero point offset value : Fourth–axis G58 workpiece zero point offset value
#5321 : #5324	First–axis G59 workpiece zero point offset value : Fourth–axis G59 workpiece zero point offset value

16.3 ARITHMETIC AND LOGIC OPERATION

The operations listed in Table 16.3 (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 16.3 (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 Cosine Arccosine 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: 0o 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 tool to the original position.

This difference comes from whether addition is performed before or after rounding off. G00X–[ROUND[#1]+ROUND[#2]] must be specified to return the tool to the original position.

- **Rounding up and down to an integer**

With CNC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handling negative numbers.

Example:

Suppose that #1=1.2 and #2=–1.2.
When #3=FUP[#1] is executed, 2.0 is assigned to #3.
When #3=FIX[#1] is executed, 1.0 is assigned to #3.
When #3=FUP[#2] is executed, –2.0 is assigned to #3.
When #3=FIX[#2] is executed, –1.0 is assigned to #3.

- **Abbreviations of arithmetic and logic operation commands**

When a function is specified in a program, the first two characters of the function name can be used to specify the function (See III–9.6).

Example:

ROUND → RO
FIX → FI

- **Priority of operations**

$\boxed{1}$ Functions
$\boxed{2}$ Operations such as multiplication and division (*, /, AND, MOD)
$\boxed{3}$ Operations such as addition and subtraction (+, –, OR, XOR)

Example) #1=#2+#3*SIN[#4];

$$\#1=\underbrace{\#2+\underbrace{\#3*\underbrace{SIN[\#4]}_{\boxed{1}}}_{\boxed{2}}}_{\boxed{3}};$$

$\boxed{1}$, $\boxed{2}$ and $\boxed{3}$ indicate the order of operations.

- **Bracket nesting**

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, alarm No. 118 occurs.

Example) #1=SIN [[[#2+#3] *#4 +#5] *#6] ;

$$\#1=\underbrace{SIN\,[\underbrace{[\underbrace{[\underbrace{\underbrace{\#2+\#3}_{\boxed{1}}]*\#4}_{\boxed{2}}+\#5}_{\boxed{3}}]*\#6}_{\boxed{4}}]}_{\boxed{5}};$$

$\boxed{1}$ 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 16.3 (b) Errors involved in operations

Operation	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	Relative error(*1) $\left\lvert \frac{\varepsilon}{a} \right\rvert$
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}	(*2) 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(*3) $\lvert \varepsilon \rvert$ degrees
a = ATAN [b] / [c] (*4)	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 x 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.

16.4 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 6000 is 1.
- Macro blocks are not regarded as blocks that involve no movement in the cutter compensation mode (see Section 16.7).

- **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 also include an O, N, P, or L address have the same property as macro statements.
- NC statements that include M99 and an O, N, L, or P address have the same property as macro statements.

16.5 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 ...)

16.5.1 Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999 is specified, alarm No. 128 occurs. A sequence number can also be specified using an expression.

GOTO n ;	n: Sequence number (1 to 99999)

Example:

```
GOTO1;
GOTO#10;
```

16.5.2 Conditional Branch (IF Statement)

Specify a conditional expression after IF. If the specified conditional expression is satisfied, a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.

If the condition is not satisfied

```
IF [#1 GT 10] GOTO 2 ;
  Processing
N2 G00 G91 X10.0 ;
  :
```

If the condition is satisfied

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 16.5.2 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(≦)

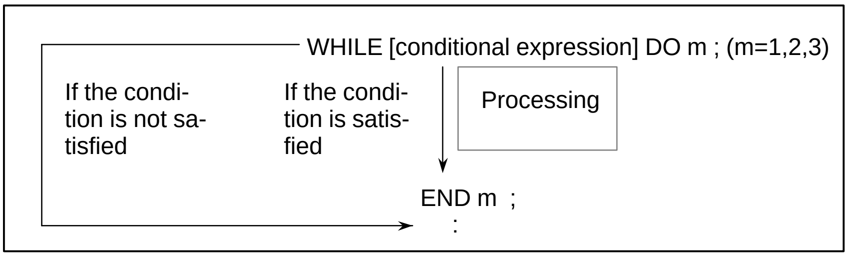
Sample program

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
```

16.5.3 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.



Explanations

While the specified condition is satisfied, the program from DO to END after WHILE is executed. If the specified condition is not satisfied, program execution proceeds to the block after END. The same format as for the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The numbers 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, alarm No. 126 occurs.

- **Nesting**

The identification numbers (1 to 3) in a DO–END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), alarm No. 124 occurs.

1. The identification numbers (1 to 3) can be used as many times as required.

```
WHILE [ … ] DO 1 ;
  Processing
END 1 ;
  :
WHILE [ … ] DO 1 ;
  Processing
END 1 ;
```

2. DO ranges cannot overlap.

```
WHILE [ … ] DO 1 ;
  Processing
WHILE [ … ] DO 2 ;
  :
END 1 ;
  Processing
END 2 ;
```

3. DO loops can be nested to a maximum depth of three levels.

```
WHILE [ … ] DO 1 ;
  :
  WHILE [ … ] DO 2 ;
    :
    WHILE [ … ] DO 3 ;
      Processing
    END 3 ;
    :
  END 2 ;
  :
END 1 ;
```

4. Control can be transferred to the outside of a loop.

```
WHILE [ … ] DO 1 ;
IF [ … ] GOTO n ;
END 1 ;
Nn
```

5. Branches cannot be made to a location within a loop.

```
IF [ … ] GOTO n ;
  :
WHILE [ … ] DO 1 ;
Nn … ;
END 1 ;
```

Limitations

- **Infinite loops**

When DO m is specified without specifying the WHILE statement, an infinite loop ranging from DO to END is produced.

- **Processing time**

When a branch to the sequence number specified in a GOTO statement occurs, the sequence number is searched for. For this reason, processing in the reverse direction takes a longer time than processing in the forward direction. Using the WHILE statement for repetition reduces processing time.

- **Undefined variable**

In a conditional expression that uses EQ or NE, a null value and zero have different effects. In other types of conditional expressions, a null value is regarded as zero.

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO 1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

16.6 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.

16.6.1 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 ;
```

```
O9100 ;
#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.
- Addresses do not need to be specified alphabetically. They conform to word address format.
  I, J, and K need to be specified alphabetically, however.

Example

B_A_D_ … J_K_ Correct
B_A_D_ … J_I_ Incorrect

Argument specification II

Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three–dimensional coordinates as arguments.

Address	Variable number
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11

Address	Variable number
K_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22

Address	Variable number
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

- Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

Limitations

- **Format**

G65 must be specified before any argument.

- **Mixture of argument specifications I and II**

The NC 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.

Example

G65 A1.0 B2.0 I–3.0 I4.0 D5.0 P1000;

<Variables>
#1:1.0
#2:2.0
#3:
#4:–3.0
#5:
#6:
#7:~~4.0~~ 5.0

When both the I4.0 and D5.0 arguments are commanded for variable #7 in this example, the latter, D5.0, is valid.

- **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.

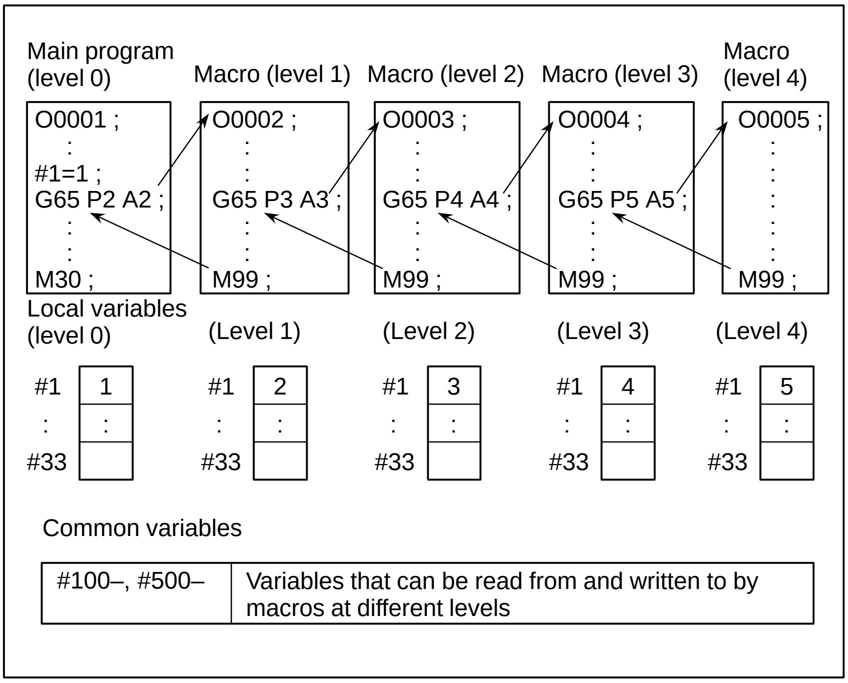
- **Call nesting**

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**

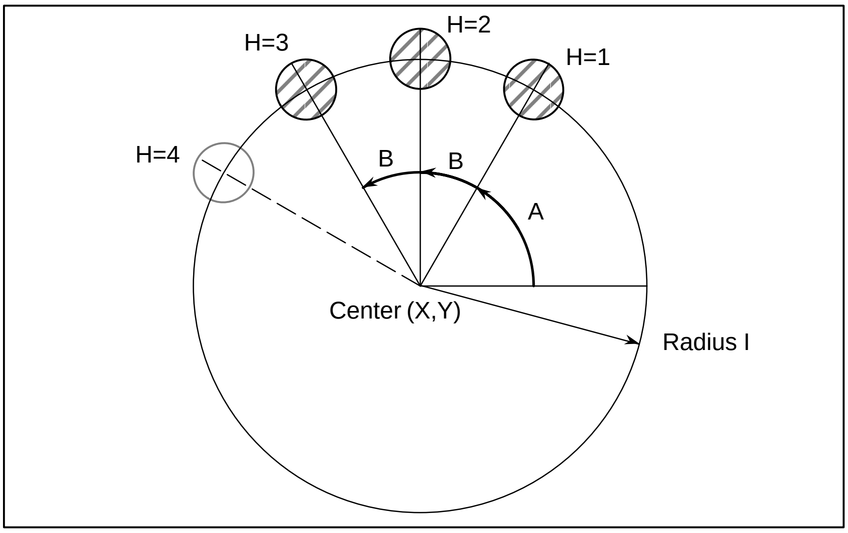
- Local variables from level 0 to 4 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G65 or G66), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the NC.

- When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.



Sample program (bolt hole circle)

A macro is created which drills H holes at intervals of B degrees after a start angle of A degrees along the periphery of a circle with radius I. The center of the circle is (X,Y). Commands can be specified in either the absolute or incremental mode. To drill in the clockwise direction, specify a negative value for B.



- **Calling format**

```
G65 P9100 Xx Yy Zz Rr Ff Ii Aa Bb Hh;
```

X: X coordinate of the center of the circle (absolute or incremental specification) . (#24)
Y: Y coordinate of the center of the circle (absolute or incremental specification) . (#25)
Z: Hole depth . . . (#26)
R: Coordinates of an approach point (#18)
F: Cutting feedrate (#9)
I: Radius of the circle (#4)
A: Drilling start angle (#1)
B: Incremental angle (clockwise when a negative value is specified) . . . . (#2)
H: Number of holes (#11)

- **Program calling a macro program**

O0002;
G90 G92 X0 Y0 Z100.0;
G65 P9100 X100.0 Y50.0 R30.0 Z–50.0 F500 I100.0 A0 B45.0 H5;
M30;

- **Macro program (called program)**

O9100;
#3=#4003; . . . . Stores G code of group 3.
G81 Z#26 R#18 F#9 K0; (Note) . . . . Drilling cycle.
. . . . Note: L0 can also be used.
IF[#3 EQ 90]GOTO 1; . . . . Branches to N1 in the G90 mode.
#24=#5001+#24; . . . . Calculates the X coordinate of the center.
#25=#5002+#25; . . . . Calculates the Y coordinate of the center.
N1 WHILE[#11 GT 0]DO 1;
. . . . Until the number of remaining holes reaches 0
#5=#24+#4*COS[#1]; . . Calculates a drilling position on the X–axis.
#6=#25+#4*SIN[#1]; . . Calculates a drilling position on the Y–axis.
G90 X#5 Y#6; . Performs drilling after moving to the target position.
#1=#1+#2; . . . . Updates the angle.
#11=#11–1; . . . . Decrements the number of holes.
END 1;
G#3 G80; . . . . Returns the G code to the original state.
M99;

Meaning of variables:
#3: Stores the G code of group 3.
#5: X coordinate of the next hole to drill
#6: Y coordinate of the next hole to drill

16.6.2 Modal Call (G66)

Once G66 is issued to specify a modal call a macro is called after a block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.

```
G66  P p  L  ℓ  <argument–specification> ;

     P :  Number of the program to call
     ℓ :  Repetition count (1 by default)
     Argument :  Data passed to the macro
```

```
O0001 ;
   :
G66 P9100 L2 A1.0 B2.0 ;
G00 G90 X100.0 ;
Y200.0 ;
X150.0 Y300.0 ;
G67 ;
   :
M30 ;
```

```
O9100 ;
   :
G00 Z–#1 ;
G01 Z–#2 F300 ;
   :
   :
   :
   :
M99 ;
```

Explanations

- **Call**

- After G66, specify at address P a program number subject to a modal call.
- When a number of repetitions is required, a number from 1 to 9999 can be specified at address L.
- As with a simple call (G65), data passed to a macro program is specified in arguments.

- **Cancellation**

When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.

- **Call nesting**

Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).

- **Modal call nesting**

Modal calls can be nested by specifying another G66 code during a modal call.

Limitations

- In a G66 block, no macros can be called.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a miscellaneous function that does not involve movement along an axis.
- Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

Sample program

The drilling cycle is created using a custom macro and the machining program makes a modal macro call. For program simplicity, all drilling data is specified using absolute values.

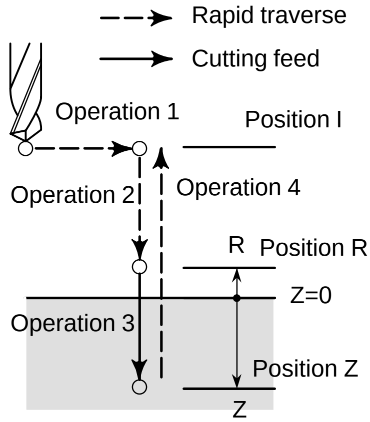
The canned cycle consists of the following basic operations:

Operation 1:
Positioning along the X–axis and Y–axis

Operation 2:
Rapid traverse to point R

Operation 3:
Cutting feed to point Z

Operation 4:
Rapid traverse to point R or I



• Calling format

```
G65 P9110 X x Y y Z z R r F f L l ;
```

X: X coordinate of the hole (absolute specification only) . . . . . (#24)
Y: Y coordinate of the hole (absolute specification only) . . . . . (#25)
Z : Coordinates of position Z (absolute specification only) . . . . (#26)
R: Coordinates of position R (absolute specification only) . . . . (#18)
F : Cutting feedrate (#9)
L : Repetition count

• Program that calls a macro program

```
O0001;
G28 G91 X0 Y0 Z0;
G92 X0 Y0 Z50.0;
G00 G90 X100.0 Y50.0;
G66 P9110 Z–20.0 R5.0 F500;
G90 X20.0 Y20.0;
X50.0;
Y50.0;
X70.0 Y80.0;
G67;
M30;
```

• Macro program (program called)

```
O9110;
  #1=#4001; . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . Stores G00/G01.
  #3=#4003; . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . Stores G90/G91.
  #4=#4109; . . . . . . . . . . . . . . . . . . . . . . . . Stores the cutting feedrate.
  #5=#5003; . . . . . . . . . Stores the Z coordinate at the start of drilling.
  G00 G90 Z#18; . . . . . . . . . . . . . . . . . . . . . . Positioning at position R
  G01 Z#26 F#9; . . . . . . . . . . . . . . . . . . . . . . Cutting feed to position Z
  IF[#4010 EQ 98]GOTO 1; . . . . . . . . . . . . . . . . . . Return to position I
  G00 Z#18; . . . . . . . . . . . . . . . . . . . . . . . . . Positioning at position R
  GOTO 2;
N1 G00 Z#5; . . . . . . . . . . . . . . . . . . . . . . . . . Positioning at position I
N2 G#1 G#3 F#4; . . . . . . . . . . . . . . . . . . . Restores modal information.
  M99;
```

16.6.3 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 6050 = 81
```

Explanations

By setting a G code number from 1 to 9999 used to call a custom macro program (9010 to 9019) in the corresponding parameter (6050 to 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.

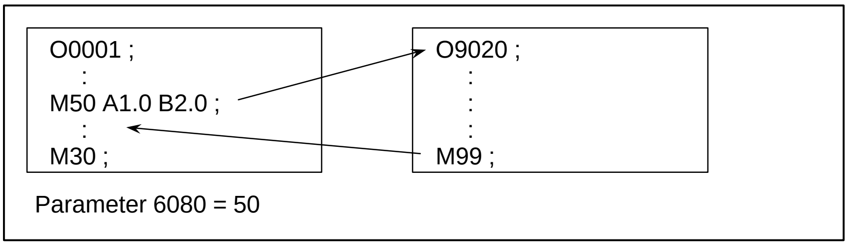
Limitations

- **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.

16.6.4 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).



Explanations

By setting an M code number from 1 to 99999999 used to call a custom macro program (9020 to 9029) in the corresponding parameters (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**

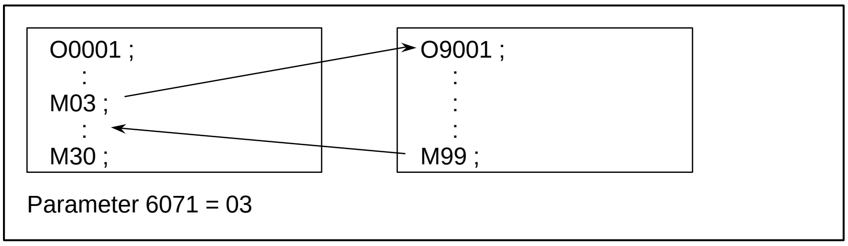
As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

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.

16.6.5 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).



Explanations

By setting an M code number from 1 to 99999999 used to call a subprogram in a parameters (No.6071 to No.6079), the corresponding custom macro program (9001 to 9009) 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

- **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**

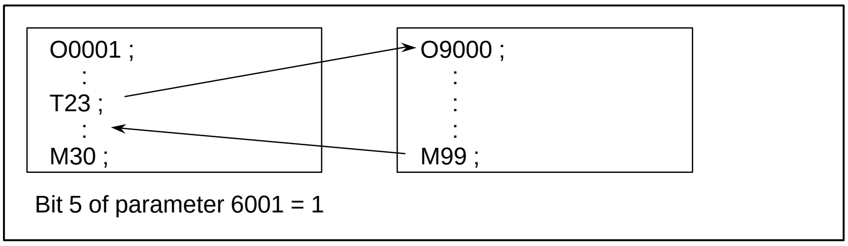
An M code in a macro program that has been called is treated as an ordinary M code.

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.

16.6.6 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 (TCS) of parameter 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.

16.6.7 Sample Program

By using the subprogram call function that uses M codes, the cumulative usage time of each tool is measured.

Conditions

- The cumulative usage time of each of tools T01 to T05 is measured. No measurement is made for tools with numbers greater than T05.
- The following variables are used to store the tool numbers and measured times:

#501	Cumulative usage time of tool number 1
#502	Cumulative usage time of tool number 2
#503	Cumulative usage time of tool number 3
#504	Cumulative usage time of tool number 4
#505	Cumulative usage time of tool number 5

- Usage time starts being counted when the M03 command is specified and stops when M05 is specified. System variable #3002 is used to measure the time during which the cycle start lamp is on. The time during which the machine is stopped by feed hold and single block stop operation is not counted, but the time used to change tools and pallets is included.

Operation check

● Parameter setting

Set 3 in parameter No.6071, and set 5 in parameter No.6072.

● Variable value setting

Set 0 in variables #501 to #505.

● Program that calls a macro program

```
O0001;
T01 M06;
M03;
G04 X20.0;
M05; ..................... Changes #501.
T02 M06;
M03;
G04 X20.0;
M05; ..................... Changes #502.
T03 M06;
M03;
G04 X20.0;
M05; ..................... Changes #503.
T04 M06;
M03;
G04 X20.0;
M05; ..................... Changes #504.
T05 M06;
M03;
G04 X20.0;
M05; ..................... Changes #505.
M30;
```

Macro program (program called)

```
O9001(M03);  . . . . . . . . . . . . . . . . . . . . . . . . . . Macro to start counting
  M01;
  IF[#4120 EQ 0]GOTO 9; . . . . . . . . . . . . . . . . . . . . . . No tool specified
  IF[#4120 GT 5]GOTO 9; . . . . . . . . . . . . .  Out–of–range tool number
  #3002=0;  . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . Clears the timer.
N9 M03;  . . . . . . . . . . . . . . Rotates the spindle in the forward direction.
  M99;

O9002(M05);  . . . . . . . . . . . . . . . . . . . . . . . . . . . Macro to end counting
  M01;
  IF[#4120 EQ 0]GOTO 9; . . . . . . . . . . . . . . . . . . . . . . No tool specified
  IF[#4120 GT 5]GOTO 9; . . . . . . . . . . . . .  Out–of–range tool number
  #[500+#4120]=#3002+#[500+#4120];  . . .  Calculates cumulative time.

N9 M05; . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . .  Stops the spindle.
  M99;
```

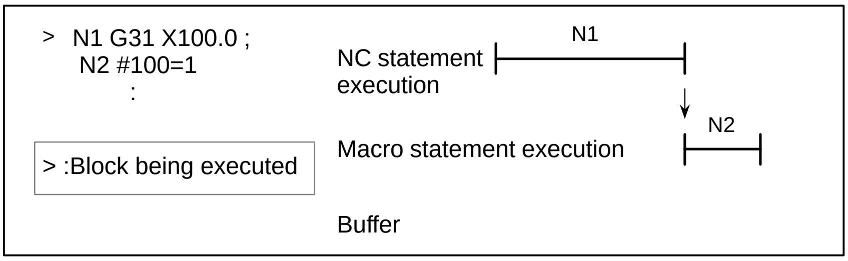
16.7 PROCESSING MACRO STATEMENTS

For smooth machining, the NC prereads the NC statement to be performed next. This operation is referred to as buffering. In cutter compensation mode (G41, G42), the NC prereads NC statements two or three blocks ahead to find intersections. Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer.

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.

16.7.1 Details of NC Statements and Macro Statements Execution

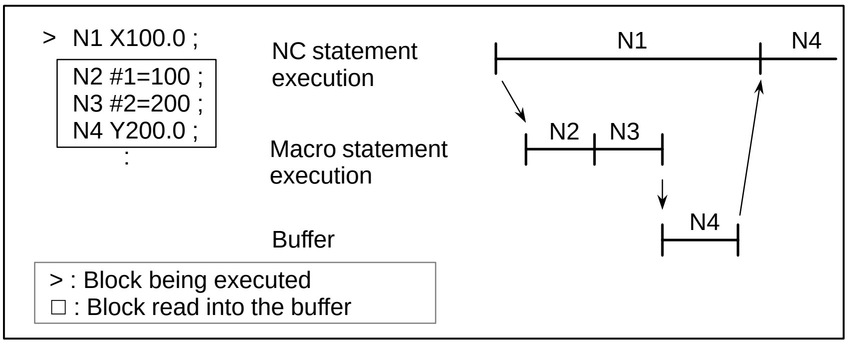
- **When the next block is not buffered (M codes that are not buffered, G53, G31, etc.)**



CAUTION

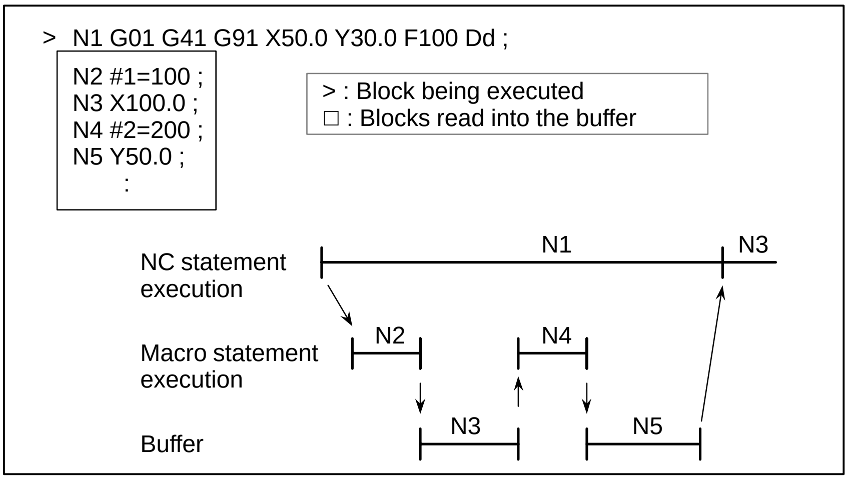
In case that you need to execute the macro statement after completing the block just before the macro statement, specify M code or G code that are not buffered just before the macro statement. Specially, in case of reading / writing the system variables to control signals, coordinates, offset value, 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)**



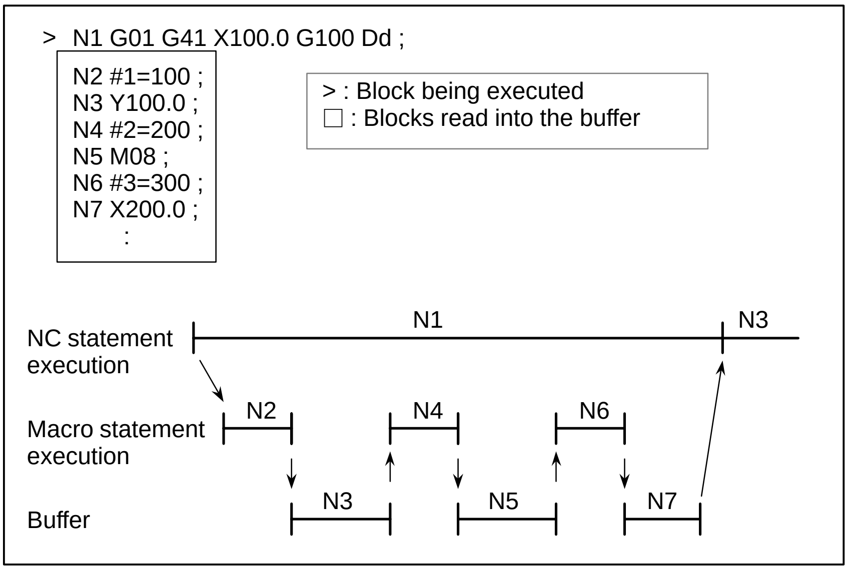
When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- **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 N1 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.

16.7.2 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.
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 – #5024	The uncertain position in moving is read.
Current position (Workpiece coordinate system)	Read	#5041 – #5044	The uncertain position in moving is read.
Deviated servo position	Read	#5101 – #5104	The uncertain deviation in moving is read.
Workpiece zero point offset value	Write	#5201 – #5324	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.

16.8 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.

16.9 LIMITATIONS

- **MDI operation**

The macro call command can be specified in MDI mode. During automatic operation, however, it is impossible to switch to the MDI mode for a macro program call.

- **Sequence number search**

A custom macro program cannot be searched for a sequence number.

- **Single block**

Even while a macro program is being executed, blocks can be stopped in the single block mode (except blocks containing macro call commands, arithmetic operation commands, and control commands).
A block containing a macro call command (G65, G66, or G67) does not stop even when the single block mode is on. Blocks containing arithmetic operation commands and control commands can be stopped in single block mode by setting SBKM (bit 5 of parameter 6000) to 1.
Single block stop operation is used for testing custom macro programs. When SBKM (bit 5 of parameter 6000) is set to 1, a single block stop takes place at every macro statement.
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**

Registered custom macro programs and subprograms should be protected from being destroyed by accident. 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. 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**

When memory is cleared with a reset operation, local variables and common variables #100 to #199 are cleared to null values. They can be prevented from being cleared by setting, CLV and CCV (bits 7 and 6 of parameter 6001). System variables #1000 to #1133 are not cleared.
A reset operation clears any called states of custom macro programs and subprograms, and any DO states, and returns control to the main program.

- **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.

16.10 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 NC outputs a DC2 control code.

• Data output command BPRNT

BPRNT [a #b [c] …]

c: Number of significant decimal places
#b: Variable
a: Character

The BPRNT command outputs characters and variable values in binary.

(i) Specified characters are converted to corresponding ISO codes according to the setting (ISO) that is output at that time.
Specifiable characters are as follows:
- **Letters (A to Z)**
- **Numbers**
- **Special characters (*, /, +, –, etc.)**

An asterisk (*) is output by a space code.

(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2–word (32–bit) data, including the decimal digits. It is output as binary data starting from the highest byte.

(iii) When specified data has been output, an EOB code is output according to the ISO code settings on the parameter screen.

(iv) Null variables are regarded as 0.

Example)

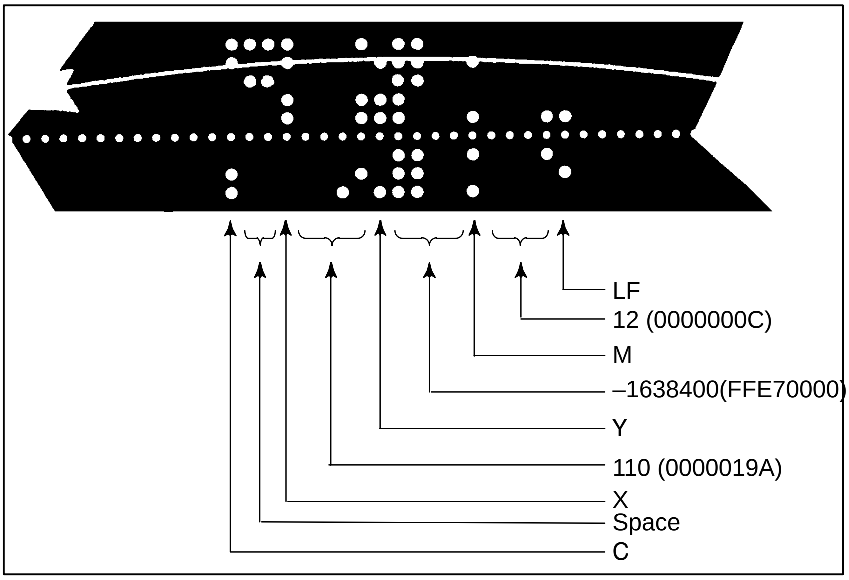
BPRINT [C** X#100 [3] Y#101 [3] M#10 [0]]

Variable value

#100=0.40596

#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 PRT is 0, a space code is output each time a zero is encountered.
When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output.
When PRT (bit 1 of parameter 6001) is 0, a space code is output to indicate a positive number instead of +; if PRT is 1, no code is output.

Example)

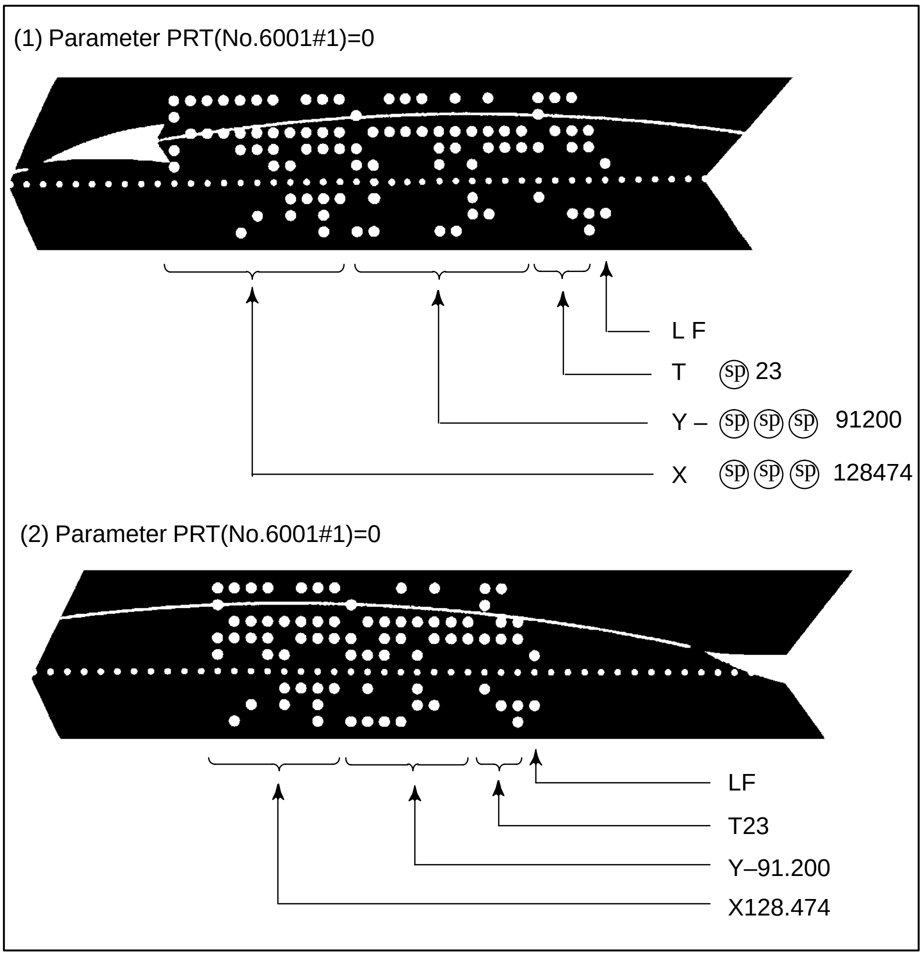
DPRINT [X#2 [53] Y#5 [53] T#30 [20]]

Variable value

#2=128.47398

#5=–91.2

#30=123.456



- **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 parameter 020. According to the specification of this parameter, set data items (such as the baud rate) for the reader/punch interface.

I/O channel 0 : Parameters 101 and 103
I/O channel 1 : Parameters 111 and 113
I/O channel 2 : Parameters 121 and 123

Specify parameter 102, 112 or 122 so that the reader/punch interface is used as the output device for punching. (Never specify output to the Fanuc Cassette or floppy disks.)

When specifying a DPRNT command to output data, specify whether leading zeros are output as spaces (by setting PRT (bit 1 of parameter 6001) to 1 or 0).

To indicate the end of a line of data in ISO code, specify whether to use only an LF (NCR, of bit 3 of parameter 0103 is 0) or an LF and CR (NCR 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.

16.11 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.

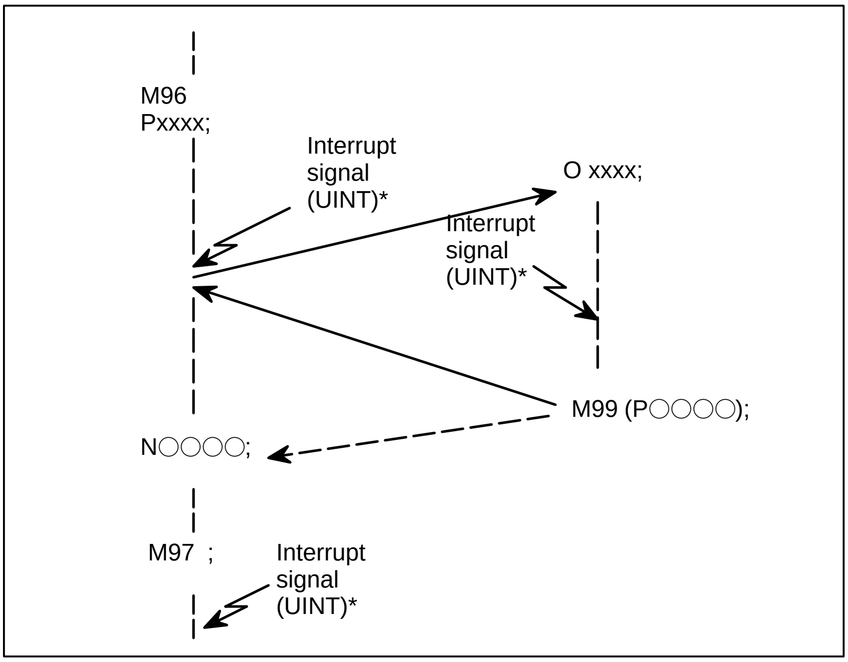


Fig. 16.11 Interruption type sustom macro function

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.

CAUTION
When the interrupt signal (UINT, marked by * in Fig. 16.11) is input after M97 is specified, it is ignored.
And, the interrupt signal must not be input during execution of the interrupt program.

16.11.1 Specification Method

Explanations

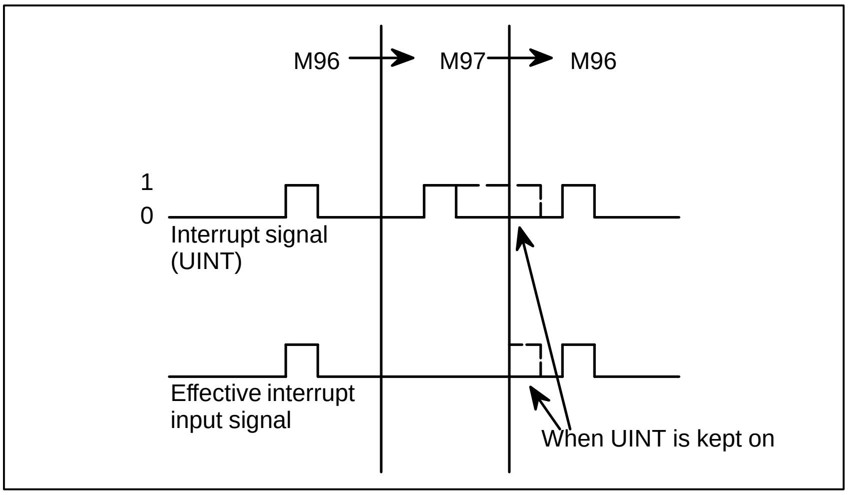
- **Interrupt conditions**

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 Subsec. 16.11.2.

16.11.2 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**

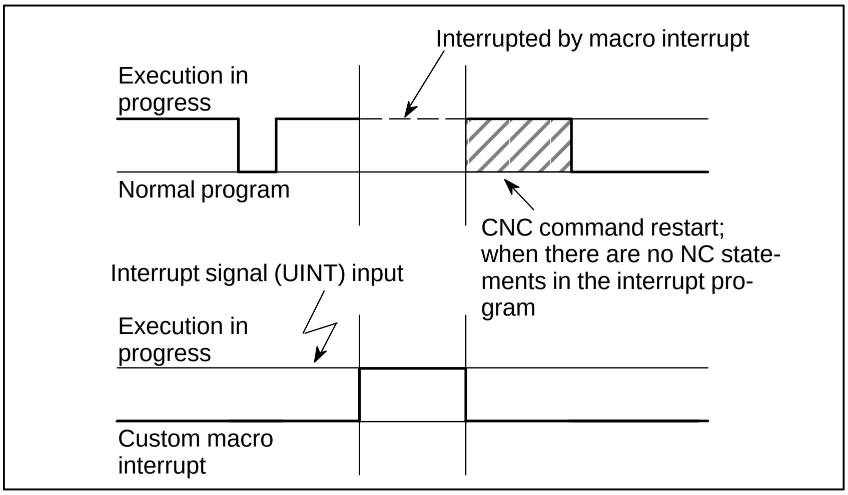
When performing a custom macro interrupt, the user may want to interrupt the NC statement being executed, or the user may not want to perform the interrupt until the execution of the current block is completed. MIN (bit 2 of parameter 6003)is used to select whether to perform interrupts even in the middle 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.

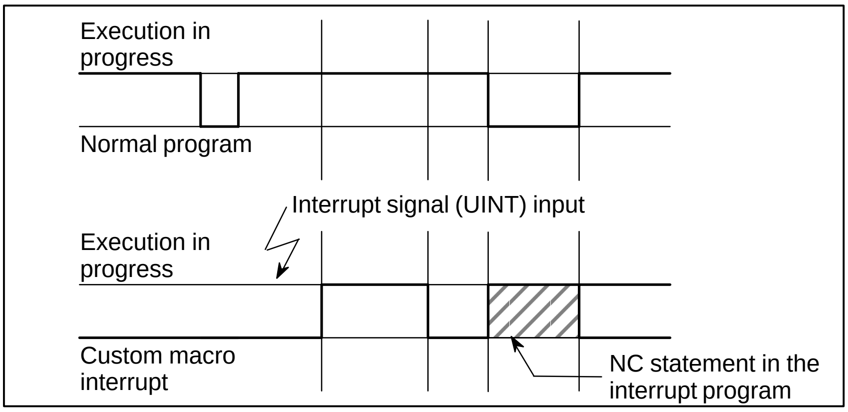
(iii) If there are no NC statements in the interrupt program, control is returned to the interrupted program by M99, then the program is restarted from the command in the interrupted block.



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 a pattern function and 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

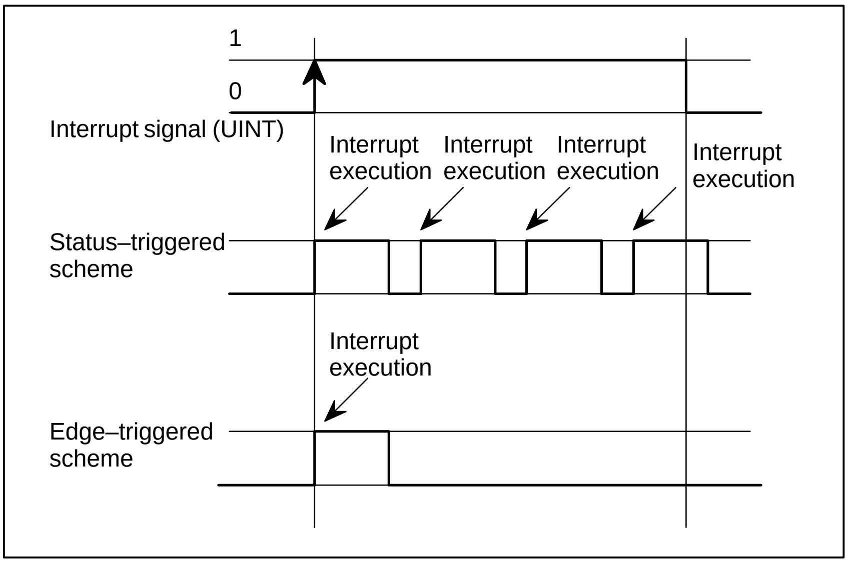
Even when cycle operation is in progress, movement is interrupted, and the interrupt program is executed. If the interrupt program contains no NC statements, the cycle operation is restarted after control is returned to the interrupted program. If there are NC statements, the remaining operations in 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)**

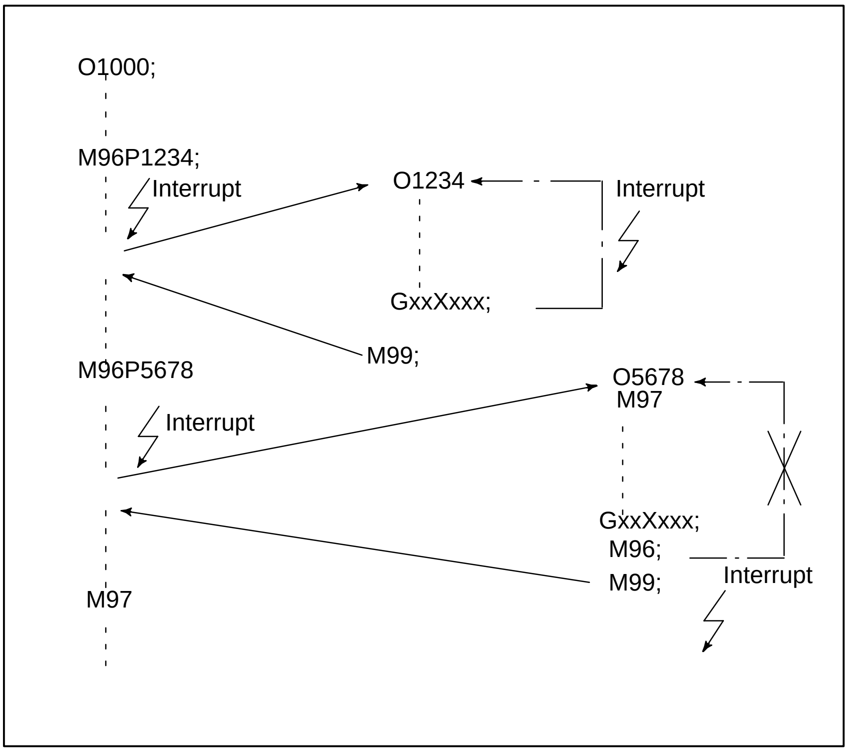
There are two schemes for custom macro interrupt signal (UINT) input: The status–triggered scheme and edge– triggered scheme. When the status–triggered scheme is used, the signal is valid when it is on. When the edge triggered scheme is used, the signal becomes valid on the rising edge when it switches 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 Gff is executed before M99 is recognized.)

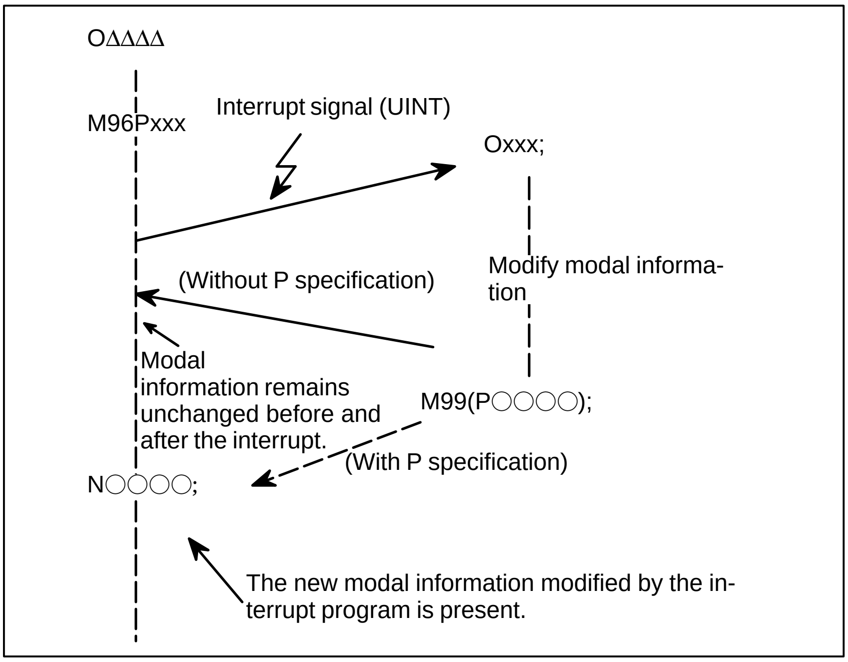
(1) G○○ X○○○ ;
M99 ;
(2) G○○ X○○○ M99 ;

- **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

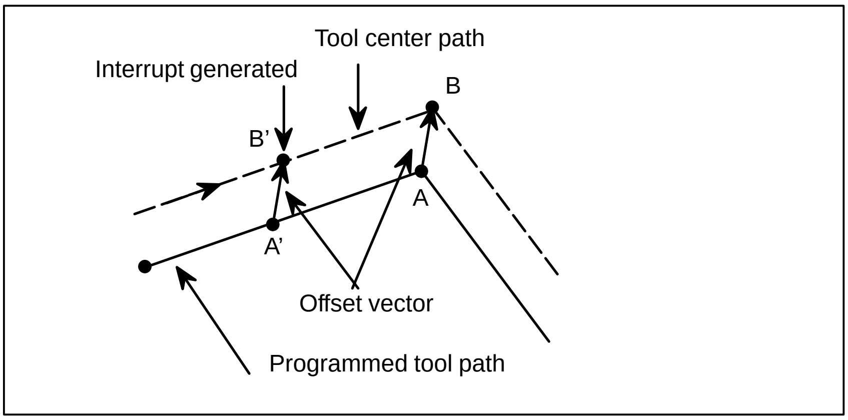
The modal information present before the interrupt becomes valid. The new modal information modified by the interrupt program is made invalid.

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.

- **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.

17 PROGRAMMABLE DATA ENTRY (G10)

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.

17.1 PROGRAMMABLE PARAMETER ENTRY

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. (5digits) or compensation posi tion No. for pitch errors compensation +10,000 (5digit)
R_: Parameter setting value (Leading zeros can be omitted.)
P_: Axis No. 1 to 4 (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 4 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
1 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.
2 For processing which reads multiple blocks in advance, such as that in cutter compensation mode, rewriting a parameter using G10 will cause that parameter to change during machining. In such a case, therefore, specify a G40 command (to cancel cutter compensation mode), before specifying G10.
3 In principle, specify G10 after stopping command distribution by, for example, specifying an M code which masks buffering.
If a PMC axis is operating while the machining program is being executed, specifying G10 in the machining program causes parameter rewriting to wait until command distribution to the PMC axis is stopped. In such a case, the execution of the machining program is also caused to wait.

NOTE
Other NC statements cannot be specified while in parameter input mode.

Examples

1. Set bit 2 (SPB) of bit type parameter No. 3404

G10L50 ;	Parameter entry mode
N3404 R 00000100 ;	SBP setting
G11 ;	cancel parameter entry mode

2. Change the values for the Z–axis and A–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

17.2 TOOL DATA ENTRY

Format

```
G10L30;  Tool data entry mode setting
N_P_R_;  Tool data entry
  :
  :
G11;     Tool data entry mode cancel
```

Meaning of command

N_ : Tool data No. or multiple tool data No. +200
P01 : Tool No. or multi–tool No. setting
P02 : Turret position or angle for indexing turret of multiple tool setting
P03 : X–axis tool offset setting
P04 : Y–axis tool offset setting
P05 : Tool change No. setting
P06 : Punch count setting
P08 : Tool figure setting for graphic operation
P09 : X dimension of a tool setting for graphic operation
P10 : Y dimension of a tool setting for graphic operation
P11 : Tool angle setting for graphic operation
R_ : Tool data setting value

Examples

Change the value of tool number for the tool data No. 10.

```
G10L30 ;        Tool data entry mode s
N10P01R333 ;    Modify tool number
G11 ;           Tool data entry mode cancel
```

18 AXIS CONTROL FUNCTIONS

18.1 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 1008 to 1.

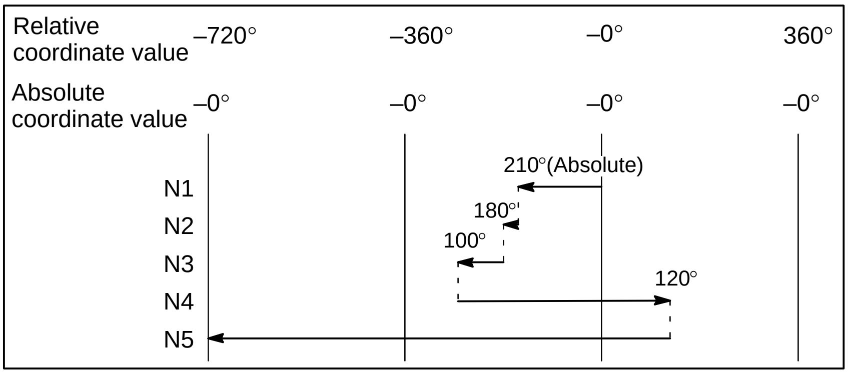
Explanations

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 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 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.

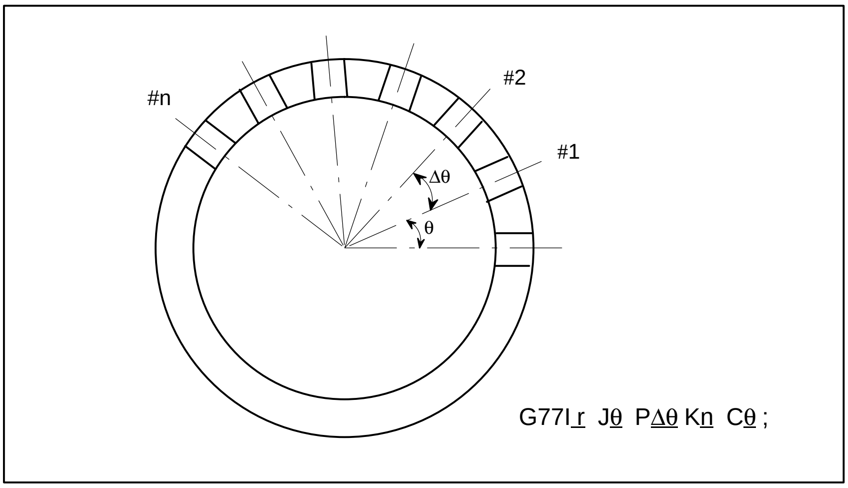
G90 A0 ;	Sequence number	Actual movement value	Absolute coordinate value after movement end
N1 G90 A–150.0 ;	N1	–150	210
N2 G90 A540.0 ;	N2	–30	180
N3 G90 A–620.0 ;	N3	–80	100
N4 G91 A380.0 ;	N4	+380	120
N5 G91 A–840.0 ;	N5	–840	0



WARNING
This function cannot be used together with the T–axis control and C–axis control.

18.2 C AXIS CONTROL (DIE ANGLE INDEXING)

For predetermined dies (tools) on a turret, the angular position of the die can be changed with a command from a tape, a memory or MDI.
In the past, it was necessary to use many dies when the die shape is the same but the die arrangement is different. Even in such a case, this new function makes it possible to conduct the operations only with one die since the function can freely change the angular position of the die. Further, since chamfering of four corners of a workpiece can be performed only with one die, the time required for die change is reduced, resulting in shorter machining time.
Further, in pattern function for a circular geometry bolt hole circle, arc pattern, and arc nibbling function, the C axis is automatically controlled so that one side of the die always faces the center of arc at each punching position.



There may be some restrictions of functions depending on the machine tools as shown in the following. Always refer to the manual issued by machine tool builder.

- Inhibition of simultaneous control of T and C axes in the manual mode.
- T axis can move only when C axis is at reference point (fixed position on the machine tool where the C axis can reach by the C axis manual reference point return.)

18.2.1 Simultaneously Controlled Axes

X, Y and T or X, Y and C when automatic operation.

18.2.2 Increment System

Least input increment IS-A : 0.01 deg, IS-B : 0.001 deg
Least command increment IS-A : 0.01 deg, IS-B : 0.001 deg

18.2.3 Maximum Programmable Dimension

IS-A : ±999999.99 deg
IS-B : ±99999.999 deg

18.2.4 Automatic Acceleration/ Deceleration

Linear acceleration/deceleration is possible in manual and automatic rapid traverse rate.

18.2.5 Manual Continuous Feed, Incremental Feed, Manual Reference Point Return

Specifications are the same as X, Y and T axes.

18.2.6 Relationship with Absolute/Incremental Command (G90/G91)

The C-axis command is always regarded as absolute regardless of designation of absolute command (G90) and incremental command (G91).
However, parameter No. 16363#0 can also be used to enable the G91 command.

18.2.7 Positioning in Smaller Angle Rotating Direction

When the C-axis command is designated, it is possible to automatically position the C axis from the present position to the commanded position in the smaller angle rotating direction. When the rotation angle is 180 deg, positioning occurs in the forward direction (CW or CCW according to the connections on the machine side) .

18.2.8 Blocks Where C–axis Command is Possible

C axis command can be specified in the following blocks:

(a) A block where no one shot G code exists.
However, a block with U, V, W or B command is excluded:
(Example)
X _ Y _ C _ ;

(b) G70 command
(Example)
G70X _ Y _ C _ ;

(c) Pattern function (Including nibbling function) G26, G45, G46, G47, G68, G69, G76, G77, G78, G79, G86, G87, G88, G89
(Example)
G26I _ J _ K _ C _ ;

(d) G01, G02 or G03 command in nibbling mode
(Example)
M12 ;
G01X _ Y _ Q _ C _ ;
X _ Y _ C _ ;
:
M13;

No C-axis command is allowed in a block other than above.
If C axis command is specified in a block other than above, no alarm will be generated. However, if a C-axis command (excluding the nibbling mode) is specified during the linear interpolation or the circular interpolation mode, an alarm (No. 4600) in generated.
Parameter No. 16360#5 can also be used to enable a C–axis command while in an interpolation mode.

18.2.9 C–axis Command and its Operation

In 18.2.8, the blocks which C-axis command can be performed were listed. However, unless the die (tool) which allows C-axis control has been selected, C-axis commands cannot be made. Therefore, if the die which does not allow C-axis control is selected, it is necessary to select a die which allows C-axis control before the C-axis command block or to select a die which allows C-axis control to the same block as the C-axis command with the T-command. In the following description of functions, <u>the die (tool) which allows C-axis control is assumed to be T15</u>. Namely, it is assumed that the axis transfer of the C-axis is possible only when die T15 has been selected. In the case of a machine which allows axis transfer of the T-axis only when the C-axis is located at the reference point (fixed location on the machine which can be reached by manual reference point return of C-axis).

(a) When there was T command
After the C-axis was positioned to the reference point, execution for the T-command is started.
Namely, after the <u>C CR</u>; (CR: setting value of parameter No. 1250, 1251) has been executed, the program command of the block where there was T-command is executed.

(b) G28 command (automatic reference point return)
After the C-axis has been positioned to the reference point, the X- and Y-axis reference point return is executed by the G28 command. The following shows the program examples of the C-axis command and its operation details:

(1) Sample program 1

```
⋮
N200  X_ Y_ T21;
N210  X_ Y_ C_ ;
```

In N210 block, C-axis control cannot be conducted since T15 is not selected.

(2) Sample program 2

```
⋮
N200  X_ Y_ T01;
N210  X_ Y_ T15C_ ;
```

In N210 block, C-axis motion starts when positioning of the X, Y and T axes is over and T-command completion signal FIN is returned. Press start signal PF is sent out when positioning of the C axis is over.
When the N210 block is programmed as follows:

```
N210T15C – 1;
```

C axis starts to move when T-axis positioning is over and T-command completion signal FIN is returned. Press start signal PF is not sent out after positioning is finished.
Namely,

(1) signal PF is sent out when there is the X or Y command, and
(2) signal PF is not sent out when there are no X and Y commands.

18.2.10 Pattern Function, Nibbling Function and C–axis Command

The C-axis command in blocks of G26 (Bolt Hole Circle), G76 (Line At Angle), G77 (Arc), G78/G97 (Grid), G86 (Shear Proof), G87 (Square), G68 (Nibbling Arc), G69 (Nibbling Lin), G88 (Radius) and G89 (Cut At Angle) are described below. Of these, the movement specifications by G26, G77, G68 and G88 concerning the circular geometry differ from those of the other functions.

(1) G45, G76, G78, G79, G86, G87, G69, G89

⋮

N200G72X_ Y_ ;

N210G28I_ J_ K_ T15C_ ;

In N210 block, T-axis positioning starts simultaneously when the X and Y axes start to move at the first positioning point. When the positioning of the X, Y and T axes is finished and T-command completion signal FIN is returned, positioning of the C axis starts.
After positioning of the C axis is over, press start signal PF is sent out, and punch operation is performed at the first positioning point. Thereafter, the X and Y axes are sequentially positioned following the commanded geometry without positioning of the C axis.

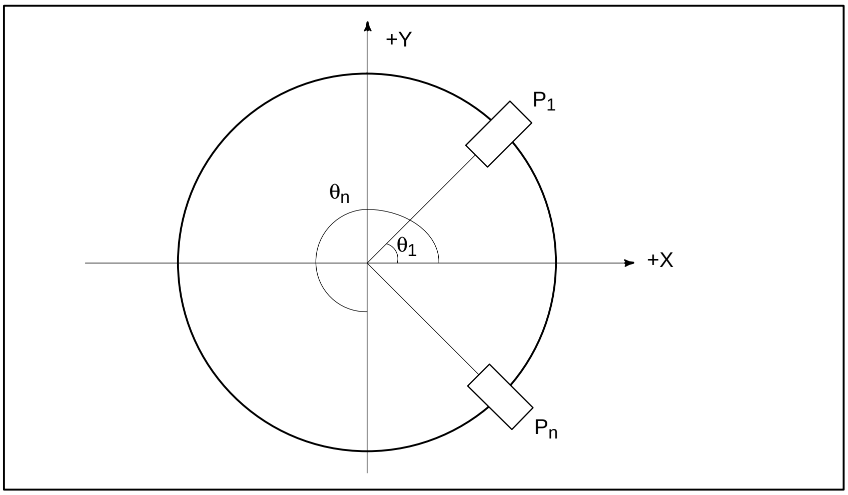
(2) G26, G46, G47, G77, G68, G88

⋮

N200G72X_ Y_ ;

N210G26I_ J_θ K_ T15C C1 ;

In N210 block, positioning to the first point is conducted in the same way as described in 18.2.10 (1). Thereafter, X and Y axes are sequentially positioned following the commanded geometry while the C axis positioning is conducted.



Pn : n-th positioning point

θn : Angle of n-th positioning point with respects to +X axis direction

Cn : C-axis position at n-th positioning point

Cn is calculated as follows:

Cn=C1+ (θn–θ1)

In case of G68, and alarm (No. 4522) occurs if the C-axis move value between adjacent positioning points exceeds the parameter value (No. 16194).

18.2.11 C–axis Command in Nibbling Mode

For the C-axis commands between the M code for nibbling mode and the M code for nibling mode cancel, an alarm is indicated if the C-axis movement amount per block exceeds the value set with the parameter (No. 16194).

18.2.12 T–axis Command Ignore Signal TNG and C–axis Command

The C-axis command is ignored when signal TNG is on.

18.2.13 Compensating the Position of the C–axis

This function automatically compensates the position of the C-axis to orient each tool in the specified direction. This simplifies mechanical adjustment such as adjusting the reference position when installing tools. Up to 20 compensation settings are specified in parameters No. 16430 to No. 16449. Up to 20 tool numbers are specified in parameters No. 16370 to No. 16389.
The C-axis is compensated when the tool moves around the C-axis after T commands have been specified.

WARNING
Tool numbers of tools must be specified with address T followed by up to four digits.

18.2.14 Compensating Backlash Along the C–axis for Each Tool Group

This function compensates backlash along the C-axis for each tool group. Changing the backlash compensation for each tool group enables higher-precision machining. Up to 20 compensation settings are specified in parameters No. 16390 to No. 16409. Up to 20 tool numbers are specified in parameters No. 16370 to No. 16389.

WARNING
The tool numbers must be specified with address T followed by up to four digits.

18.3 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.
The following operating modes are applicable to machines having two tables driven independently by separate control axes.
Synchronous control can be performed for up to two pairs of axes according to the parameter setting (parameter No. 8311).
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.

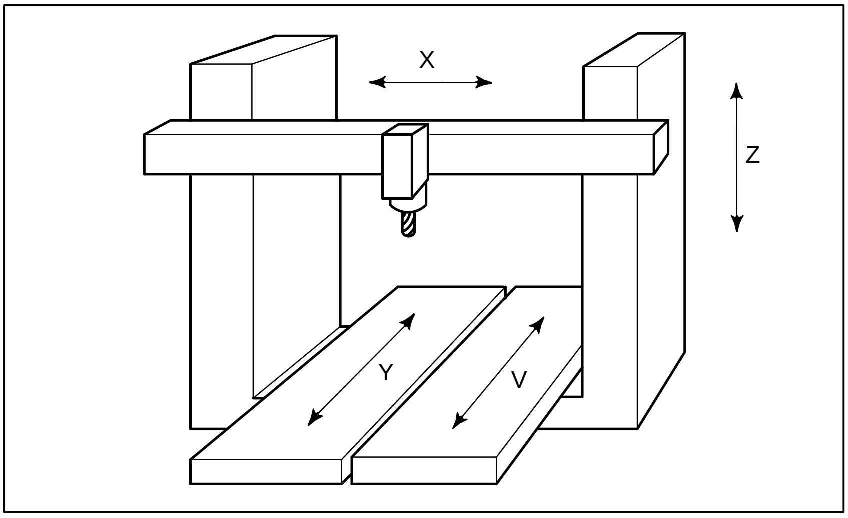


Fig. 18.3 (a) Example of axis configuration of the machine operated by simple synchronous control

Explanations

• Synchronous operation

This mode is used for 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, but is not possible during manual reference position return.

- **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 tool builder.

- **Automatic reference position return**

When the automatic reference position return command (G28) is 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 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.

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 (ABS = 1) 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 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 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.

18.4 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.

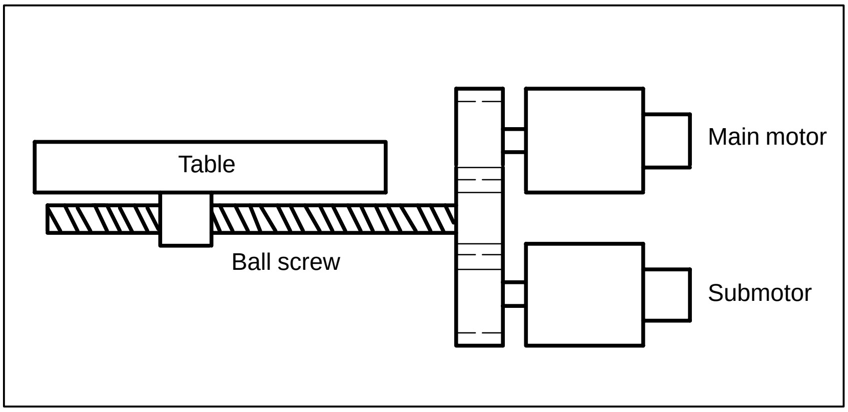


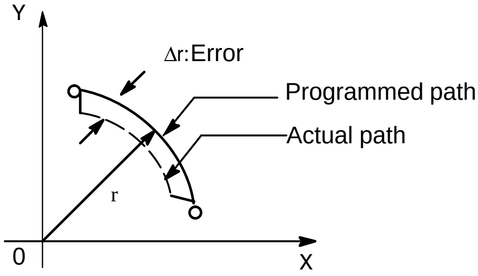
Fig. 18.4 Example of operation

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.

19 HIGH SPEED CUTTING FUNCTIONS

19.1 FEEDRATE CLAMPING BY ARC RADIUS

When an arc is cut at a high speed in circular interpolation, a radial error exists between the actual tool 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 Δr 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 tool builder.

19.2 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 tool 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 tool builder.

Format

G08 P_

P1 : Turn on advanced preview control mode.
P0 : Turn off advanced preview control mode.

Explanations

- **Available functions**

In advanced preview 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 descriptions of the functions.
For functions (1) and (2), specific parameters are provided.

- **Reset**

Advanced preview control mode is canceled by reset.

Limitations

- **G08 command**

Specify G08 code only in a block on the interpolation mode.

- **Functions that can be specified**

In the advanced preview control mode, the optional functions listed below can be specified.
It cannot be used for the function to specify punching.

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.

- 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 advanced preview control mode.)
- Helical interpolation
- 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. However, since setting this parameter to 1 changes the timing at which a signal is output, the ladder may need to be changed accordingly.)
- Custom macro B
- Inch/metric conversion
- Automatic corner override
  (Only the internal circular cutting feedrate change is valid.)
- Scaling
- Coordinate system rotation
- Workpiece coordinate system
- Cutter compensation C
- Graphic display

III. OPERATION

1 GENERAL

1.1 MANUAL OPERATION

Explanations

- **Manual reference position return**

The CNC machine tool has a position used to determine the machine position.
This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on, the tool is moved to the reference position.
Manual reference position return is to move the tool to the reference position using switches and pushbuttons located on the operator's panel (See Section III–3.1).

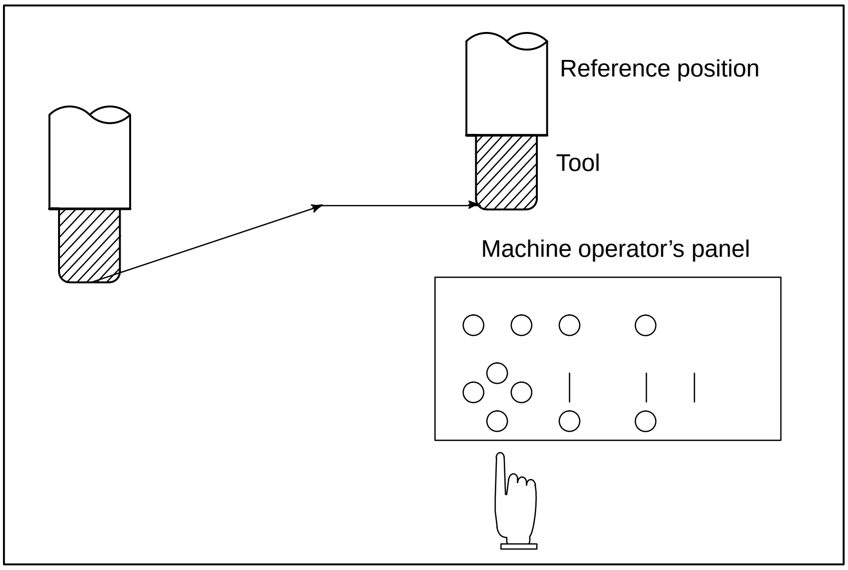


Fig. 1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.
This operation is called automatic reference position return (See Section II–6).

- **The tool movement by manual operation**

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

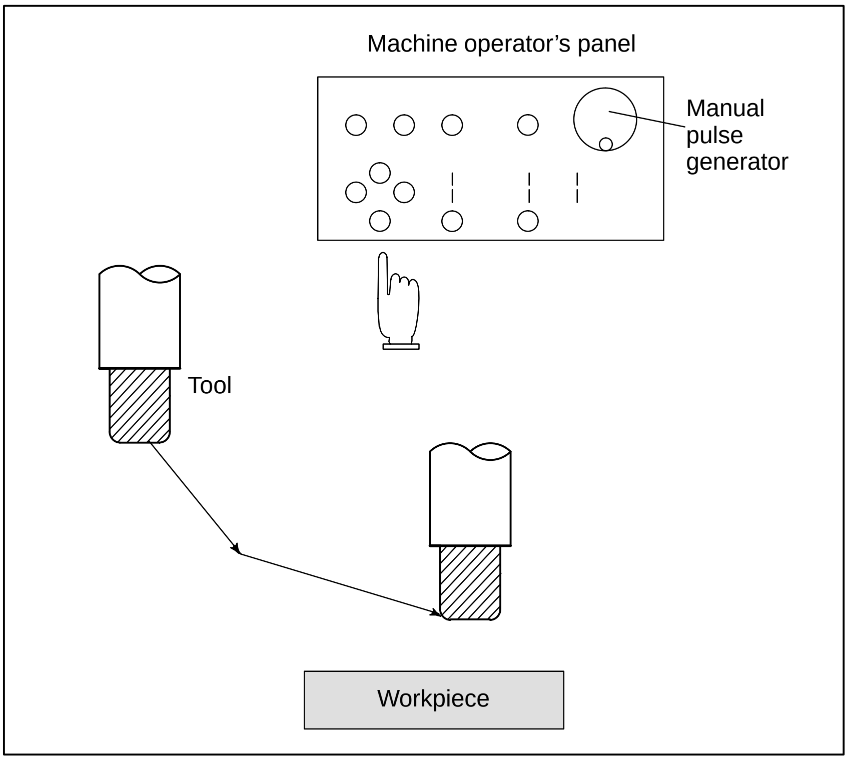


Fig. 1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

(i) Jog feed (See Section III–3.2)
The tool moves continuously while a pushbutton remains pressed.

(ii) Incremental feed (See Section III–3.3)
The tool moves by the predetermined distance each time a button is pressed.

(iii) Manual handle feed (See Section III–3.4)
By rotating the manual handle,the tool moves by the distance corresponding to the degree of handle rotation.

1.2 TOOL MOVEMENT BY PROGRAMMING– AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory and MDI operations. (See Section III–4).

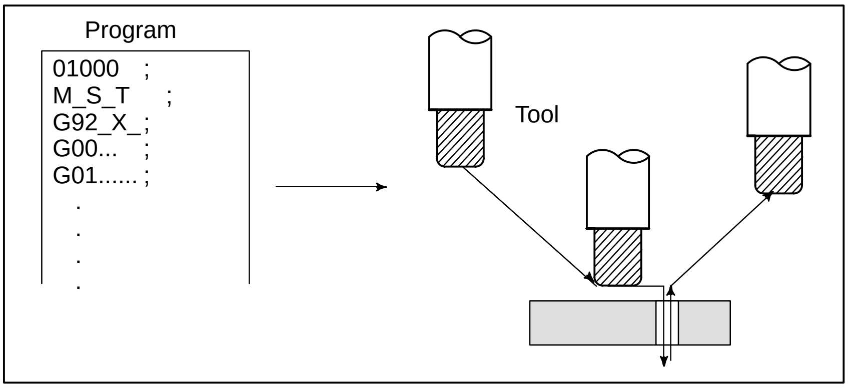


Fig. 1.2 (a) Tool movement by programming

Explanations

- **Memory operation**

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

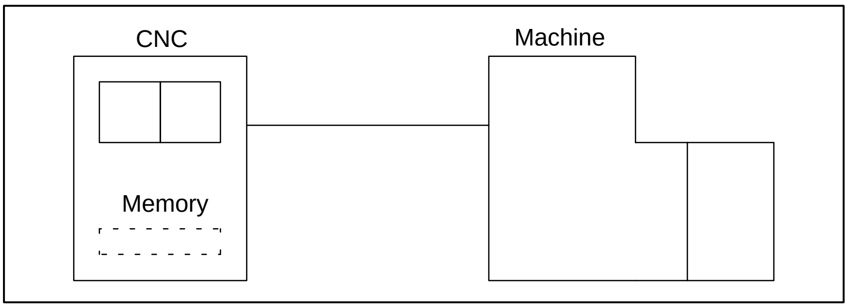


Fig. 1.2 (b) Memory operation

- **MDI operation**

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

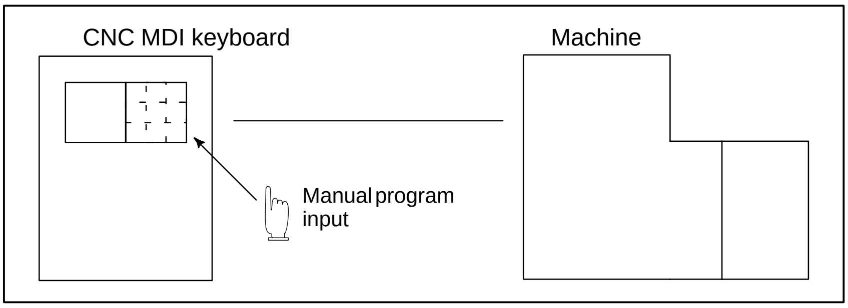


Fig. 1.2 (c) MDI operation

- **DNC operation**

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

1.3 AUTOMATIC OPERATION

Explanations

- **Program selection**

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

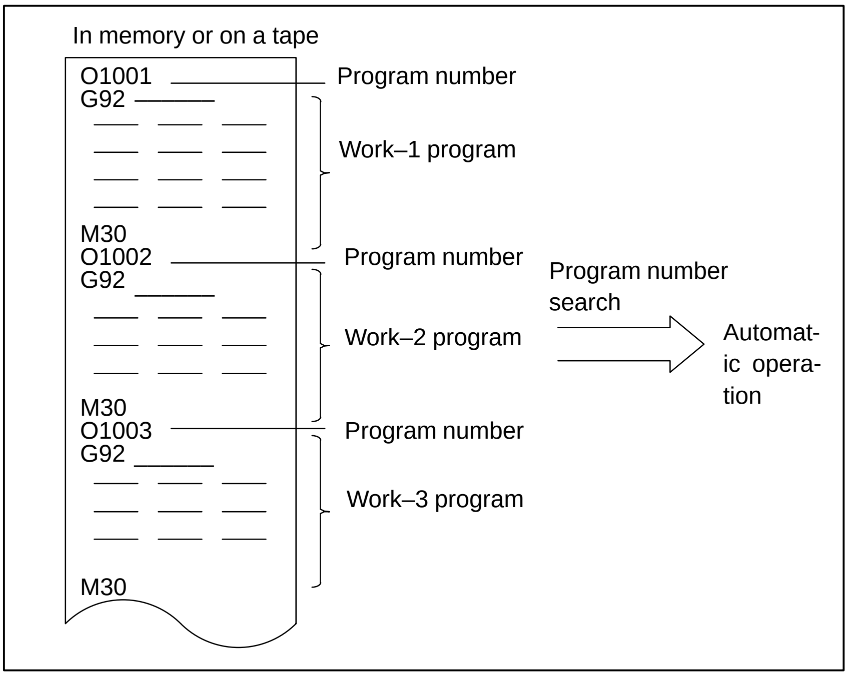


Fig. 1.3 (a) Program selection for automatic operation

- **Start and stop**

Pressing the cycle start pushbutton causes automatic operation to start. By pressing the feed hold or reset pushbutton, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops (See Section III–4).

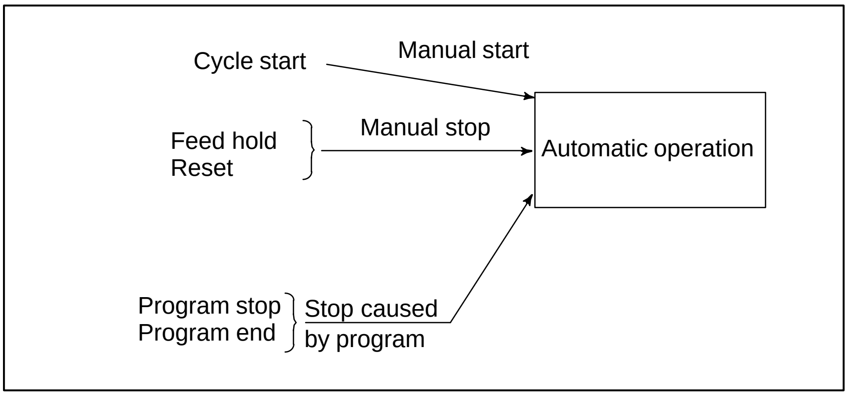


Fig. 1.3 (b) Start and stop for automatic operation

● Handle interruption

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle (See Section III–4.6).

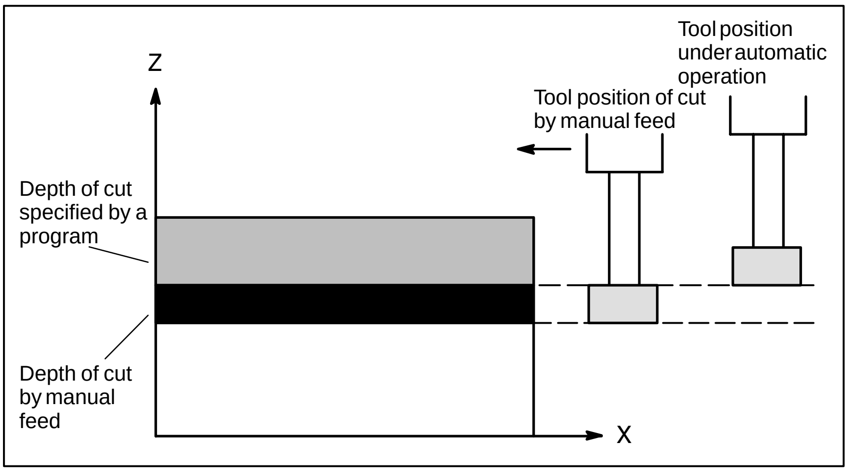


Fig. 1.3 (c) Handle interruption for automatic operation

1.4 TESTING A PROGRAM

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

1.4.1 Check by Running the Machine

Explanations

- **Dry run**

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

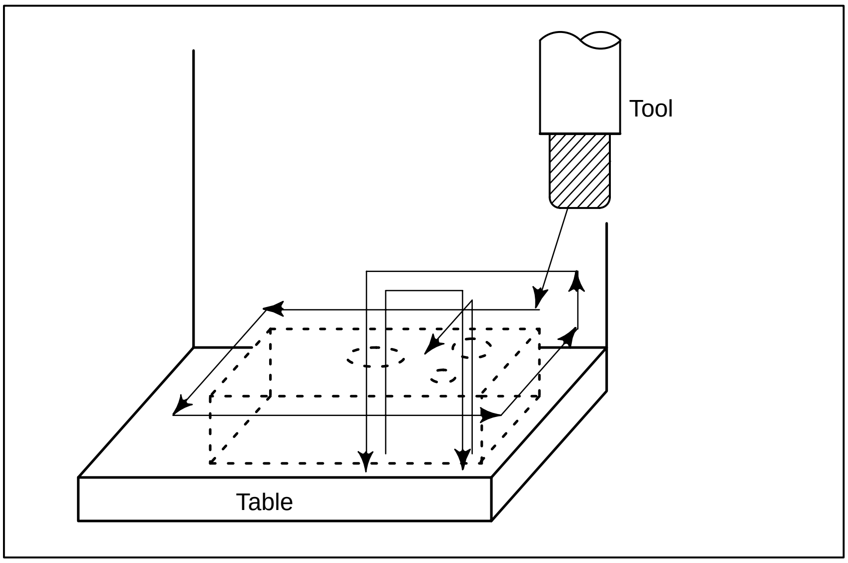


Fig. 1.4.1 (a) Dry run

- **Feedrate override**

Check the program by changing the rate specified in the program (See Section III–5.2).

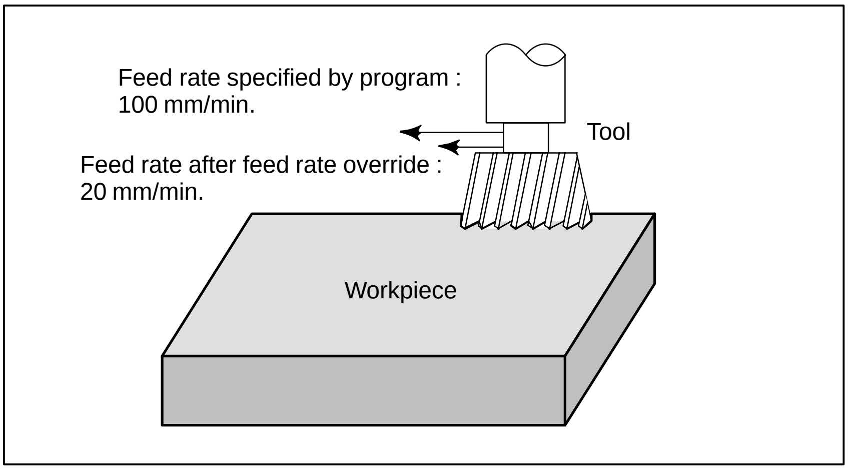


Fig. 1.4.1 (b) Feedrate override

- **Single block**

When the cycle start pushbutton is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner (See Section III–5.5).

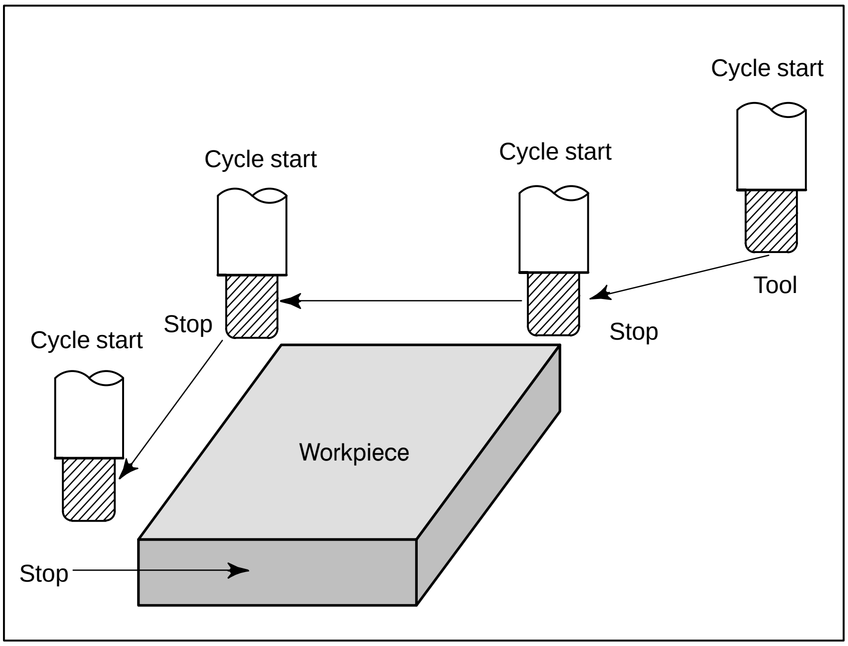


Fig. 1.4.1 (c) Single block

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

Explanations

- **Machine lock**

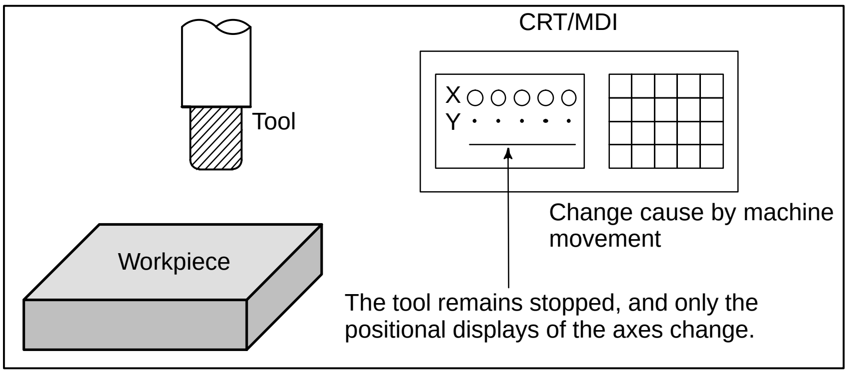


Fig. 1.4.2 Machine lock

- **Auxiliary function lock**

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

1.5 EDITING A PART PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the MDI panel (See Section III–9).
This operation can be executed using the part program storage/edit function.

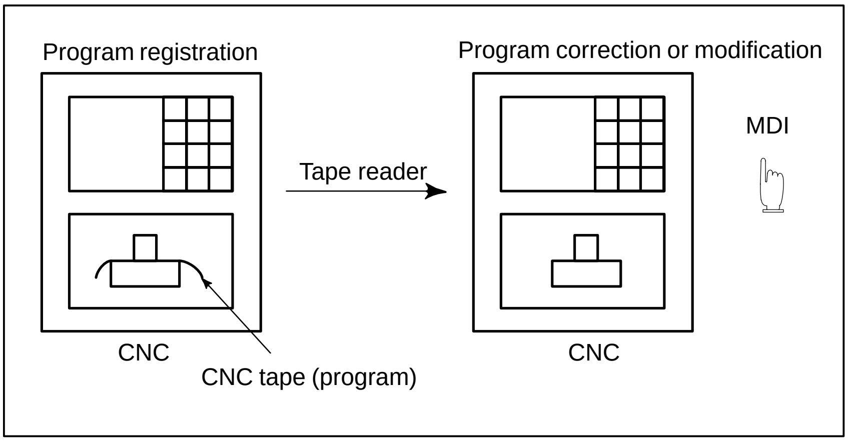


Fig. 1.5 Part program editing

1.6 DISPLAYING AND SETTING DATA

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

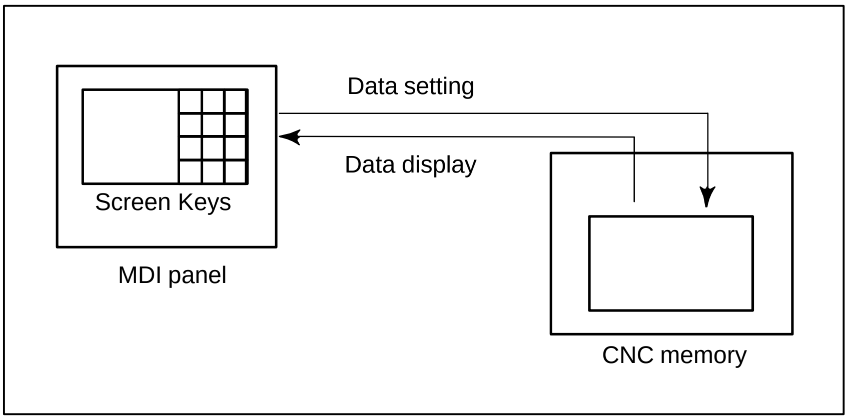


Fig. 1.6 (a) Displaying and setting data

Explanations

- **Offset value**

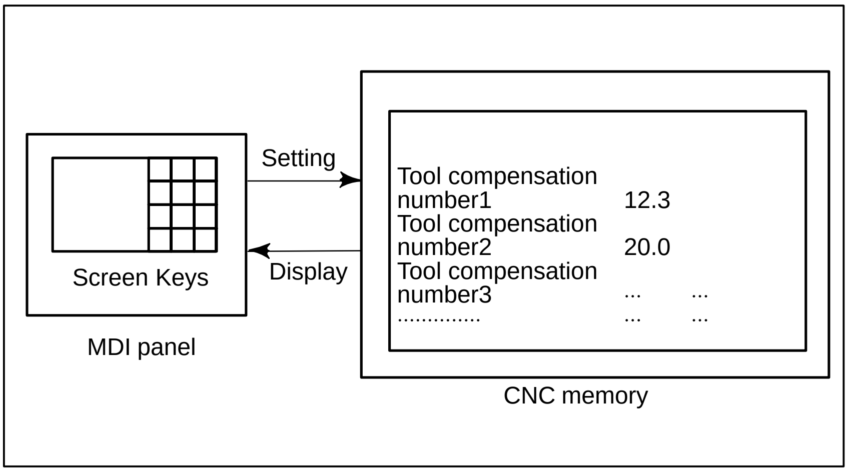


Fig. 1.6 (b) Displaying and setting offset values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement route depends on the tool dimensions.
By setting tool dimension data in CNC memory beforehand, automatically generates tool routes that permit any tool to cut the workpiece specified by the program. Tool dimension data is called the offset value (See Section III–11.4.1).

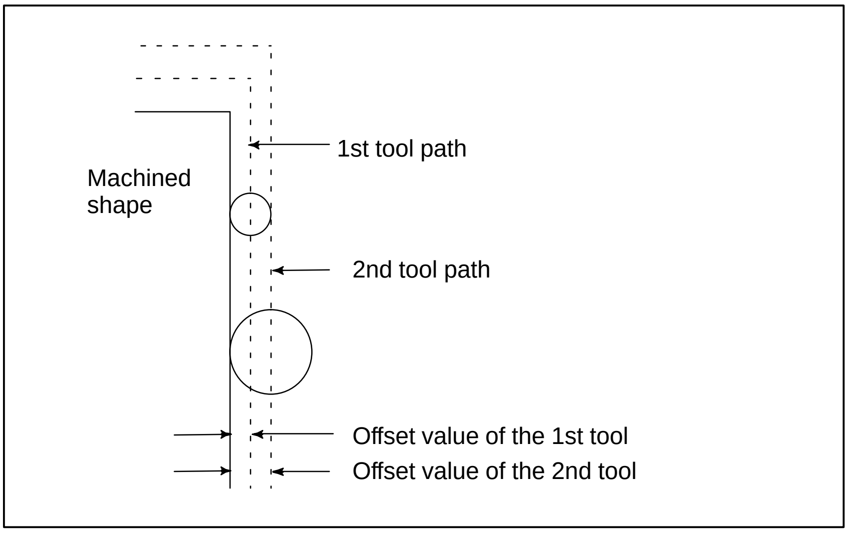


Fig. 1.6 (c) Offset value

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

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.
For example, the following data can be set:
·Inch/Metric switching
·Data related to I/O devices
·Mirror image cutting on/off
The above data is called setting data (See Section III–11.4.3).

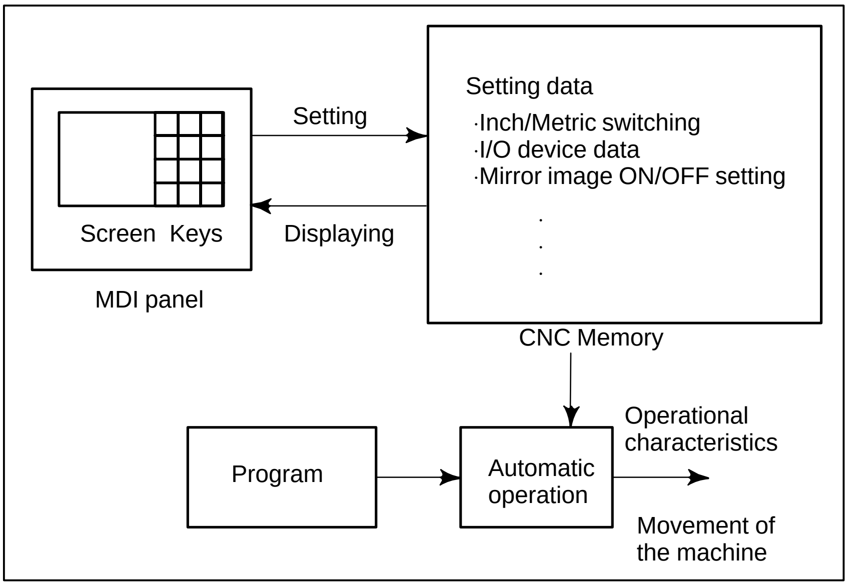


Fig. 1.6 (d) Displaying and setting operator's setting data

- **Displaying and setting parameters**

The CNC functions have versatility in order to take action in characteristics of various machines.
For example, CNC can specify the following:
·Rapid traverse rate of each axis
·Whether increment system is based on metric system or inch system.
·How to set command multiply/detect multiply (CMR/DMR)
Data to make the above specification is called parameters (See Section III–11.5.1).
Parameters differ depending on machine tool.

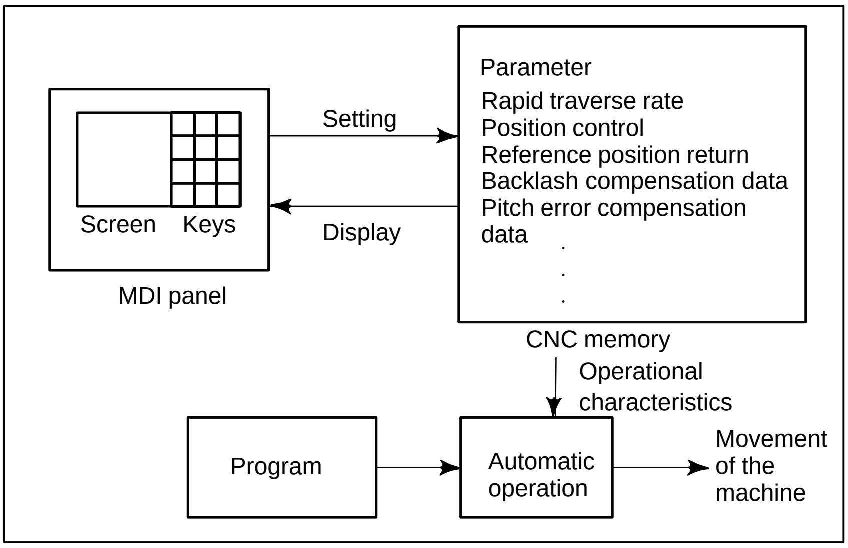


Fig. 1.6 (e) Displaying and setting parameters

- **Data protection key**

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

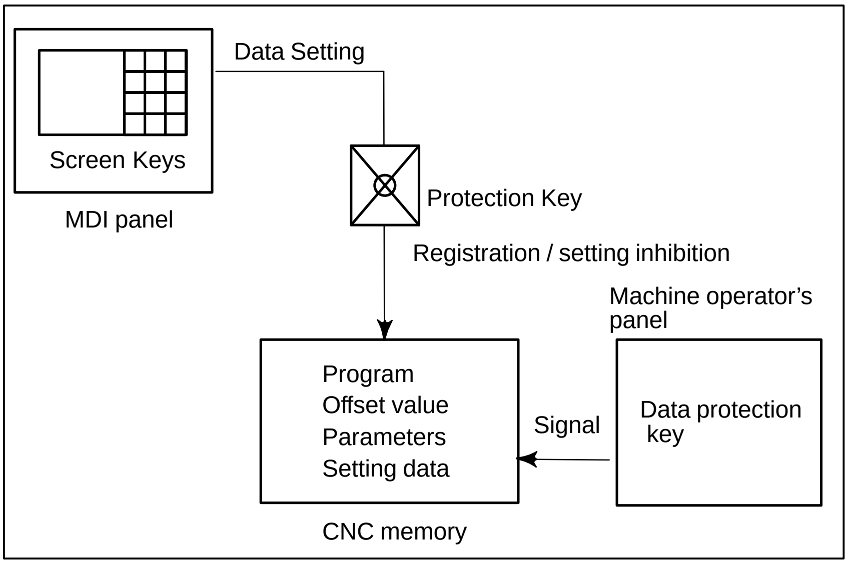
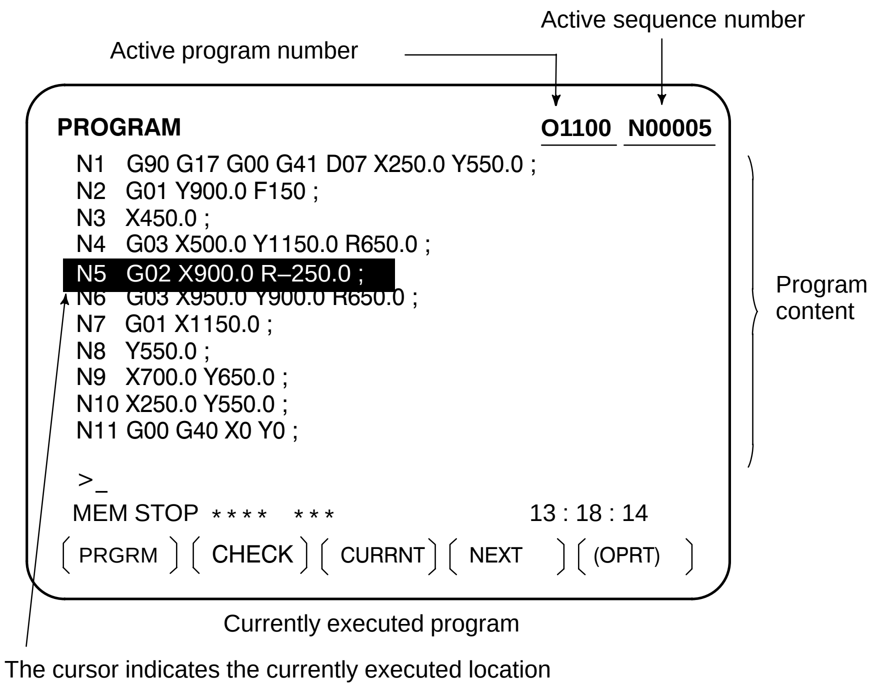


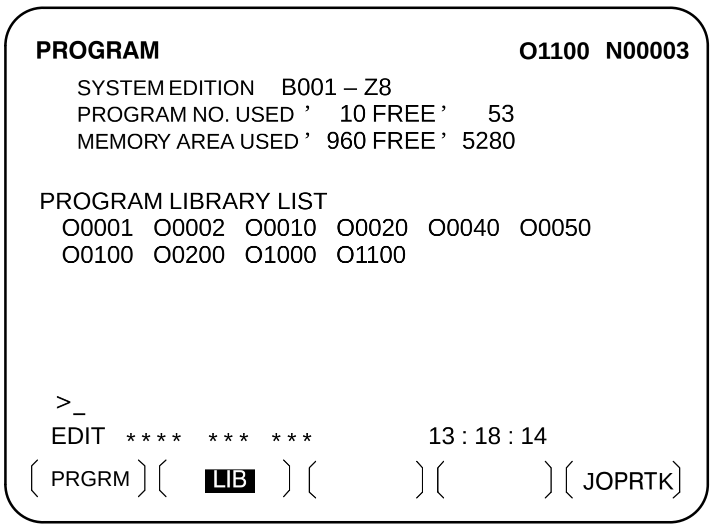
Fig. 1.6 (f) Data protection key

1.7 DISPLAY

1.7.1 Program Display

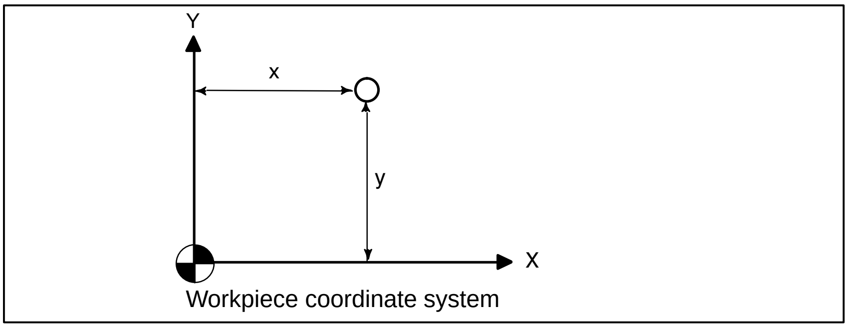
The contents of the currently active program are displayed. In addition, the programs scheduled next and the program list are displayed (See Section III–11.2.1).

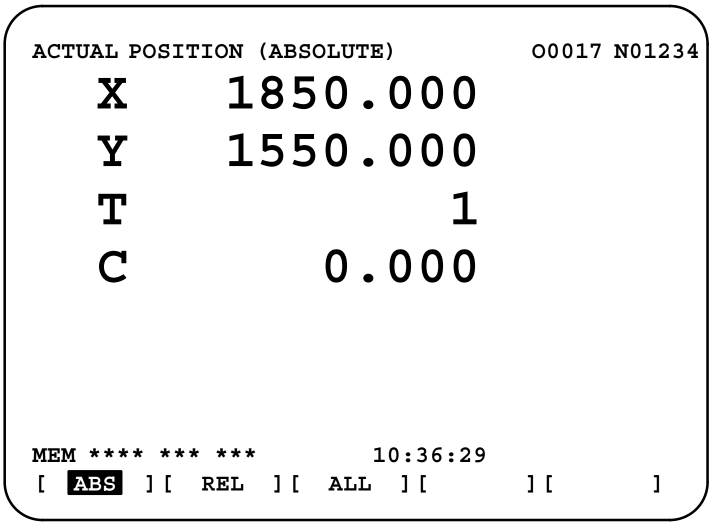




1.7.2 Current Position Display

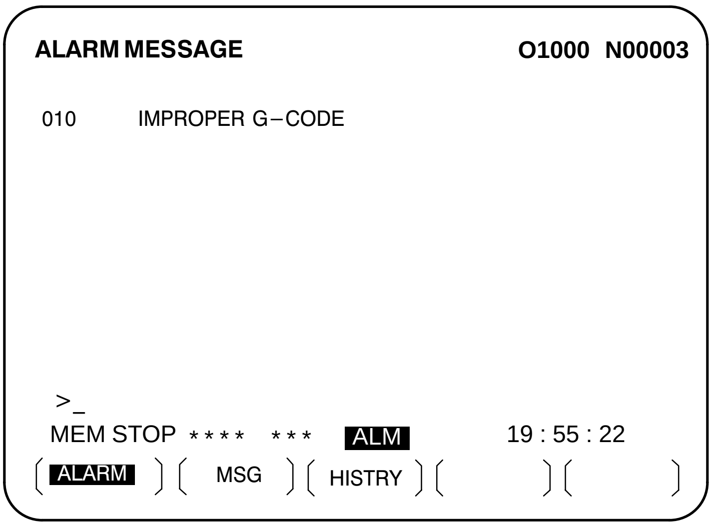
The current position of the tool is displayed with the coordinate values. The distance from the current position to the target position can also be displayed (See Section III–11.1 to 11.1.3).





1.7.3 Alarm Display

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



1.7.4 Parts Count Display, Run Time Display

When this option is selected, two types of run time and number of parts are displayed on the screen (See Section lll–11.4.6).

```
ACTUAL POSITION (ABSOLUTE)          O0017 N01234
   X    1850.000
   Y    1550.000
   T           1
   C       0.000

                     PART COUNT      493
RUN TIME      33H43M CYCLE TIME  0H 0M 0S

MEM **** *** ***       10:38:50
[ ABS  ][  REL  ][  ALL  ][      ][ (OPRT) ]
```

1.7.5 Graphic Display

Programmed tool movement can be displayed on the following planes (See Section III–12):

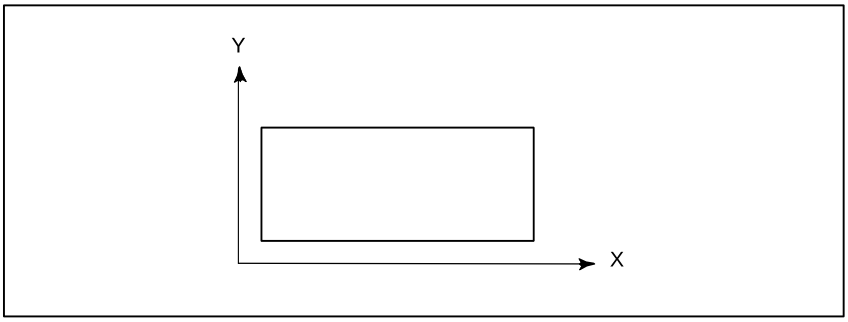


Fig. 1.7.5 Graphic display

1.8 DATA 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.

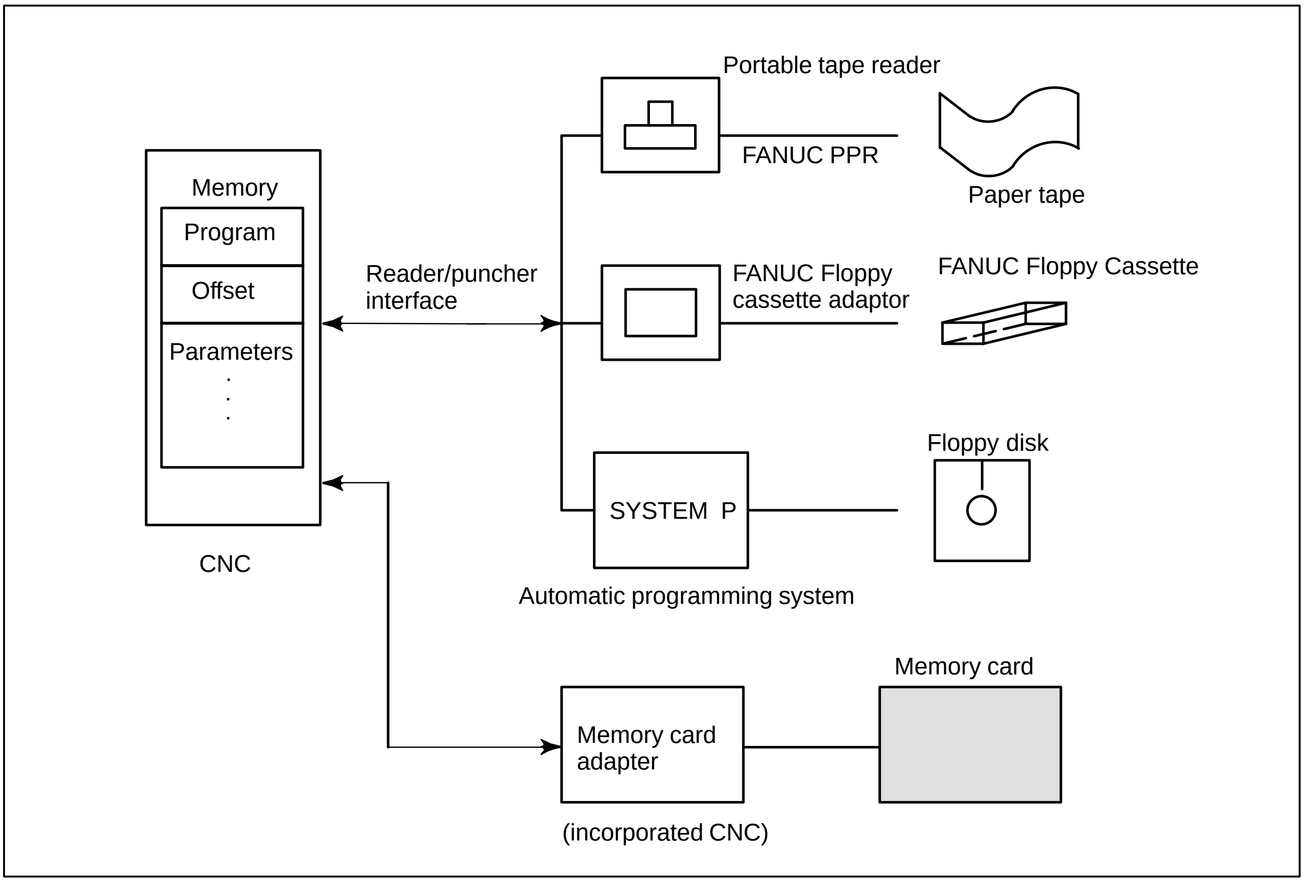


Fig. 1.8 Data Output

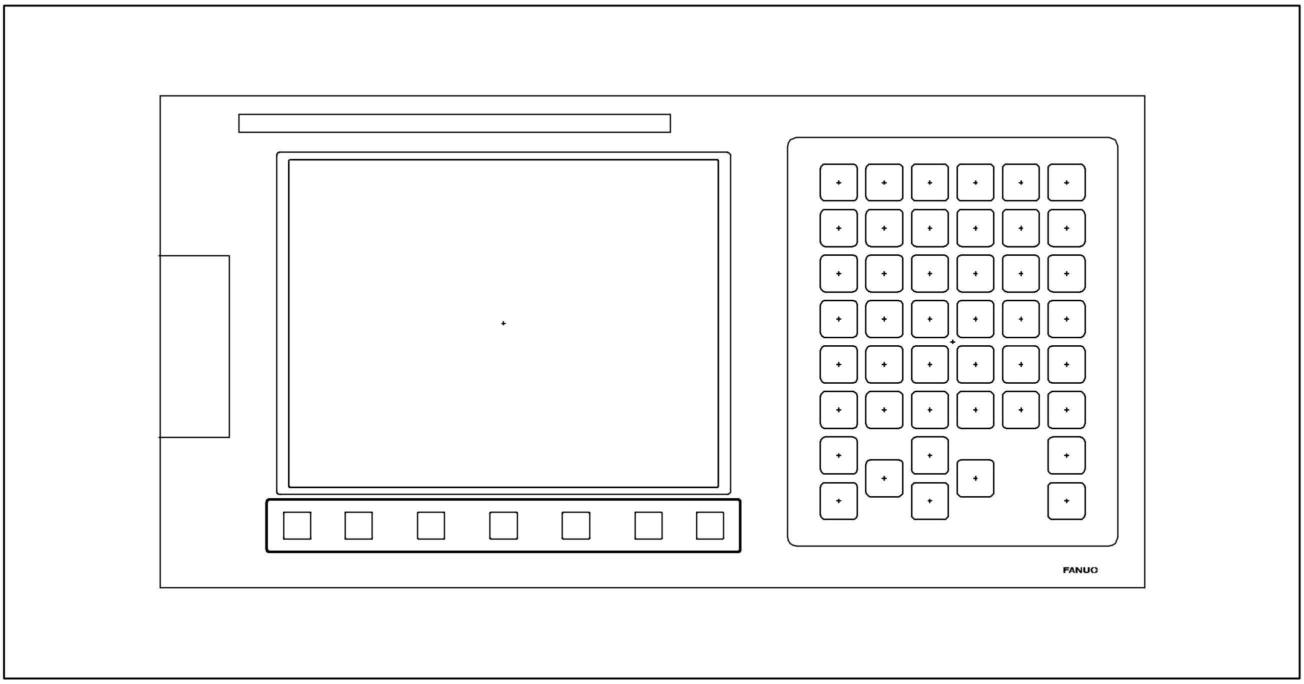
2 OPERATIONAL DEVICES

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.

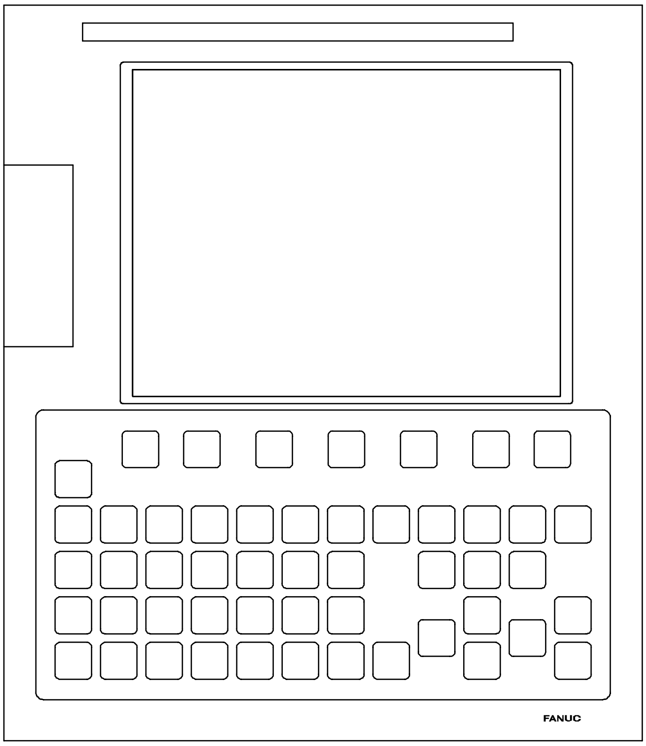
2.1 SETTING AND DISPLAY UNITS

The setting and display units are shown in Subsections 2.1.1 to 2.1.4 of Part III.

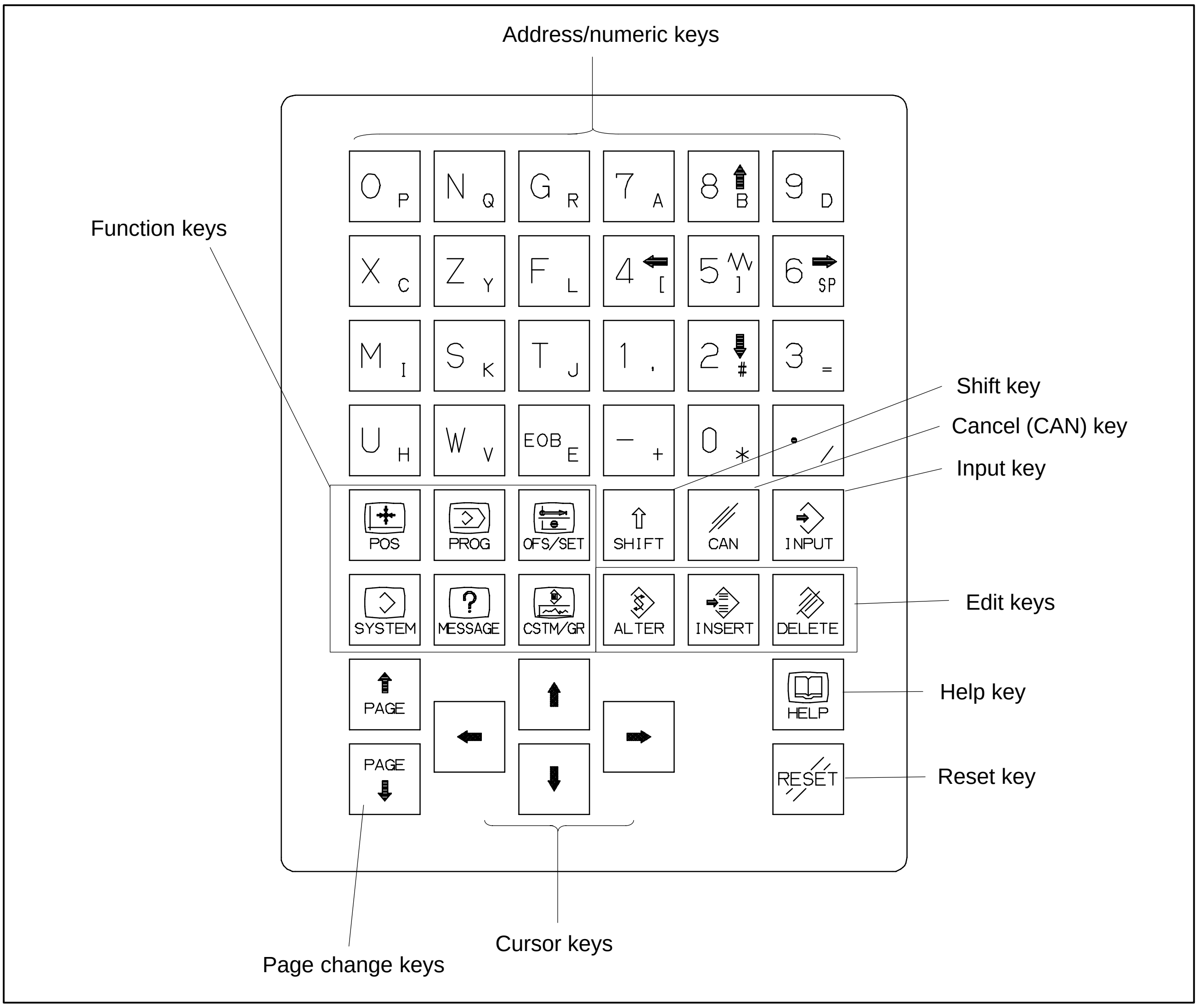
2.1.1
7.2″ Monochrome/ 8.4″ Color LCD/MDI Unit (Horizontal Type)



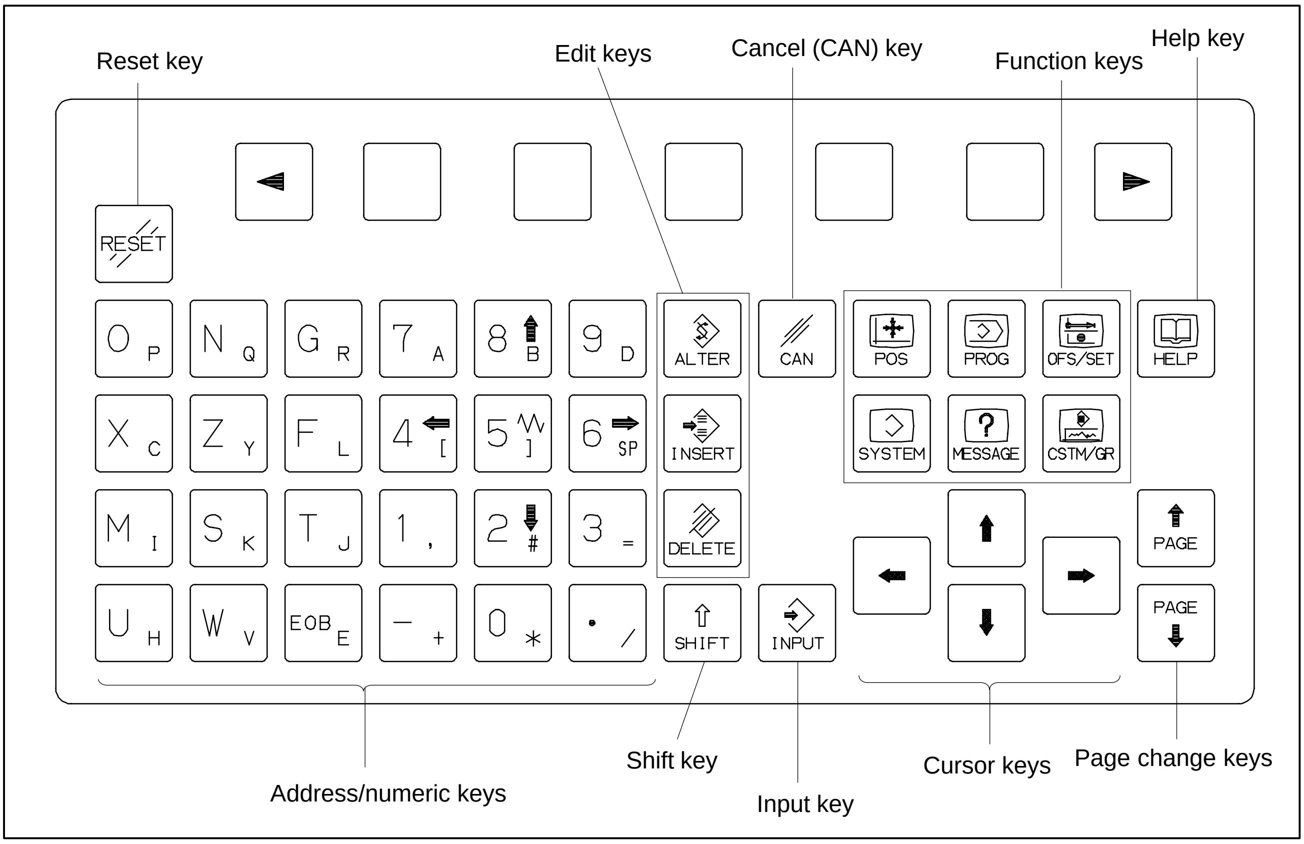
2.1.2 7.2″ Monochrome/ 8.4″ Color LCD/MDI Unit (Vertical Type)



2.1.3 Key Location of MDI (Horizontal Type LCD/MDI Unit)



2.1.4 Key Location of MDI (Vertical Type LCD/MDI Unit)

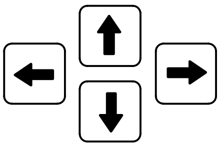
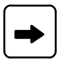
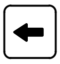
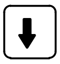


2.2 EXPLANATION OF THE KEYBOARD

Table2.2 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).
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 INSERT : Insertion DELETE : Deletion
9	Function keys POS PROG …	Press theses keys to switch display screens for each function. See lll – 2.3 for detailas of the function keys.

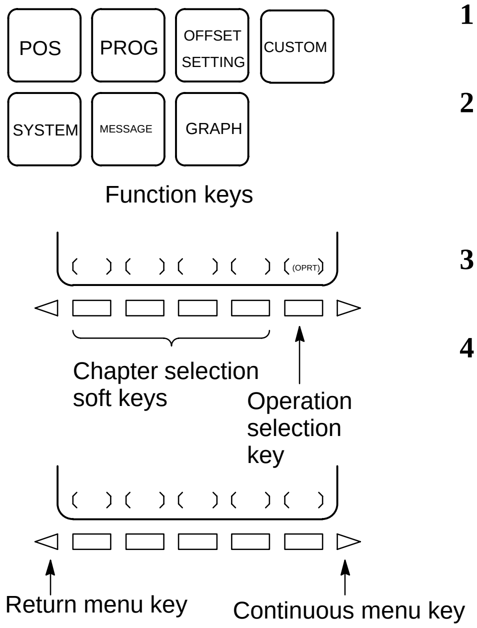
Table2.2 Explanation of the MDI keyboard

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 	Two kinds of page change keys are described below.  : This key is used to changeover the page on the screen in the forward direction.  : This key is used to changeover the page on the screen in the reverse direction.

2.3 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

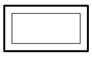
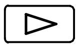
Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the MDI panel:

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**.
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 CNC with PC functions, this key is assigned to “Ctrl” key of the personal computer.
	In case of CNC with PC functions, 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

POS

- **Absolute coordinate display**
  - [ABS] — [(OPRT)]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]
- **Relative coordinate display**
  - [REL] — [(OPRT)]
    - (Axis or numeral) — [PRESET]
    - [ORIGIN]
      - [ALLEXE]
      - (Axis name) — [EXEC]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]
- **Current position display**
  - [ALL] — [(OPRT)]
    - (Axis or numeral) — [PRESET]
    - [ORIGIN]
      - [ALLEXE]
      - (Axis name) — [EXEC]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]
- **Handle interruption**
  - [HNDL] — [(OPRT)]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]

▷

- **Monitor screen**
  - [MONI] — [(OPRT)]
    - [PTSPRE] — [EXEC]
    - [RUNPRE] — [EXEC]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MEM mode

1/2

PROG

Program display screen

- [PRGRM] — [(OPRT)] (1)
  - [BG–EDT] ⇒ See“When the soft key [BG–EDT] is pressed”
  - (O number) — [O SRH]
  - (N number) — [N SRH]
  - [REWIND]
  - ▷
  - [P TYPE]
  - [Q TYPE]
  - ▷
  - [F SRH] — [CAN] / [EXEC]

Program check display screen

- [CHECK] — [ABS] / [REL] — [(OPRT)]
  - [BG–EDT] ⇒ See“When the soft key [BG–EDT] is pressed”
  - (O number) — [O SRH]
  - (N number) — [N SRH]
  - [REWIND]
  - ▷
  - [P TYPE]
  - [Q TYPE]
  - ▷
  - [F SRH] — [CAN] / [EXEC]

Current block display screen

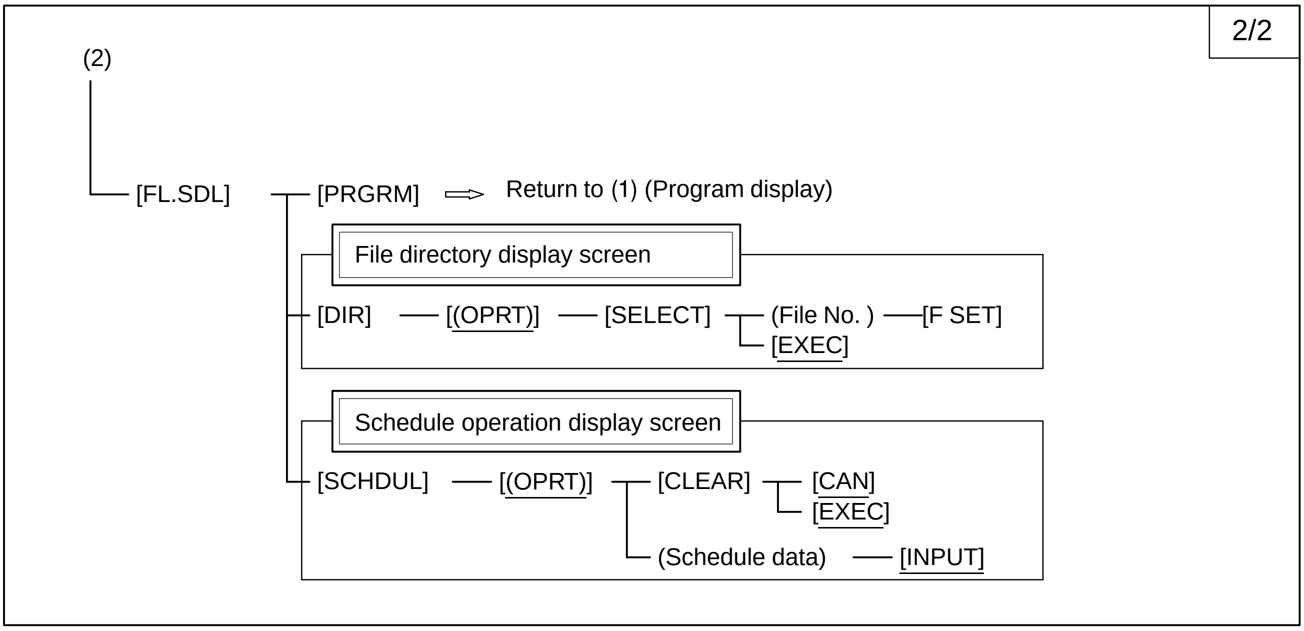
- [CURRNT] — [(OPRT)] — [BG–EDT] ⇒ See“When the soft key [BG–EDT] is pressed”

Next block display screen

- [NEXT] — [(OPRT)] — [BG–EDT] ⇒ See“When the soft key [BG–EDT] is pressed”

▷

(2)(Continued on the next page)


2/2
(2)
[FL.SDL]
[PRGRM]
Return to (1) (Program display)
File directory display screen
[DIR]
[(OPRT)]
[SELECT]
(File No.)
[F SET]
[EXEC]
Schedule operation display screen
[SCHDUL]
[(OPRT)]
[CLEAR]
[CAN]
[EXEC]
(Schedule data)
[INPUT]

PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the EDIT mode

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]

Graphic Conversational Programming

- [C.A.P.]
  - [PRGRM] ⇒ Return to the program
  - [G.MENU]
  - (G number) — [BLOCK] — (Data) — [INPUT] — INSERT
    When a G number is omitted, the standard screen appears.
  - [(OPRT)] — [INPUT]

▷

Floppy directory display

- [FLOPPY]
  - [PRGRM] ⇒ Return to the program
  - [DIR] — [(OPRT)]
    - [F SRH]
      - (Number) — [F SET]
      - [CAN]
      - [EXEC]
    - [READ]
      - (Number) — [F SET]
      - (Number) — [O SET]
      - [STOP]
      - [CAN]
      - [EXEC]
    - [PUNCH]
      - (Number) — [F SET]
      - (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 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 SCREEN

Soft key transition triggered by the function key PROG in the TJOG or THDL mode

PROG

Program input screen

[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)

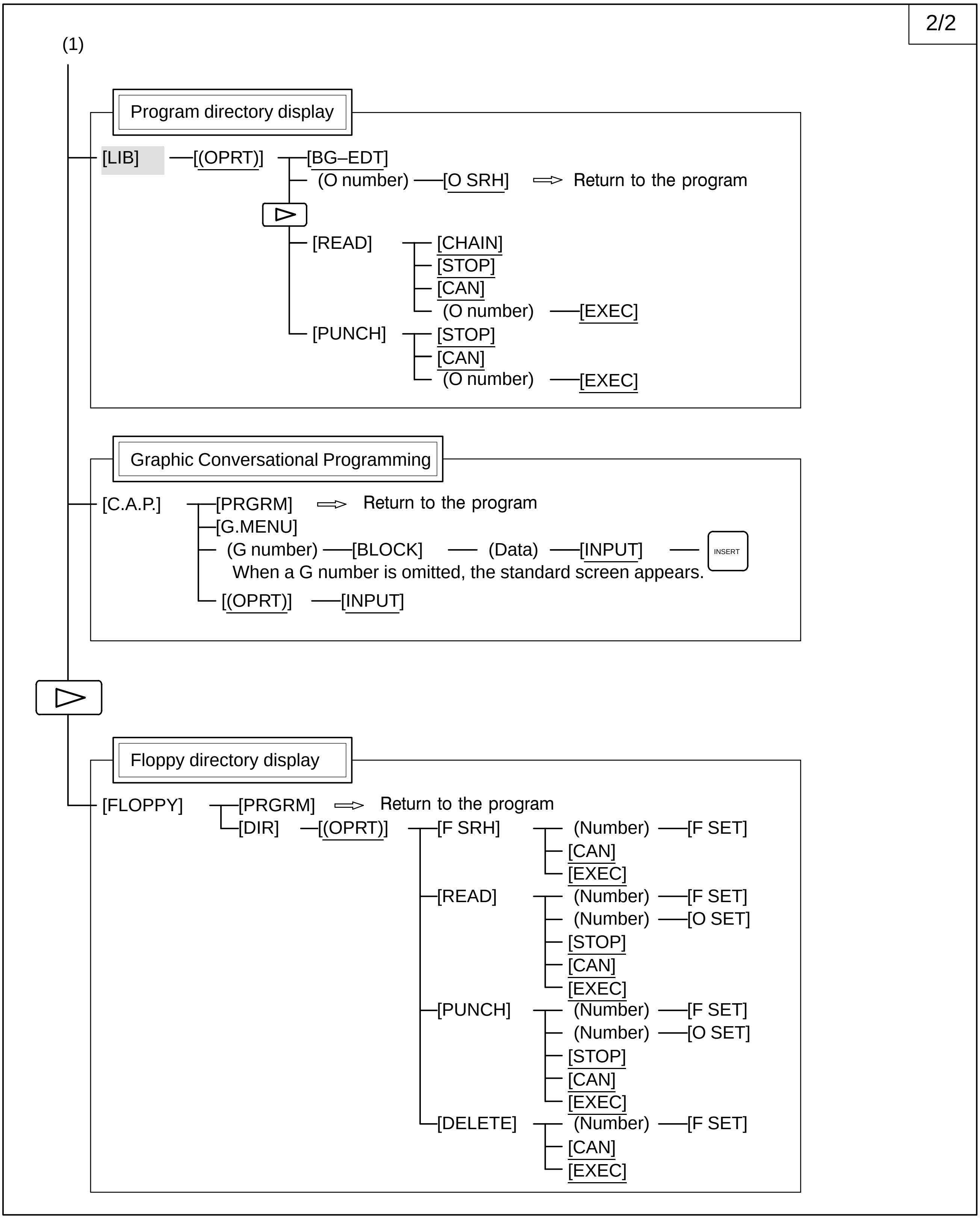
1/2

PROG

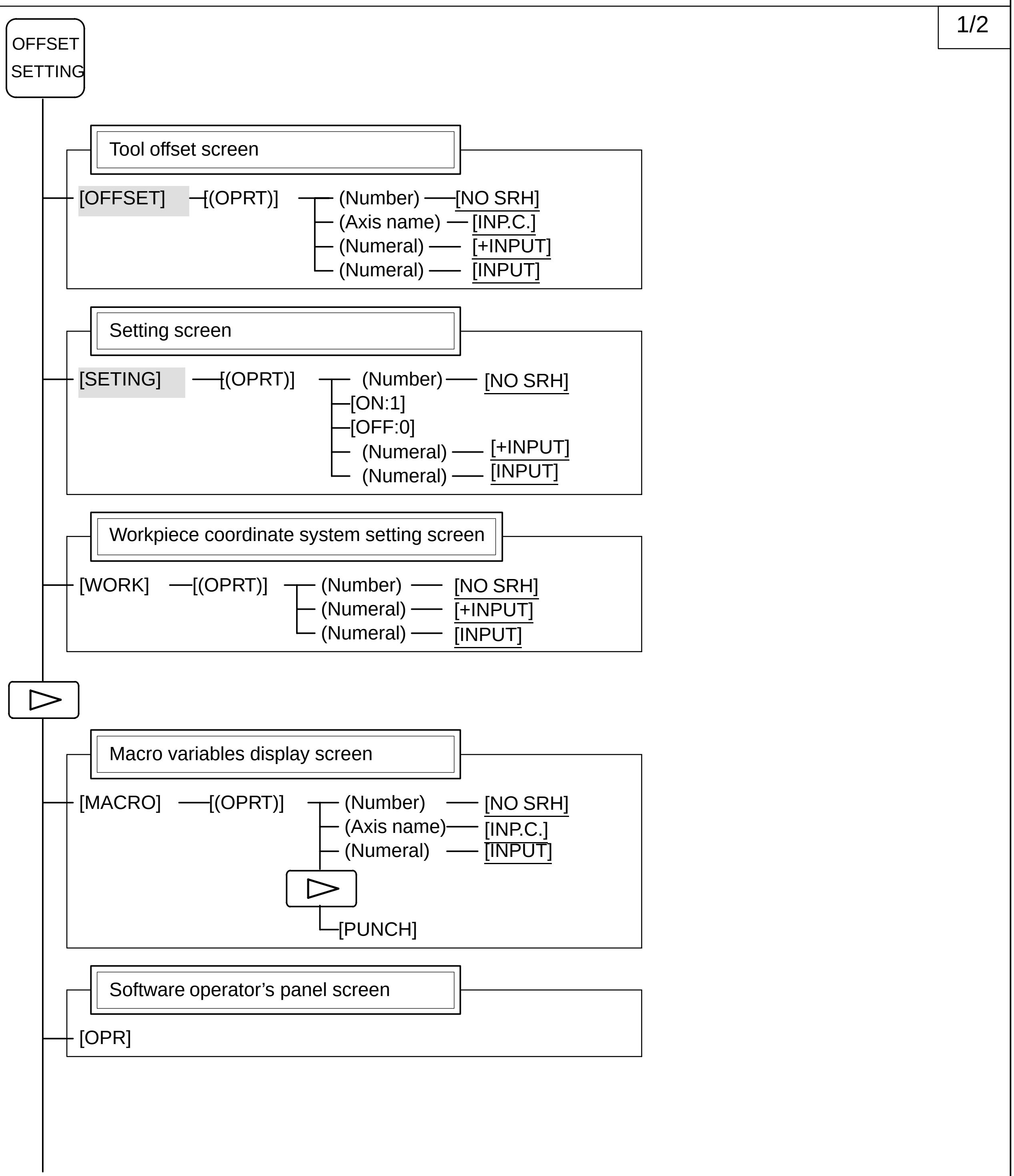
Program display

- [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]

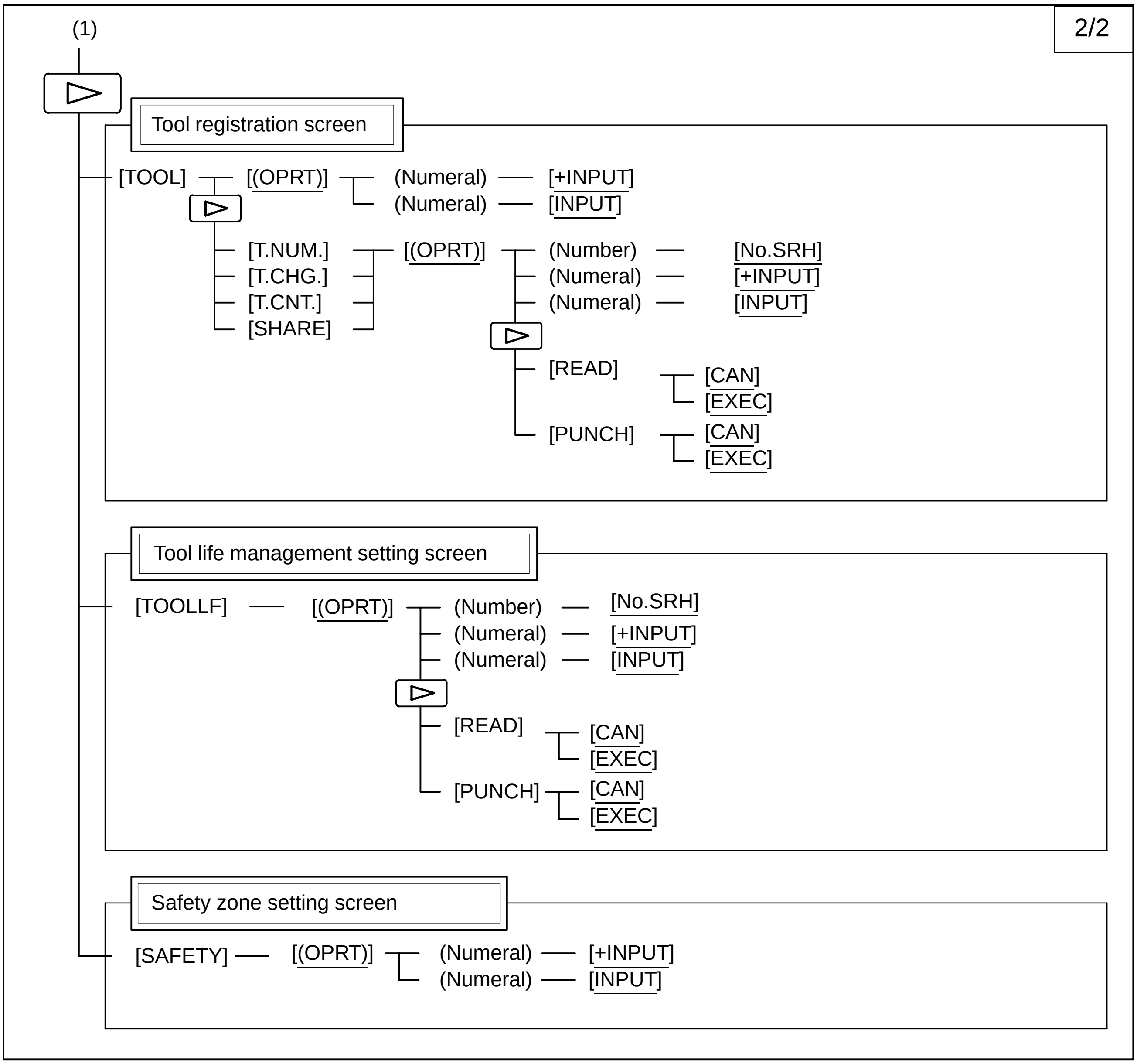
(1)(Continued on the next page)


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]
Graphic Conversational Programming
[C.A.P.] [PRGRM] Return to the program
[G.MENU]
(G number) [BLOCK] (Data) [INPUT]
INSERT
When a G number is omitted, the standard screen appears.
[(OPRT)] [INPUT]
Floppy directory display
[FLOPPY] [PRGRM] Return to the program
[DIR] [(OPRT)] [F SRH] (Number) [F SET]
[CAN]
[EXEC]
[READ] (Number) [F SET]
(Number) [O SET]
[STOP]
[CAN]
[EXEC]
[PUNCH] (Number) [F SET]
(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)(Continued on the next page)


(1)
2/2
Tool registration screen
[TOOL]
[(OPRT)]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]
[T.NUM.]
[T.CHG.]
[T.CNT.]
[SHARE]
[(OPRT)]
(Number)
[No.SRH]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]
[READ]
[CAN]
[EXEC]
[PUNCH]
[CAN]
[EXEC]
Tool life management setting screen
[TOOLLF]
[(OPRT)]
(Number)
[No.SRH]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]
[READ]
[CAN]
[EXEC]
[PUNCH]
[CAN]
[EXEC]
Safety zone setting screen
[SAFETY]
[(OPRT)]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]

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

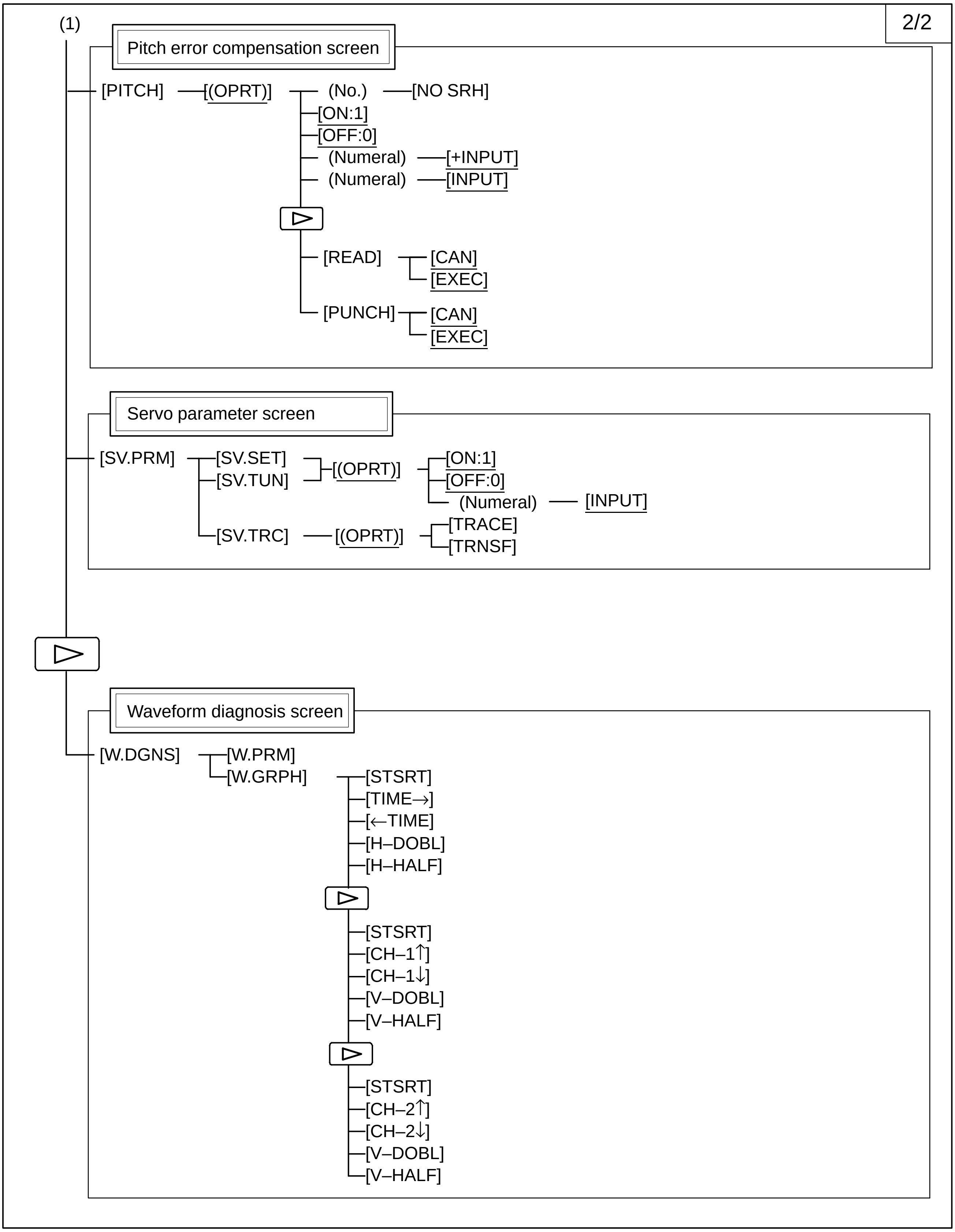
[PMC]

System configuration screen

[SYSTEM]

▷

(1)
(Continued on the next page)


2/2
(1)
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] — Alarm display screen
- [MSG] — Message display screen
- [HISTRY] —[(OPRT)] —[CLEAR] — Alarm history screen

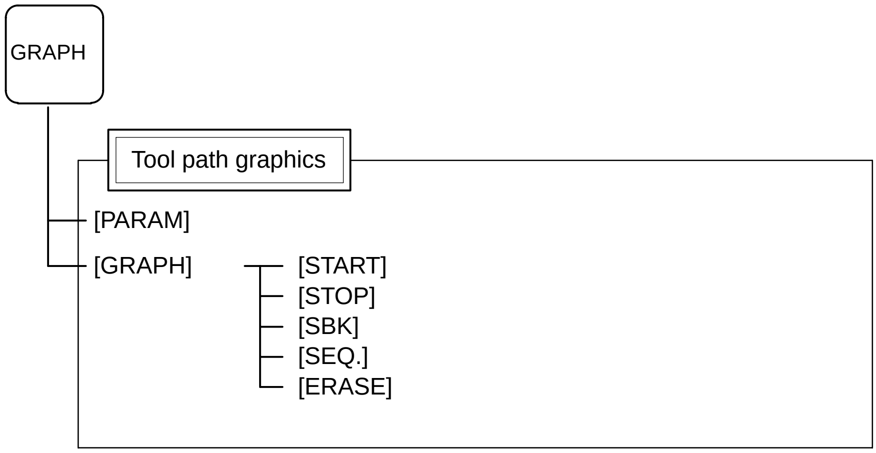
HELP SCREEN

Soft key transition triggered by the function key HELP

HELP

- [1 ALAM] —[(OPRT)] —[SELECT] — Alarm detail screen
- [2 OPR] —[(OPRT)] —[SELECT] — Operation method screen
- [3 PARA] — Parameter table screen

GRAPHIC SCREEN Soft key transition triggered by the function key GRAPH



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 screen. In order to indicate that it is key input data, a “>” symbol is displayed immediately in front of it. A “_” is displayed at the end of the key input data indicating the input position of the next character.

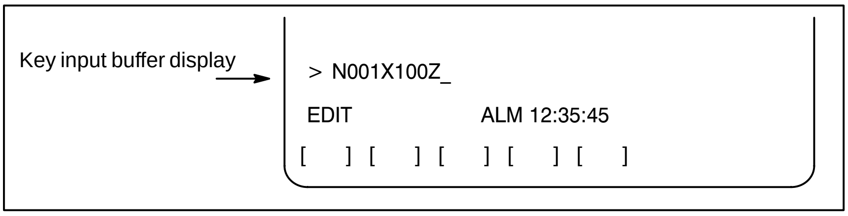


Fig. 2.3.4 Key input buffer display

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.

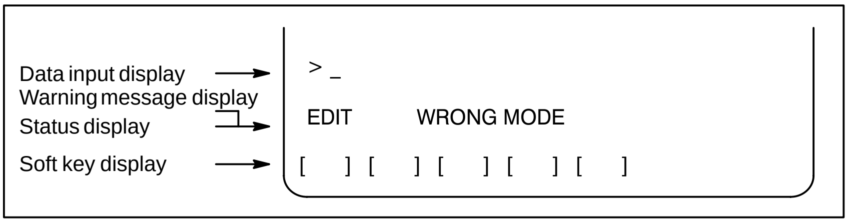


Fig. 2.3.5 Warning message display

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 signal or the parameter is not write enabled.
DATA IS OUT OF RANGE	The value searched 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. 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.

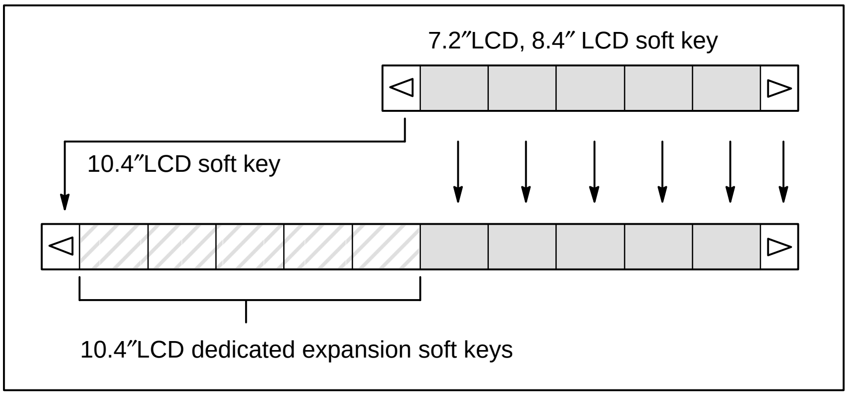
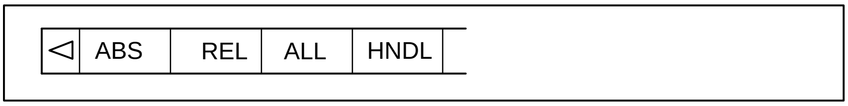


Fig. 2.3.6 LCD soft key configuration

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″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
FANUC Floppy Cassette	Input/output device. Uses floppy disks.	2500m	B–66040E
FANUC FA Card	Compact input/output device. Uses FA cards.	160m	B–61274E
FANUC PPR	Input/output device consisting of a paper tape reader, tape punch, and printer.	275m	B–58584E
Portable Tape Reader	Input device for reading paper tape.	———	

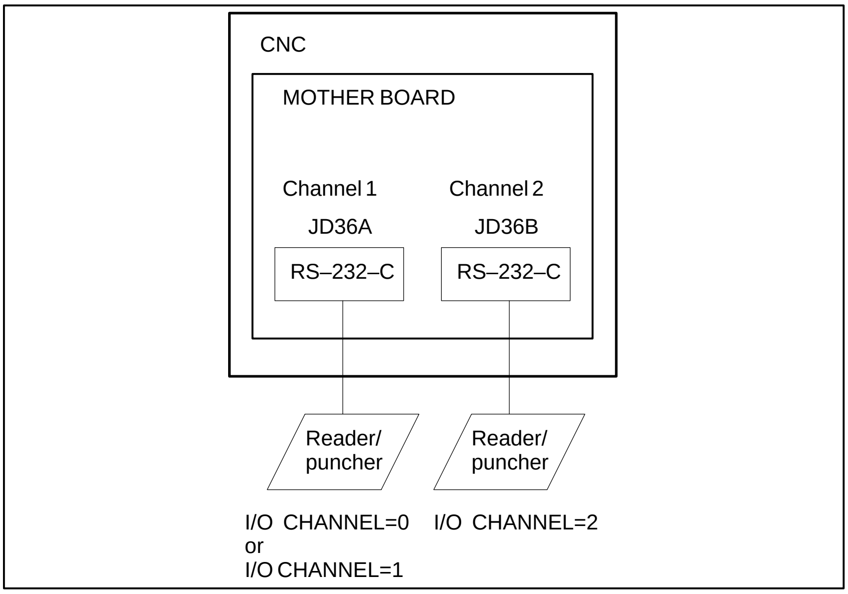
The following data can be input/output to or from external input/output devices:

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

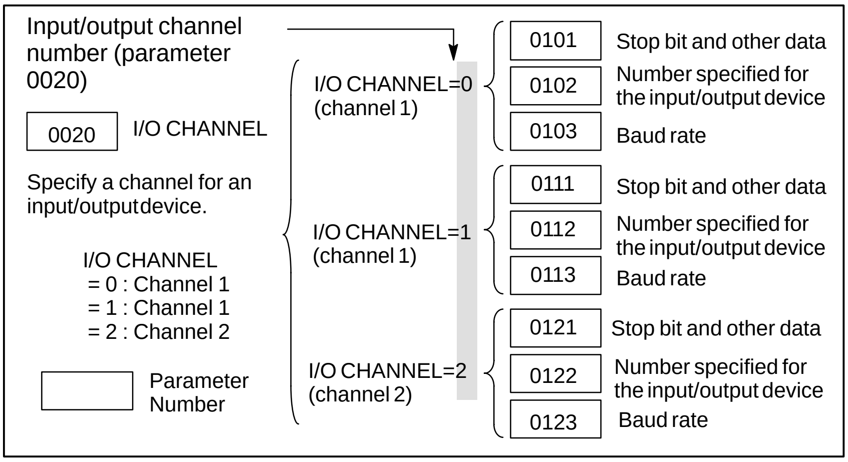
For how data is input and output, see III–8.

Parameter

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

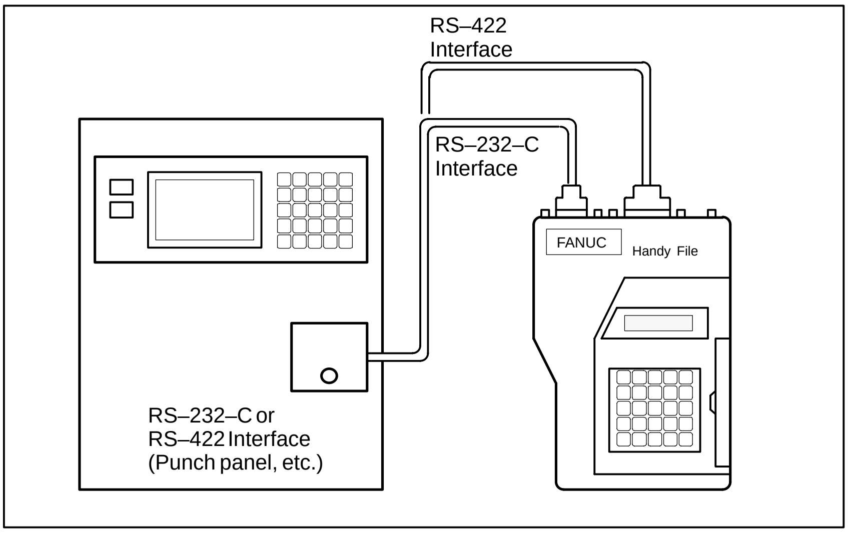


CNC has two channels of reader/punch interfaces. The input/output device to be used is specified by setting the channel (interface) connected to that device in setting parameter I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a 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.



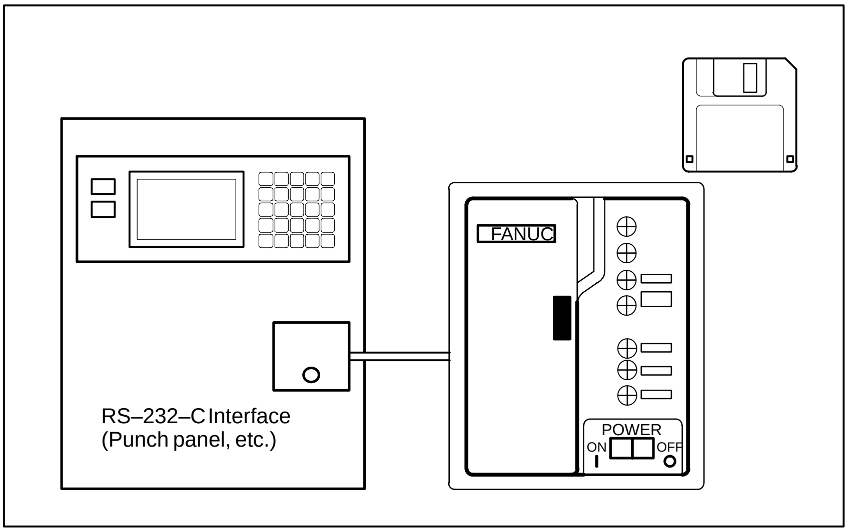
2.4.1 FANUC Handy File

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.



2.4.2 FANUC Floppy Cassette

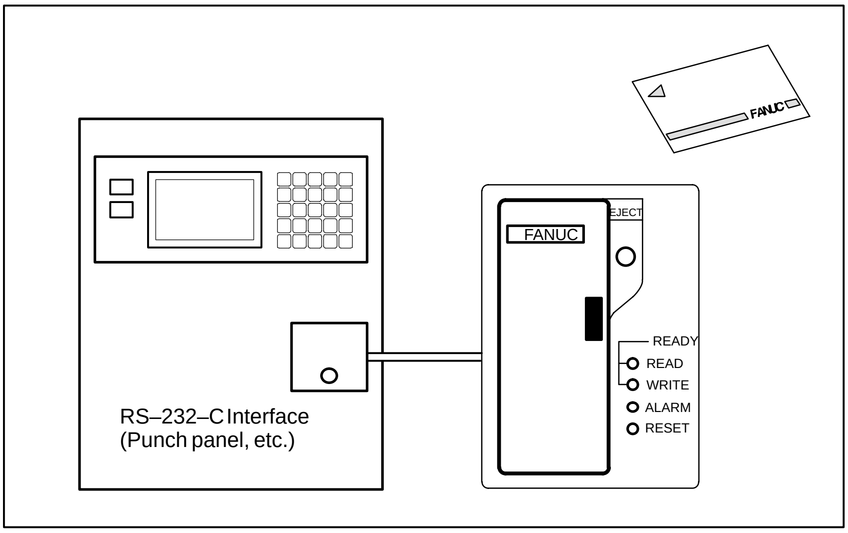
When the Floppy Cassette is connected to the CNC, machining programs stored in the CNC can be saved on a Floppy Cassette, and machining programs saved in the Floppy Cassette can be transferred to the CNC.



2.4.3 FANUC FA Card

An FA Card is a memory card used as an input medium in the FA field. It is a card–shaped input/output medium featuring a high reliability, small size, high capacity, and maintenance–free operation.

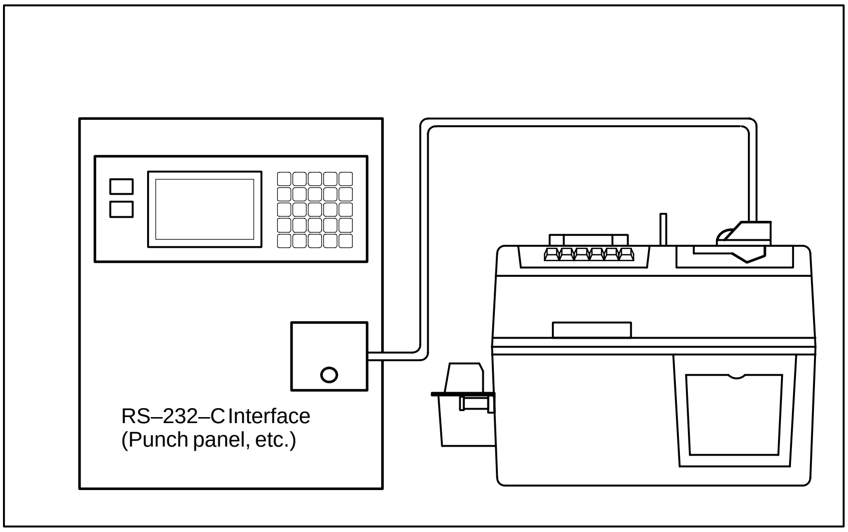
When an FA Card is connected to the CNC via the card adapter, machining programs stored in the CNC can be transferred to and saved in an FA Card. Machining programs stored on an FA Card can also be transferred to the CNC.



2.4.4 FANUC PPR

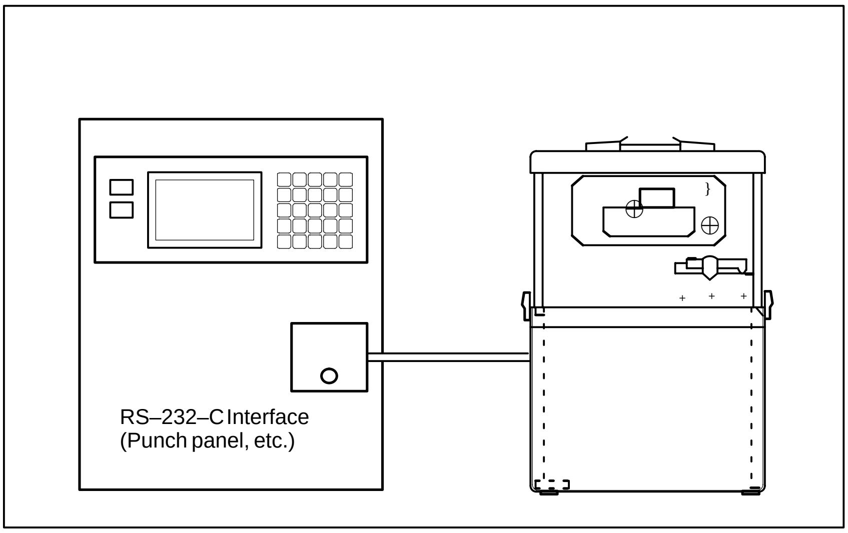
The FANUC PPR consists of three units: A printer, paper tape punch, and paper tape reader.

When the PPR is used alone, data can be read from the tape reader and printed or punched out. It is also possible to perform TH and TV checks on data that was read.



2.4.5 Portable Tape Reader

The portable tape reader is used to input data from paper tape.



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. If the screen shown in Section 2.4.2 is displayed, a system failure may have occurred.

```
ACTUAL POSITION(ABSOLUTE)      O1000 N00010

   X     123.456
   Y     363.233
   C       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 ]
```

Fig. 2.5.1 Position screen (Seven soft key type)

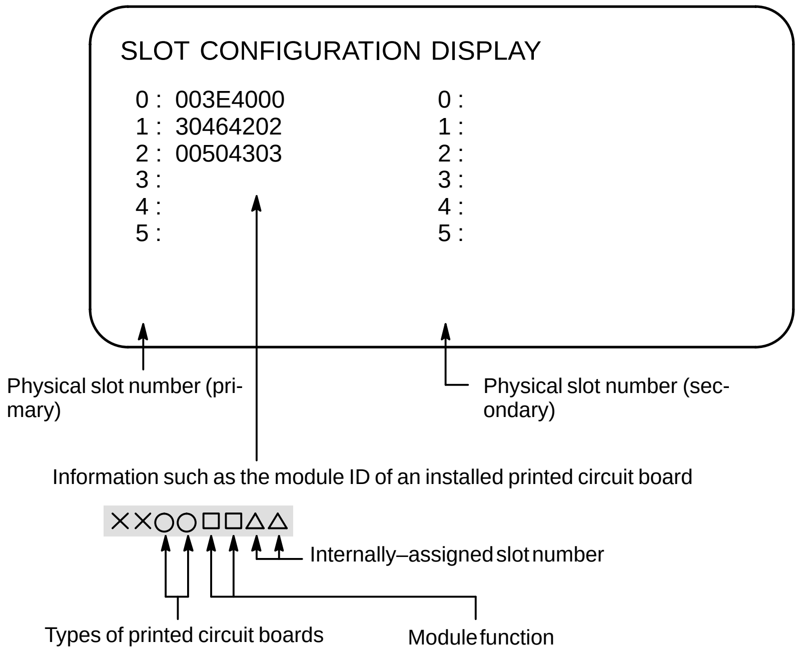
4 Check that the fan motor is rotating.

> **WARNING**
> When pressing the <POWER ON> key, do not touch any other LCD/MDI panel keys. Until the positional or alarm screen is displayed, 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

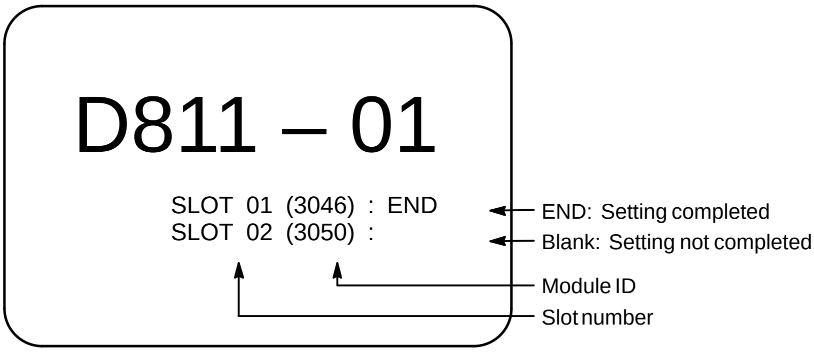
If a hardware failure or installation error occurs, the system displays one of the following three types of screens then stops.
Information such as the type of printed circuit board installed in each slot is indicated. This information and the LED states are useful for failure recovery.

Slot status display

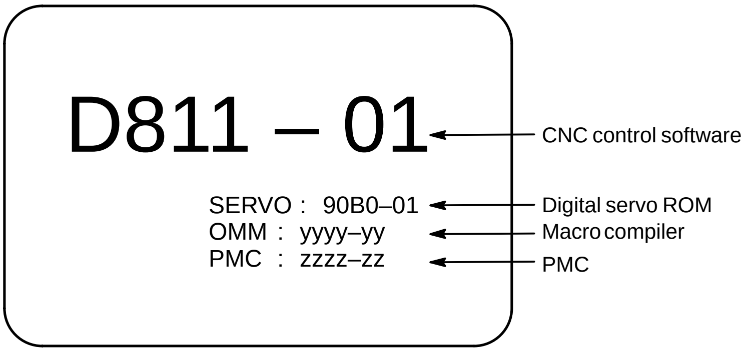


For more information about the types of printed circuit boards and module functions, refer to the maintenance manual (B–63525EN).

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–64115EN) for the system configuration screen.

2.5.3 Power Disconnection

Procedure

1 Check that the LED indicating the cycle start is off on the operator’s panel.

2 Check that all movable parts of the CNC machine tool is stopping.

3 If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.

4 Continue to press the POWER OFF pushbutton for about 5 seconds.

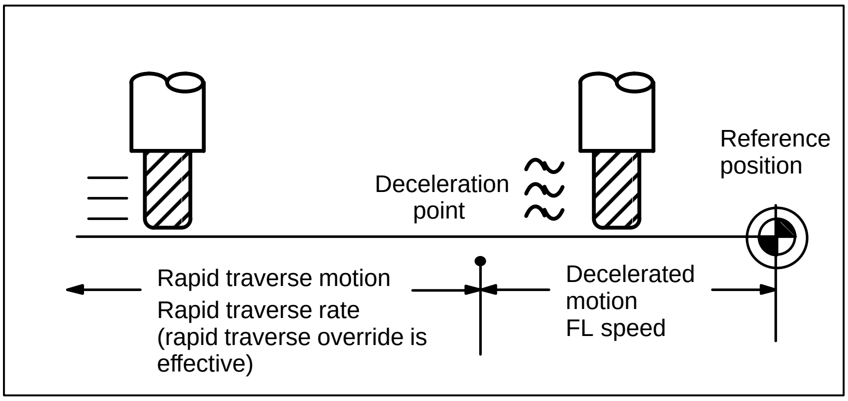
5 Refer to the machine tool builder’s manual for turning off the power to the machine.

3 MANUAL OPERATION

MANUAL OPERATION are four kinds as follows :
3.1. Manual reference position return
3.2. Jog feed (JOG)
3.3. Incremental feed
3.4. Manual handle feed
3.5. Manual absolute on

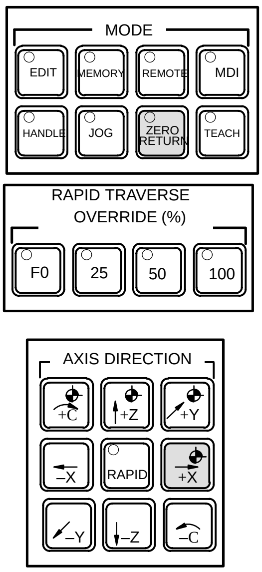
3.1 MANUAL REFERENCE POSITION RETURN

The tool is returned to the reference position as follows :
The tool is moved in the direction specified in parameter ZMI (bit 5 of No. 1006) for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 1420,1421, and 1425).
Fourstep rapid traverse override is effective during rapid traverse.
When the tool has returned to the reference position, the reference position return completion LED goes on. The tool generally moves along only a single axis, but can move along three axes simultane ously when specified so in parameter JAX(bit 0 of No.1002).



Procedure for Manual Reference Position Return

Procedure

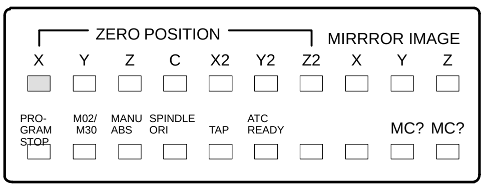


1 Press the reference position return switch, one of the mode selection swithces.

2 To decerease the feedrate, press a rapid traverse override switch. When the tool has returned to the reference position, the reference position return completion LED goes on.

3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool can be moved along three axes simultaneously when specified so in an appropriate parameter setting. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter.

4 Perform the same operations for other axes, if necessary.
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.



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 on the tool holder or the position of the tip of the reference tool 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$\underline{\alpha}$Y$\underline{\beta}$Z$\underline{\gamma}$;
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 tool does not move unless the REFERENCE POSITION RETURN switch is turned off.

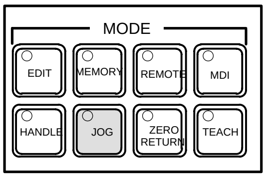
- **Reference position return completion LED**

The REFERENCE POSITION RETURN COMPLETION LED is extinguished by either of the following operations:
– Moving from the reference position.
– Entering an emergency stop state.

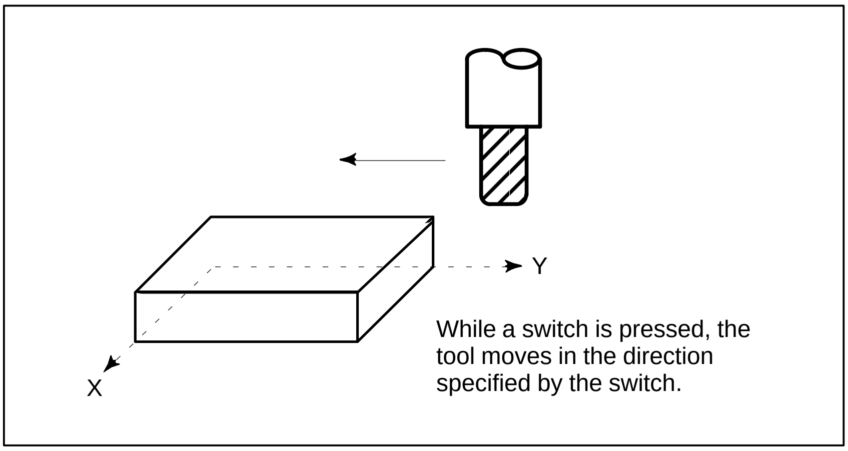
- **The distance to return to reference position**

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

3.2 JOG FEED (JOG)

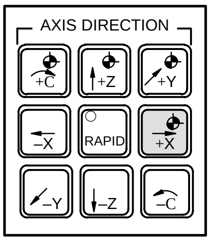


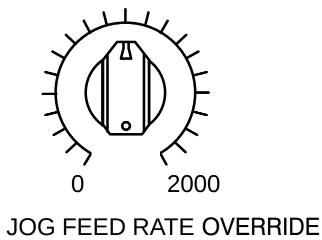
In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.
The jog feedrate is specified in a parameter (No.1423)
The jog feedrate can be adjusted with the jog feedrate override dial.
Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate regardless of the postiotion of the jog feedrate override dial.
Manual operation is allowed for one axis at a time. 3 axes can be selected at a time by parameter JAX (No.1002#0).

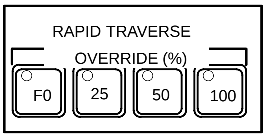


Procedure for Jog Feed

Procedure







1 Press the jog switch, one of the mode selection switches.

2 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in a parameter (No. 1423). The tool stops when the switch is released.

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

4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate while the rapid traverse switch is pressed. Rapid traverse override by the rapid traverse override switches is effective during rapid traverse.

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

Limitations

- **Acceleration/deceleration for rapid traverse**

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

- **Change of modes**

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

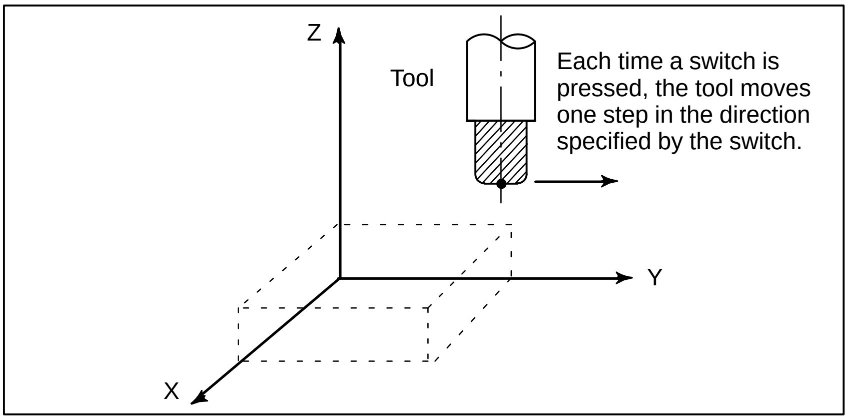
- **Rapid traverse prior to reference position return**

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).

T–axis immediately decelerates and stops when the switch is released before the manual reference point return after power ON. After the manual reference point return, if the switch is released, the T–axis will select the nearest tool that can decelerate and stop in the direction in which it is moving, and stop.

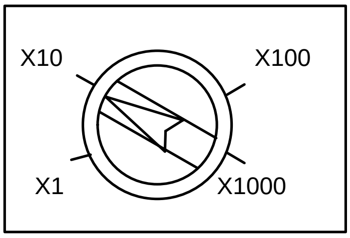
3.3 INCREMENTAL FEED

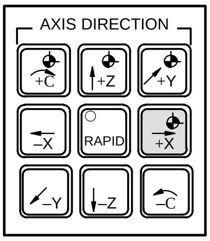
In the incremental (INC) mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.
This mode is effective when a manual pulse generator is not connected.



Procedure for Incremental Feed

Procedure





1 Press the INC switch, one of the mode selection switches.

2 Select the distance to be moved for each step with the magnification dial.

3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.

4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate. Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

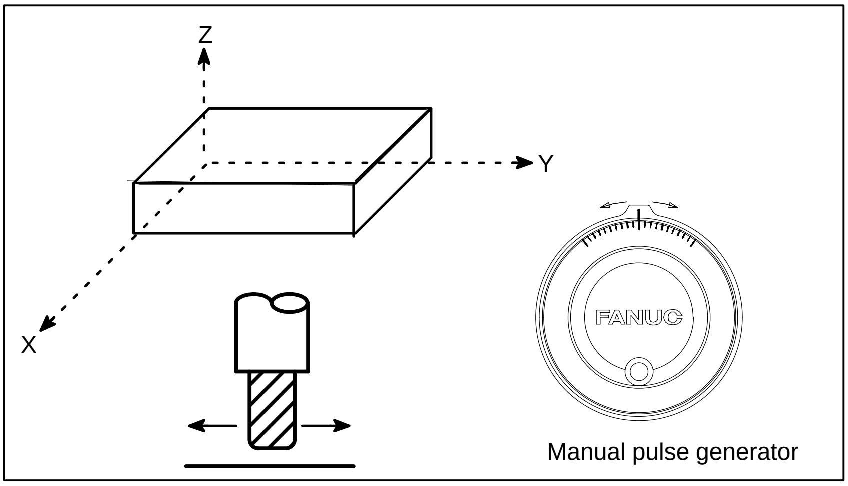
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Limitations

The incremental feed is not applicable to the T–axis.

3.4 MANUAL HANDLE FEED

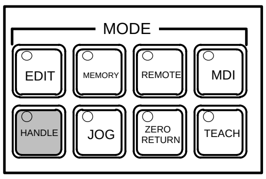
In the handle mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.
The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. Or the distance the tool is moved when the manual pulse generator is rotated by one graduation can be magnified by 10 times or by one of the two magnifications specified by parameters (No. 7113 and 7114).

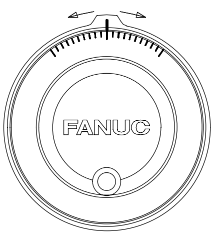


Manual pulse generator

Procedure for Manual Handle Feed

Procedure





Manual pulse generator

1 Press the HANDLE switch, one of the mode selection switches.

2 Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.

3 Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.

4 Move the tool along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduations.
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanations

- **Availability of manual pulse generator in Jog mode (JHD)**

Parameter JHD (bit 0 of No. 7100) enables or disables the manual pulse generator 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 pulse generator in the TEACH IN JOG mode.

- **A command to the MPG exceeding rapid traverse rate (HPF)**

Parameters HPF (bit 4 of No. 7100) and 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 tool is moved may not match the graduations on the manual pulse generator.)

Set value 1 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC.
(No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

• 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 tool 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 tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

- **Movement direction of an axis to the rotation of MPG (HNGX)**

Parameter HNGX (No. 7102) switches the direction in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

Restrictions

- **Number of MPGs**

Up to three manual pulse generators can be connected, one for each axis. The three manual pulse generators can be simultaneously operated. Handle feed is not effective in T and C axis.

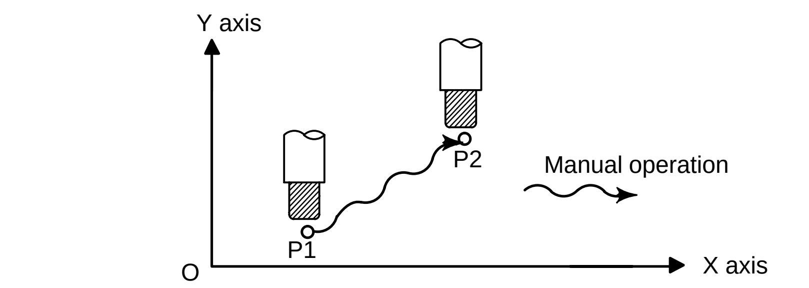
WARNING

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

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

3.5 MANUAL ABSOLUTE ON

The distance of the tool is moved by manual operation is added to the coordinates.



The coordinates values change by the amount of manual operation.

Explanation

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

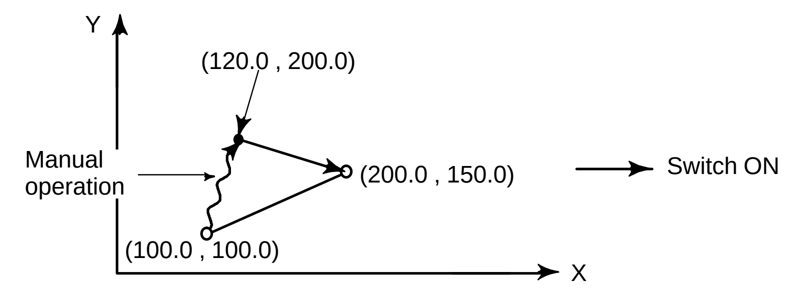
```
G01G90  X100.0Y100.0F010  ; 1
        X200.0Y150.0      ; 2
        X300.0Y200.0      ; 3
```

The subsequent figures use the following notation:

⟶ Movement of the tool when the switch is on

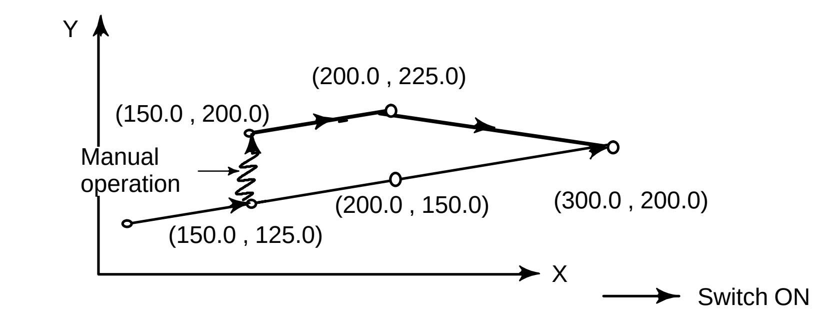
- **Manual operation after the end of block**

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 1.



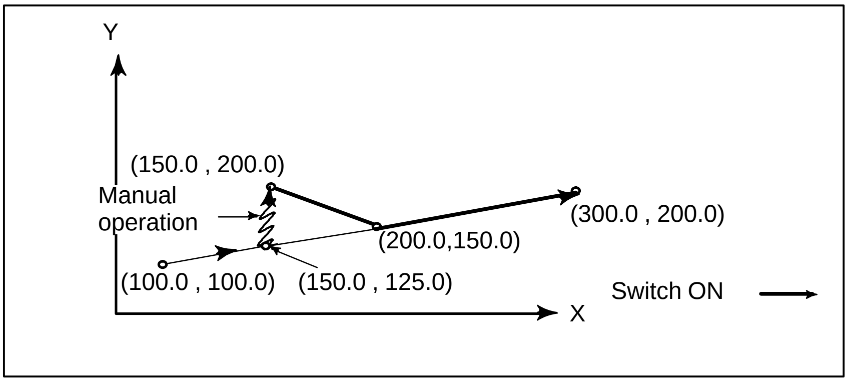
- **Manual operation after a feed hold**

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.



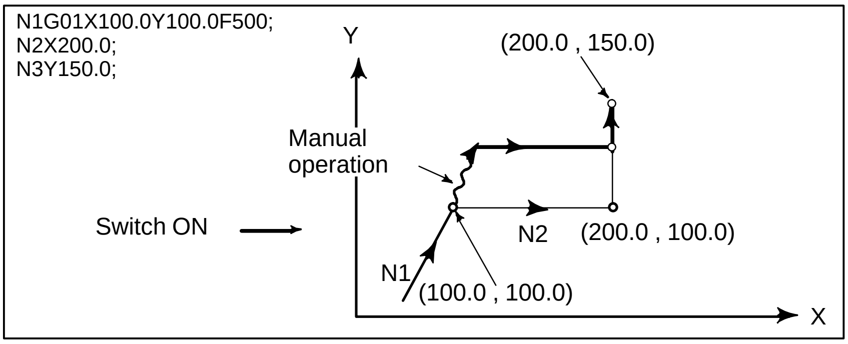
- **When reset after a manual operation following a feed hold**

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.

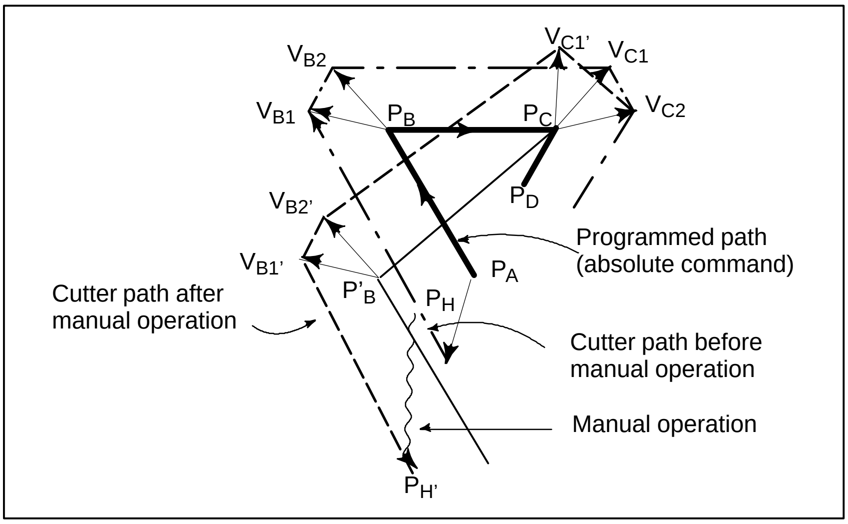
- **Manual operation during cutter compensation**

When the switch is ON during cutter compensation

Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the cutter compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

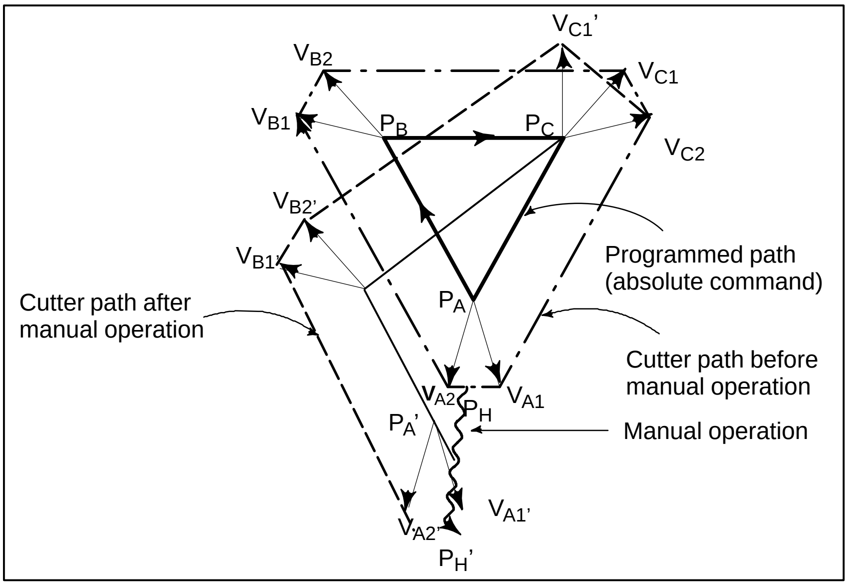
Manual operation performed in other than cornering

Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B, and P_C and that the tool was manually moved to $P_{H'}$. The block end point P_B moves to the point $P_{B'}$ by the amount of manual movement, and vectors V_{B1} and V_{B2} at P_B also move to $V_{B1'}$ and $V_{B2'}$. Vectors V_{C1} and V_{C2} between the next two blocks P_B – P_C and P_C – P_D are discarded and new vectors $V_{C1'}$ and $V_{C2'}$ ($V_{C2'}$ = V_{C2} in this example) are produced from the relation between $P_{B'}$ – P_C and P_C – P_D. However, since $V_{B2'}$ is not a newly calculated vector, correct offset is not performed at block $P_{B'}$ – P_C. Offset is correctly performed after P_C.



Manual operation during cornering

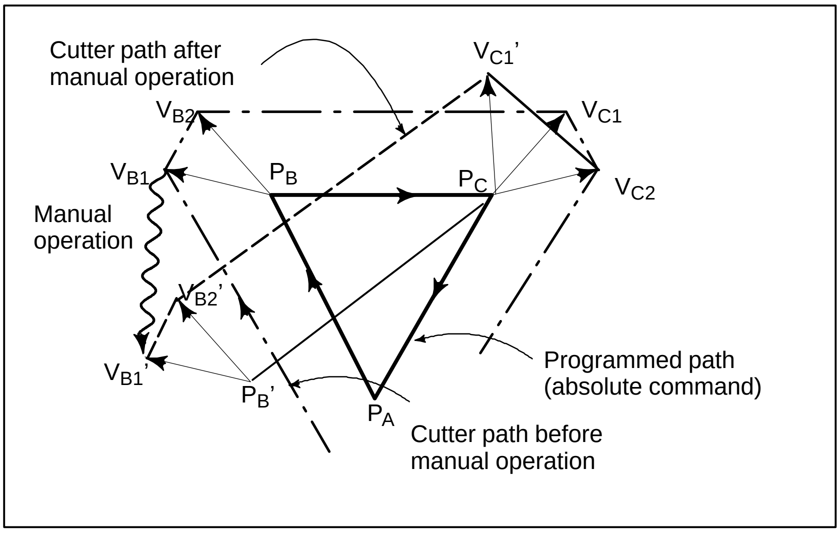
This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$, and $V_{B2'}$ are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct cutter compensation is performed for the blocks following Pc.



Manual operation after single block stop

Manual operation was performed when execution of a block was terminated by single block stop.

Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub–sequent processing is the same as case a described above. An MDI operation can also be interveneted as well as manual operation. The movement is the same as that by manual operation.



4 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.
This chapter explains the following types of automatic operation:

- **MEMORY OPERATION**
  Operation by executing a program registered in CNC memory
- **MDI OPERATION**
  Operation by executing a program entered from the MDI panel
- **DNC operation**
  Operation while reading a program from an input/output device
- **SCHEDULING FUNCTION**
  Scheduled operation by executing programs (files) registered in an external input/output device (Handy File, Floppy Cassette, and so on)
- **SUBPROGRAM CALL FUNCTION**
  Function for calling and executing subprograms (files) registered in an external input/output device (Handy File, Floppy Cassette, and so on) during memory operation
- **MANUAL HANDLE INTERRUPTION**
  Function for performing manual feed during movement executed by automatic operation
- **MIRROR IMAGE**
  Function for enabling mirror–image movement along an axis during automatic operation
- **MANUAL INTERVENTION AND RETURN**
  Function restarting automatic operation by returning the tool to the position where manual intervention was started during automatic operation

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.
When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.
When the reset switch on the LCD/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, S, or T was being executed, the operation is stopped after M, S, or T is finished.

When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.

b. Terminating memory operation

Press the RESET key on the MDI panel.

Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Explanation

Memory operation

After memory operation is started, the following are executed:

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

Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

– The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).
– There are two keys to stop memory operation: The feed hold key and reset key.

• Program stop (M00)

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

• Optional stop (M01)

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

• Program end (M02, M30)

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered.
In some machines, M30 returns control to the top of the program. For details, refer to the manual supplied by the machine tool builder.

• Feed hold

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

• Reset

Automatic operation can be stopped and the system can be made to the reset state by using RESET key on the MDI panel or external reset signal.

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

- **Optional block skip**

When the optional block skip switch on the machine operator's panel is turned on, blocks containing a slash (/) are ignored.

Calling a subprogram stored in an external input/output device

A file (subprogram) in an external input/output device such as a Floppy Cassette can be called and executed during memory operation. For details, see Section 4.5.

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)                          O0017 N00000
O0000
%




G00  G90  G21  G50  G64
G17  G22  G40  G67  G85
                         H   M
     T                   D
     F                S

)_
MDI **** *** ***         11:22:05
[ 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 Chapter 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 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)                               O0017 N00000
O0000  G00  X100.  Y100.  T10 ;
G72  X200.  Y200.  ;
G26  I100.  J0 K10   ;
%


G00  G90  G21  G50  G64
G17  G22  G40  G67  G85
                         H   M
     T                   D
     F                S

)_
MDI **** *** ***         11:23:21
[       ][       ][ SRH  ↓ ][ SRG  ↑ ][ REWIND ]
```

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, S, or T was being executed, the operation is stopped after M, S, or T is finished.

When the cycle start switch on the machine operator's panel is pressed, 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).

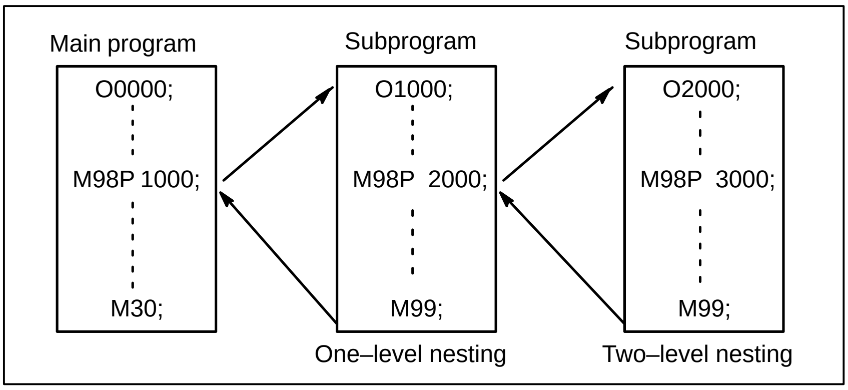


Fig. 4.2 Nesting level of subprograms called from the MDI program

- **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 Cassettes, and so on) 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.

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 key type LCD)**

```
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 key type LCD)**

```
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 key type LCD)**

```
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, 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 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, and so on) and specify the execution order and number of repetitions (scheduling) for performing automatic operation. It is also possible to select only one file from the files in the external input/output device and execute it during automatic operation.

FILE DIRECTORY

FILE NO.	FILE NAME
0001	O0010
0002	O0020
0003	O0030
0004	O0040

List of files in an external input/output device

Set file number and number of repetitions.

ORDER	FILE NO	REPETITION
01	0002	2
02	0003	1
03	0004	3
04	0001	2

Scheduling screen

Executing automatic operation

Procedure for Scheduling Function

Procedure

- **Procedure for executing one file**

1 Press the **MEMORY** switch on the machine operator's panel, then press the PROG function key on the MDI panel.

2 Press the rightmost soft key (continuous menu key), then press the **[FL. SDL]** soft key. A list of files registered in the Floppy Cassette is displayed on screen No. 1. To display more files that are not displayed on this screen, press the page key on the MDI panel. Files registered in the Floppy Cassette can also be displayed successively.

```
FILE DIRECTORY                              O0001  N00000
 CURRENT SELECTED : SCHEDULE
  NO. FILE NAME  (METER)  VOL
  0000  SCHEDULE
  0001  PARAMETER   58.5
  0002  ALL PROGRAM  11.0
  0003  O0001       1.9
  0004  O0002       1.9
  0005  O0010       1.9
  0006  O0020       1.9
  0007  O0040       1.9
  0008  O0050       1.9

 MEM  ****  ***  ***              19 : 14 : 47
( PRGRM )(        )(  DIR  )( SCHDUL )( (OPRT) )
```

Screen No.1

3 Press the **[(OPRT)]** and **[SELECT]** soft keys to display "SELECT FILE NO." (on screen No. 2). Enter a file number, then press the **[F SET]** and **[EXEC]** soft keys. The file for the entered file number is selected, and the file name is indicated after "CURRENT SELECTED:".

```
FILE DIRECTORY                              O0001  N00000
 CURRENT SELECTED:O0040
  NO. FILE NAME  (METER)  VOL
  0000  SCHEDULE
  0001  PARAMETER   58.5
  0002  ALL PROGRAM  11.0
  0003  O0001       1.9
  0004  O0002       1.9
  0005  O0010       1.9
  0006  O0020       1.9
  0007  O0040       1.9
  0008  O0050       1.9
  SELECT FILE NO.=7
 >_
 MEM  ****  ***  ***              19 : 17 : 10
( F SET )(        )(        )(        )(  EXEC  )
```

Screen No.2

4 Press the **REMOTE** switch on the machine operator's panel to enter the **RMT** mode, then press the cycle start switch. The selected file is executed. For details on the **REMOTE** switch, refer to the manual supplied by the machine tool builder. The selected file number is indicated at the upper right corner of the screen as an F number (instead of an O number).

```
FILE DIRECTORY                                  F0007  N00000
 CURRENT SELECTED:O0040





RMT  ****  ***  ***                  13 : 27 : 54
( PRGRM )(          )( DIR    )( SCHDUL )( (OPRT) )
```

Screen No.3

- **Procedure for executing the scheduling function**

1 Display the list of files registered in the Floppy Cassette. The display procedure is the same as in steps 1 and 2 for executing one file.

2 On screen No. 2, press the **[(OPRT)]** and **[SELECT]** soft keys to display "SELECT FILE NO."

3 Enter file number 0, and press the **[F SET]**, and **[EXEC]** soft keys. "SCHEDULE" is indicated after "CURRENT SELECTED:".

4 Press the leftmost soft key (return menu key) and the **[SCHDUL]** soft key. Screen No. 4 appears.

```
FILE DIRECTORY                                  F0000  N02000
   ORDER FILE NO.    REQ.REP      CUR.REP
     01
     02
     03
     04
     05
     06
     07
     08
     09
     10

>_
MEM  ****  ***  ***                  22 : 07 : 00
( PRGRM )(          )( DIR    )( SCHDUL )( (OPRT) )
```

Screen No.4

Move the cursor and enter the file numbers and number of repetitions in the order in which to execute the files. At this time, the current number of repetitions "CUR.REP" is 0.

5 Press the **REMOTE** switch on the machine operator's panel to enter the **RMT** mode, then press the start switch. The files are executed in the specified order. When a file is being executed, the cursor is positioned at the number of that file.
The current number of repetitions CUR.REP is increased when M02 or M30 is executed in the program being run.

```
FILE DIRECTORY                              O0000  N02000

   ORDER FILE NO.          REQ.REP        CUR.REP
    01      0007               5              5
    02     0003               23             23
    03     0004             9999            156
    04     0005             LOOP              0
    05
    06
    07
    08
    09
    10

 RMT   ****  ***  ***          10 : 10 : 40

(PRGRM )(        )(  DIR  )(SCHDUL )((OPRT) )
```

Screen No.5

Explanations

- **Specifying no file number**

If no file number is specified on screen No. 4 (the file number field is left blank), program execution is stopped at that point. To leave the file number field blank, press numeric key 0 then INPUT .

- **Endless repetition**

If a negative value is set as the number of repetitions, <**LOOP**> is displayed, and the file is repeated indefinitely.

- **Clear**

When the **[(OPRT)]**, **[CLEAR]**, and **[EXEC]** soft keys are pressed on screen No. 4, all data is cleared. However, these keys do not function while a file is being executed.

- **Return to the program screen**

When the PROG soft key is pressed on screen No. 1, 2, 3, 4, or 5, the program screen is displayed.

Restrictions

- **Number of repetitions**

Up to 9999 can be specified as the number of repetitions. If 0 is set for a file, the file becomes invalid and is not executed.

- **Number of files registered**

By pressing the page key on screen No. 4, up to 20 files can be registered.

- **M code**

When M codes other than M02 and M30 are executed in a program, the current number of repetitions is not increased.

- **Displaying the floppy disk directory during file execution**

During the execution of file, the floppy directory display of background editing cannot be referenced.

- **Restarting automatic operation**

To resume automatic operation after it is suspended for scheduled operation, press the reset button.

Alarm

Alarm No.	Description
086	An attempt was made to execute a file that was not registered in the floppy disk.
210	M198 and M099 were executed during scheduled operation, or M198 was executed during DNC operation.

4.5 SUBPROGRAM CALL FUNCTION

The subprogram call function is provided to call and execute subprogram files stored in an external input/output device(Handy File, FLOPPY CASSETTE, FA Card)during memory operation.

When the following block in a program in CNC memory is executed, a subprogram file in the external input/output device is called:

Format

1. FS15 tape format

M198 P○○○○ L∆∆∆∆ ;

- L∆∆∆∆: Number of repetitions
- P○○○○: File number for a file in the I/O device
- M198: I/O devices call instruction

2. Other than FS15 tape format

M198 P○○○○ ∆∆∆∆ ;

- ∆∆∆∆: File number for a file in the I/O device
- P○○○○: Number of repetitions
- M198: I/O devices call instruction

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.

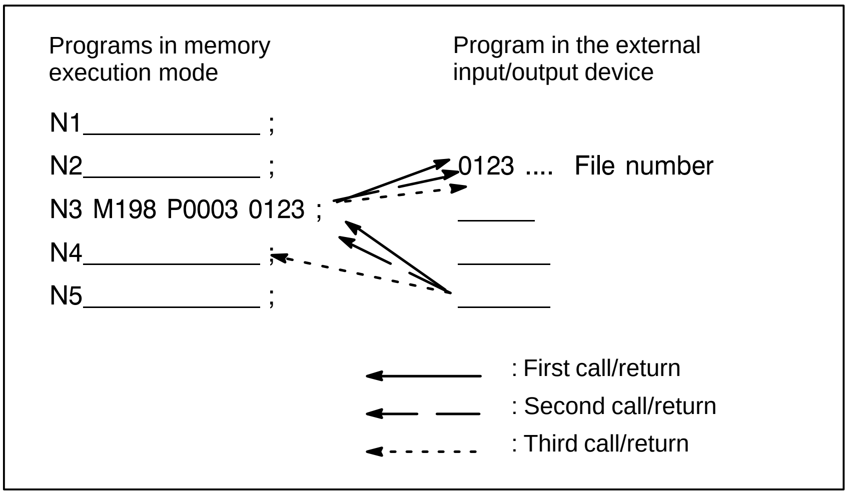


Fig. 4.5 Program flow when M198 is specified

CAUTION

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.

4.6 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.

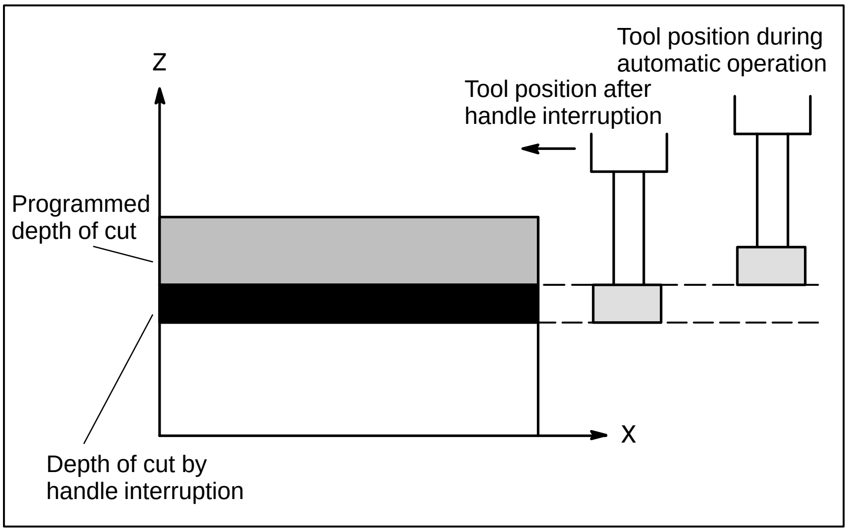


Fig. 4.6 Manual handle interruption

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)

CAUTION
Handle interruption is disabled when the machine is locked during automatic operation.

Explanations

- **Relation with other functions**

The following table indicates the relation between other functions and the movement by handle interrupt.

Signal	Relation
Machine lock	Machine lock is effective. When the machine lock signal is on, handle interrupt is ignored.
Interlock	Interlock is effective. The tool does not move even when this signal turns on.
Mirror image	Mirror image is not effective. Interrupt functions on the plus direction by plus direction command, even if this signal turns on.

- **Position display**

The following table shows the relation between various position display data and the movement by handle interrupt.

Display	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Handle interruption does not change relative coordinates.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

- **Travel distance display**

Press the function key POS, then press the chapter selection soft key **[HNDL]**.
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
  C       –61.439                        C –61.439

  (RELATIVE)                             (DISTANCE TO GO)
  X          0.000                       X 0.000
  Y          0.000                       Y 0.000
  C          0.000                       C 0.000
                                   PART COUNT 287
 RUN TIME   1H 12M CYCLE TIME  0H 0M 0S

MDI   ****  ***  ***                10 : 29 : 51
( ABS )( REL )( ALL )( HNDL )( (OPRT) )
```

(a) INPUT UNIT : Handle interrupt move amount in input unit system
Indicates the travel distance specified by handle interruption according to the least input increment.

(b) OUTPUT UNIT : Handle interrupt move amount in output unit system
Indicates the travel distance specified by handle interruption according to the least command increment.

(c) RELATIVE : Position in relative coordinate system
These values have no effect on the travel distance specified by handle interruption.

(d) DISTANCE TO GO : The remaining travel distance in the current block has no effect on the travel distance specified by handle interruption.

The handle interrupt move amount is cleared when the low speed reference position return (the first reference position return when power is turned on) ends every axis.

4.7 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 LCD/MDI panel.

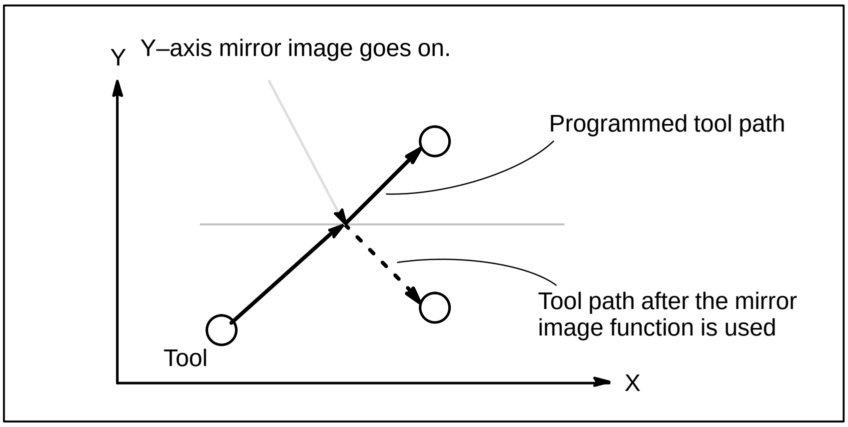


Fig. 4.7 Mirror image

Procedure

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

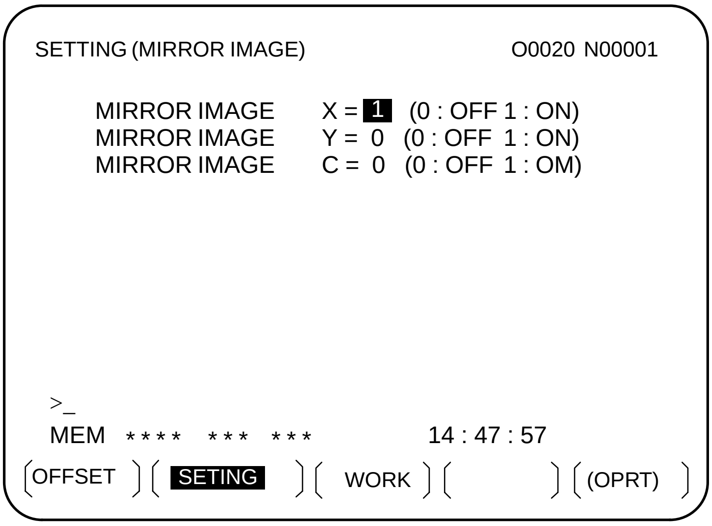
1 Press the single block switch to stop automatic operation. When the mirror image function is used from the begining of operation, this step is omitted.

2 Press the mirror image switch for the target axis on the machine operator's panel.
Alternatively, turn on the mirror image setting by following the steps below:

2–1 Set the **MDI** mode.

2–2 Press the OFFSET SETTING function key.

2–3 Press the **[SETING]** soft key for chapter selection to display the setting screen.



2–4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (memory mode or MDI mode), then press the cycle start button to start automatic operation.

Explanations

· The mirror image function can also be turned on and off by setting bit 0 of parameter 0012 to 1 (on) or 0 (off).

· For the mirror image switches, refer to the manual supplied by the machine tool builder.

Limitations

Operations, such as manual operaton, automatic reference position return, movement in repositioning, and movement related to tool position compensation and C–axis position compensation cannot be reversed.

4.8 DNC OPERATION WITH MEMORY CARD

4.8.1 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. 4.8.1 (a))

(b) It is possible to read sub–programs written in the memory card and execute them by the command Subprogram call (M198). (Fig. 4.8.1 (b))

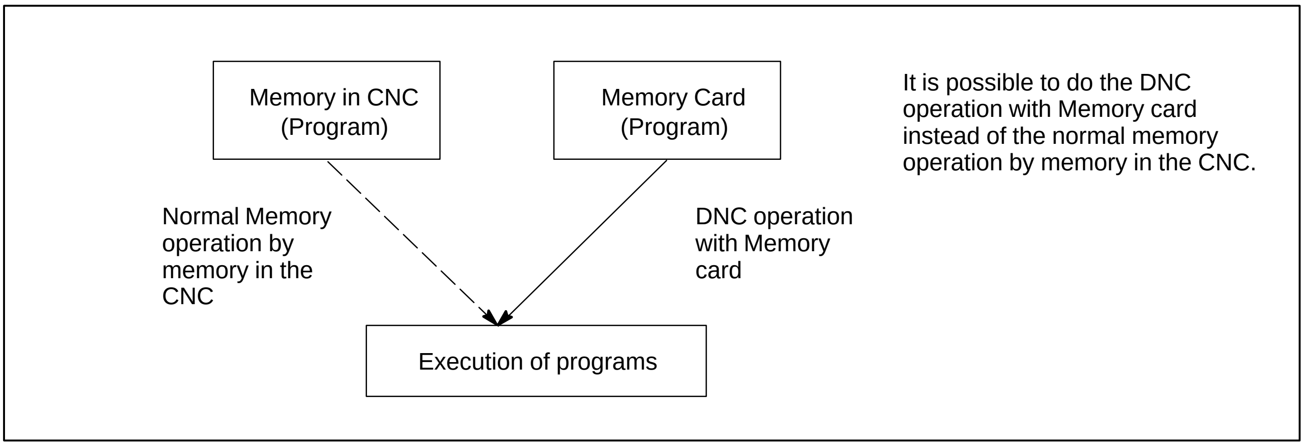


Fig. 4.8.1 (a)

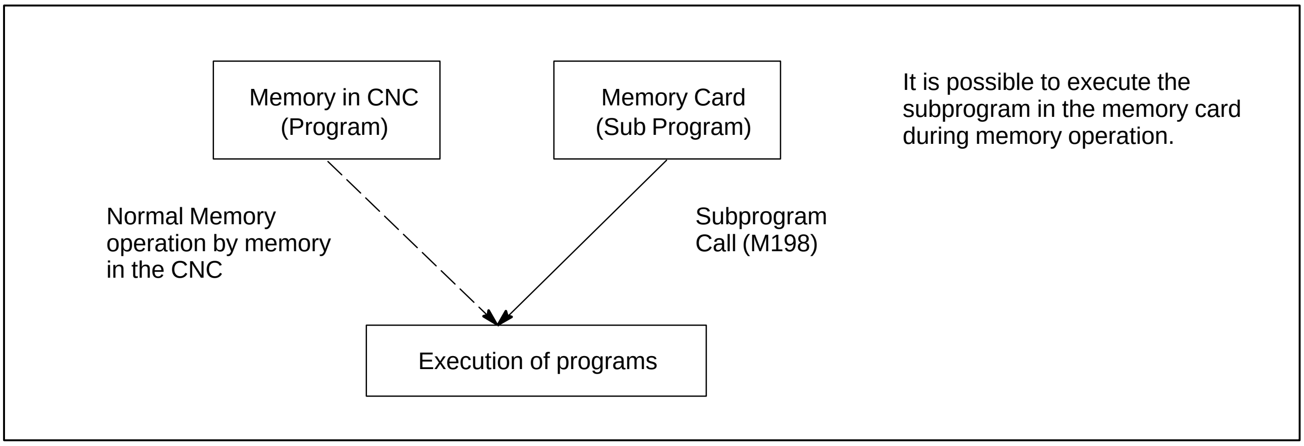


Fig. 4.8.1 (b)

NOTE

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.

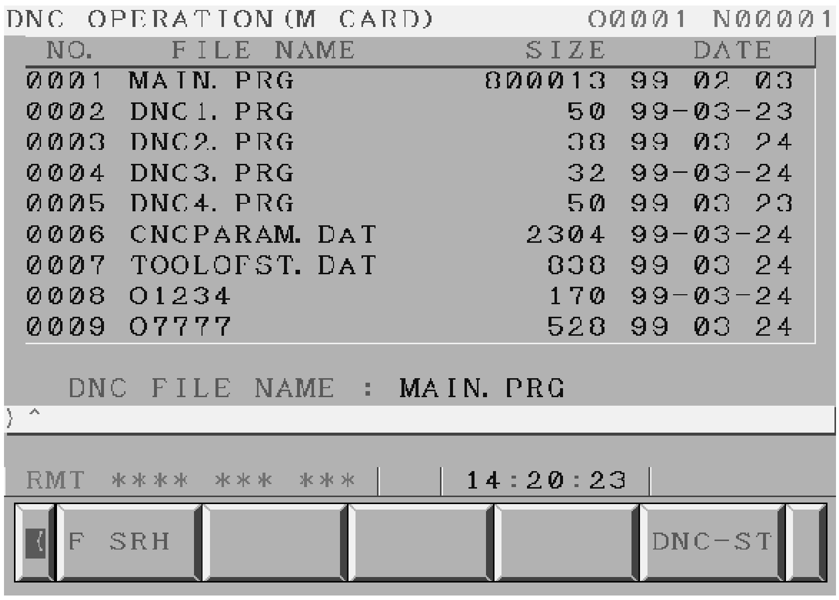
4.8.2 Operations

4.8.2.1 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.



4.8.2.2
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

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.

4.8.3 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) 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.

(3) Please do not pull out and insert memory card during the DNC operation with memory card.

(4) It is not possible to call a program in the memory card from the DNC operation program.

(5) 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.

4.8.4 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.

4.8.5 Connecting PCMCIA Card Attachment

4.8.5.1 Specification number

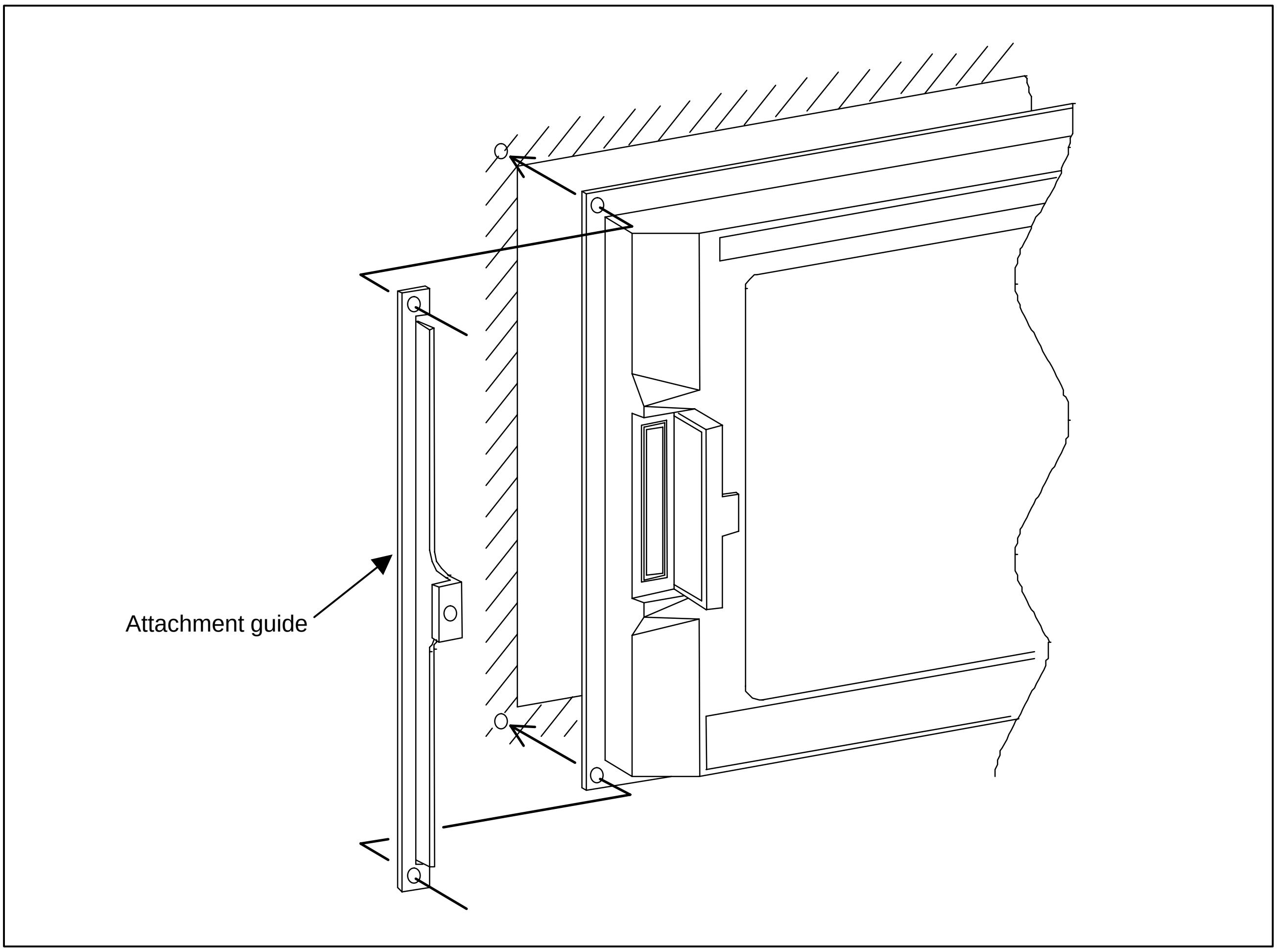
Specification	Remarks
A02B–0236–K160	For 7.2″ LCD or 8.4″ LCD
A02B–0236–K161	For 9.5″ LCD or 10.4″ LCD

4.8.5.2 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.

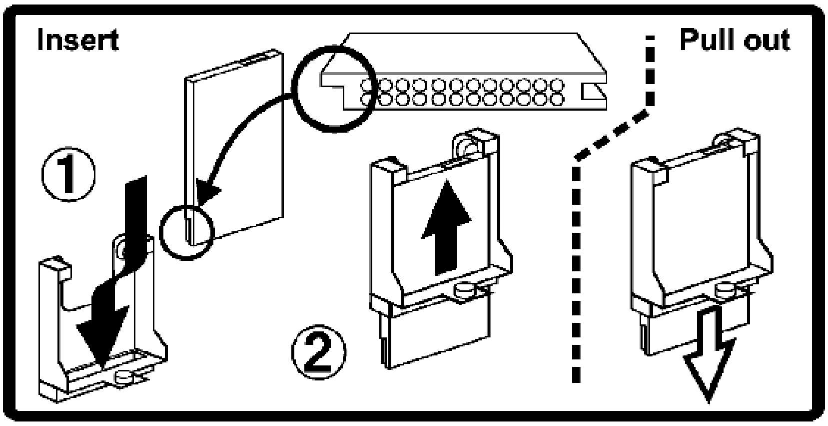
The attachment guide is 1.6mm thick. Pay attention for the length of the screws when you assemble them.



2) How to mount the card

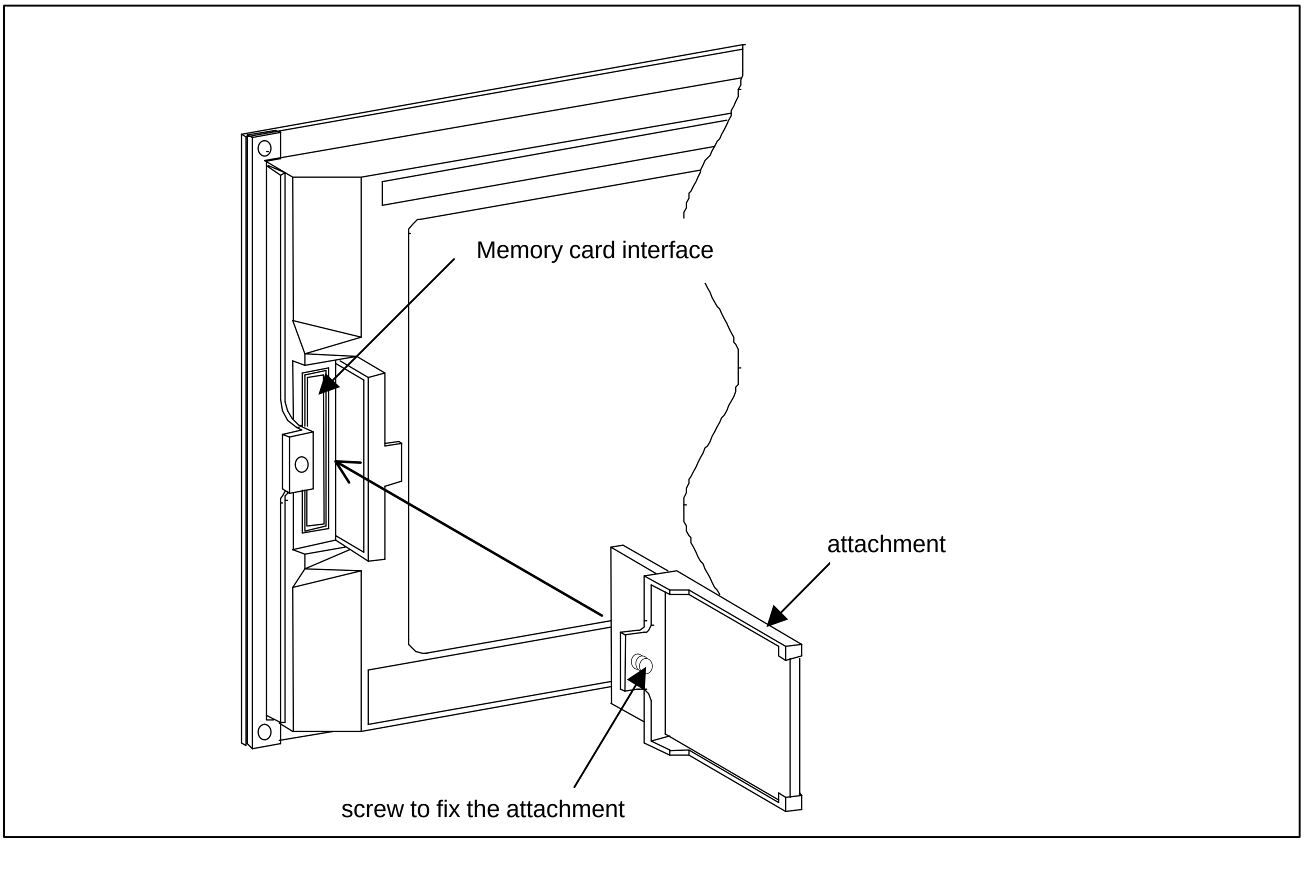
(a) Insert the card to slit of the attachment. Please pay attention to the direction of the card. (Please mach the direction of ditch on the card.)

(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 The memory card interface where is the left side of the screen of the display unit.
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.

4.8.6 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 tool, use machine lock.
There are two types of machine lock: all–axis machine lock, which stops the movement along all axes, and specified–axis machine lock, which stops the movement along specified axes only. In addition, auxiliary function lock, which disables M, S, and T commands, is available for checking a program together with machine lock.

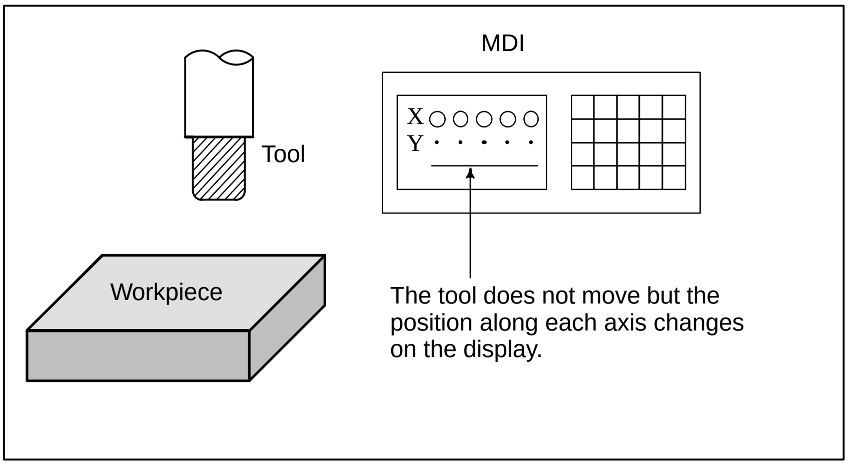


Fig. 5.1 Machine lock

Procedure for Machine Lock and Auxiliary Function Lock

Procedure

- **Machine Lock**

Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving.
Some machines have a machine lock switch for each axis. On such machines, press the machine lock switches for the axes along which the tool is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.

> **WARNING**
> Performing automatic operation with machine lock applied may cause the positional relationship between the workpiece coordinates and machine coordinates to differ from that prior to automatic operation. In such a case, reset the workpiece coordinate system by specifying coordinate system setting or by performing manual reference position return.

- **Auxiliary Function Lock**

Press the auxiliary function lock switch on the operator's panel. M, S, and T codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Restrictions

- **M, S, T command by only machine lock**

M, S, and T commands are executed in the machine lock state.

- **Reference position return under Machine Lock**

When a G28 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

- **M codes not locked by auxiliary function lock**

M00, M01, M02, M30, M98, M99, and M198 (sub program call 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.

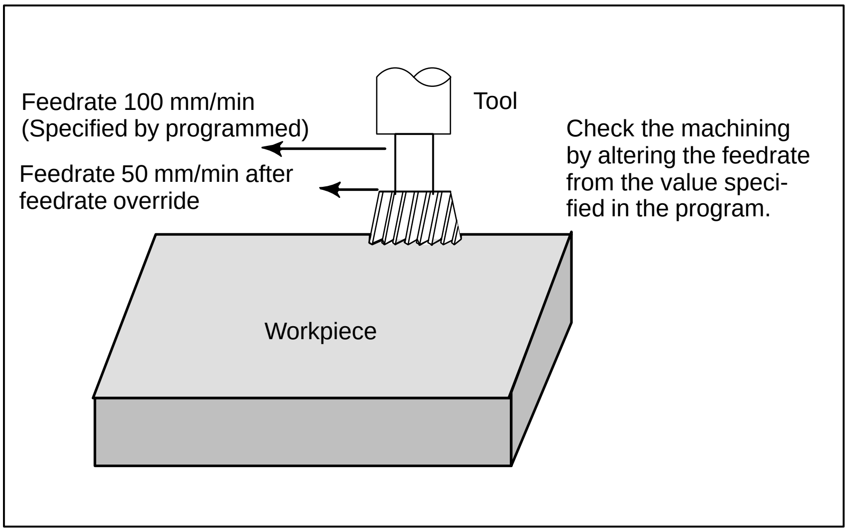
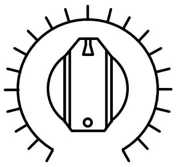


Fig. 5.2 Feedrate override

Procedure for Feedrate Override

Procedure



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 (25%, 50%, 75% and 100%) can be applied to the rapid traverse rate.

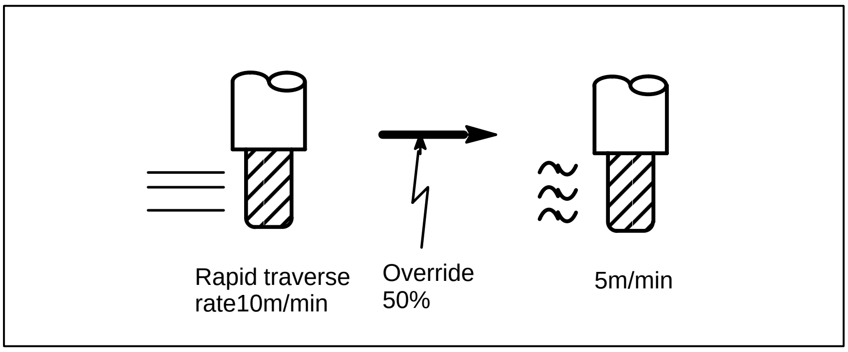
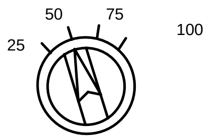


Fig. 5.3 Rapid traverse override

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 pattern function (G26, G76, G77, G78, G79)
3) Rapid traverse during positioning to the first punch point in nibbling function (G68, G69, and nibbling by M function)
4) Rapid traverse by positioning & punch off (G70)
5) Rapid traverse in automatic repositioning (G75)
6) Rapid traverse in automatic reference position return (G28)

For the T axis, the rate is 100% of the T–axis rapid traverse rate when the switch is set to 100 or 75, while the rate is 50% of the T–axis rapid traverse rate when the switch is set to 50 or 25.
However, the override is always set to 100% by parameter setting TROC (No. 0423#1).

WARNING

1 For the manual rapid traverse and rapid traverse in manual reference point return, the rapid traverse override function is ineffective.
2 For the rapid traverse attained to each pitch from the first punch point to the last punch point in nibbling function, the rapid traverse override is ineffective, and it is always fixed to 100%.
3 Since whether the rapid traverse override is effected or not is judged when data are read from the external device or memory into buffer storage, the traverse rate in the block being executed or the block already in the buffer storage remain unchanged even when the rapid traverse override switch has been changed.

CAUTION

The rapid traverse rates in manual and automatic operation modes are separately settable for each axis.

5.4 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state taht the workpiece is removed from the table.

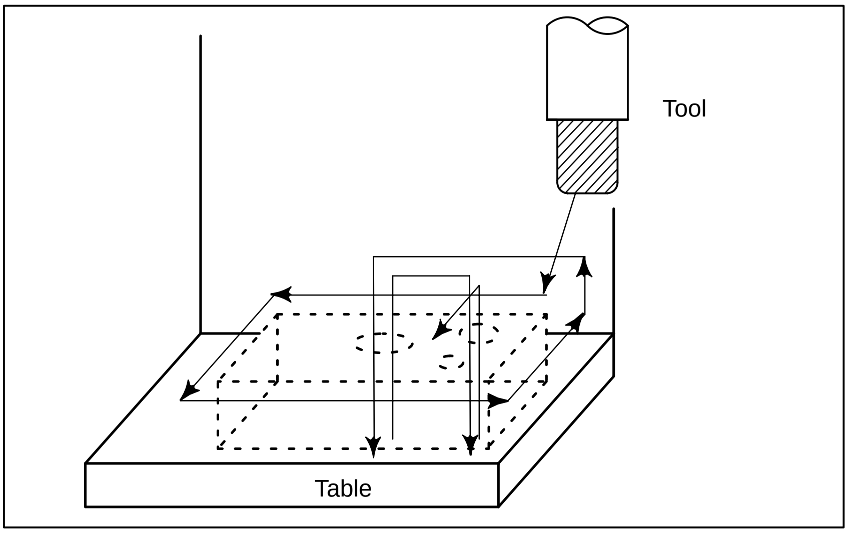


Fig. 5.4 Dry run

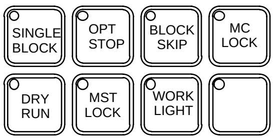
Procedure for Dry Run

Procedure

Press the dry run switch on the machine operator's panel during automatic operation.
The tool moves at the feedrate specified in a parameter. The rapid traverse switch can also be used for changing the feedrate.
Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

- **Dry run feedrate**



The dry run feedrate changes as shown in the table below according to the rapid traverse switch and parameters.

Rapid traverse button	Program command	
	Rapid traverse	**Feed**
ON	Rapid traverse rate	Dry run feedrate × Max.JV
OFF	Dry run speed × JV,or rapid traverse rate ***1)**	Dry run feedrate × JV

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.

5.5 SINGLE BLOCK

Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.

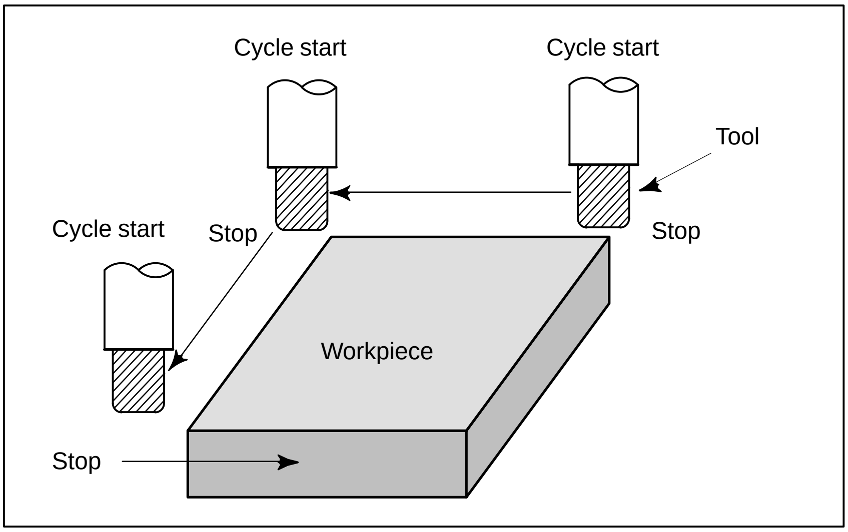


Fig. 5.5 Single block

Procedure for Single block

Procedure

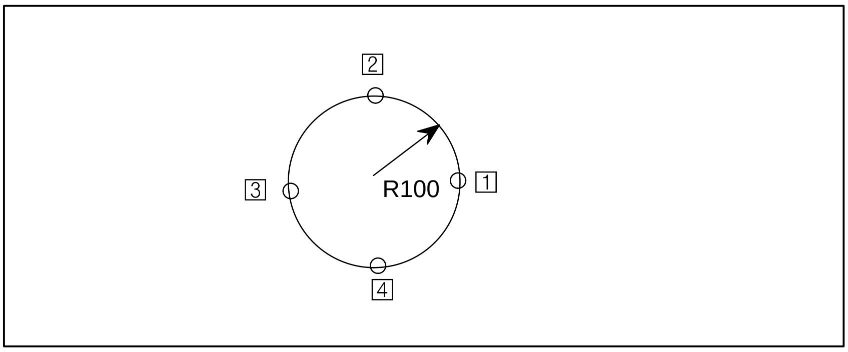
1. Press the single block switch on the machine operator's panel. The execution of the program is stopped after the current block is executed.
2. Press the cycle start button to execute the next block. The tool stops after the block is executed.

   Refer to the appropriate manual provided by the machine tool builder for single block execution.

Explanation

- **Single block during a pattern function**

Example) G26I100.0J0K4 ;



When single block stop has been made in 1, 2, 3, the feed hold lamp lights.
When single block stop has been made in 4, the feed hold lamp does not light.

WARNING

1 If a pattern function (G26, G76, G77, G78, G79) is executed by the single block operation, the single block stop is made after each positioning and punching to respective punch points.
The feed hold lamp lights except when the single block stop is made at the last punch point.

2 If nibbling function (G68, G69) is executed by the single block operation, the single block stop is made after punching to the last punch point.
In nibbling is executed by M function, the single block stop is made after executing from the block including the M code for nibbling mode to the block including the M code for nibbling mode cancel.

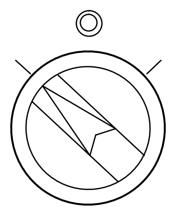
3 If automatic repositioning (G75) is executed by the single block operation, the single block stop is made after a series of motions for the repositioning has been fully completed.

- **Subprogram call and single block**

Single block stop is not performed in a block containing M98P_;. M99; or G65.
However, single block stop is even performed in a block with M98P_ or M99 command, if the block contains an address other than O, N or P.

5.6 TOOL SELECTION

This switch selects whether the T code command is effective or not in the RMT, memory, and MDI modes.

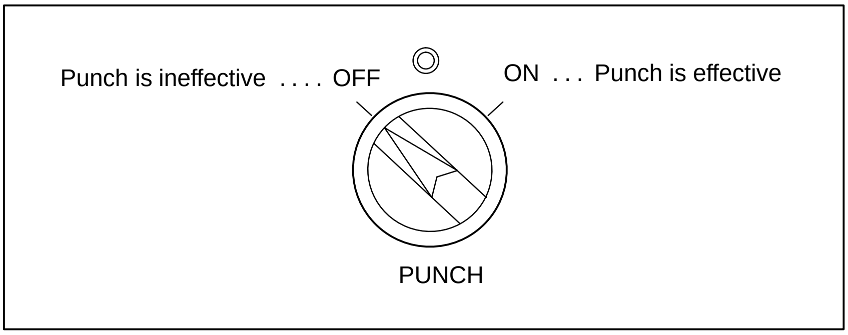


WARNING

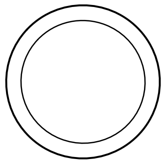
Since whether the T-code function is effective or not is judged when data are read from the external device or memory into the buffer storage, this function is not effective for the block which has already been read, even if this switch is selected.

5.7 PUNCH

This function makes punch (including nibbling) ineffective in a block where punching is made by press motion during the RMT or memory mode operation.



5.8 MANUAL PUNCH



Manual Punch

When depressing this button, punching is made by press motion. When depressing this button again after releasing it once, punching is made again.
Generally, when this button is depressed, while the punch ON/OFF switch in 6.8 is being set to ON, punching executed, while if the switch is set to OFF, punching is not executed.

WARNING
No punching is executed by depressing this button when the cycle start lamp is lighting or during manual axis movement. Since effective conditions of this button depend upon the machine tool builders, refer to the machine tool builder’s manual.

6 SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the tool from exceeding the stroke ends, Overtravel check and Stroke check are available. This chapter describes emergency stop., overtravel check, and stroke check.

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

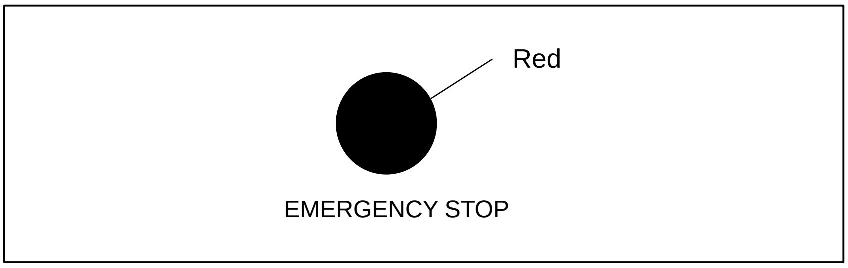


Fig. 6.1 Emergency stop

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

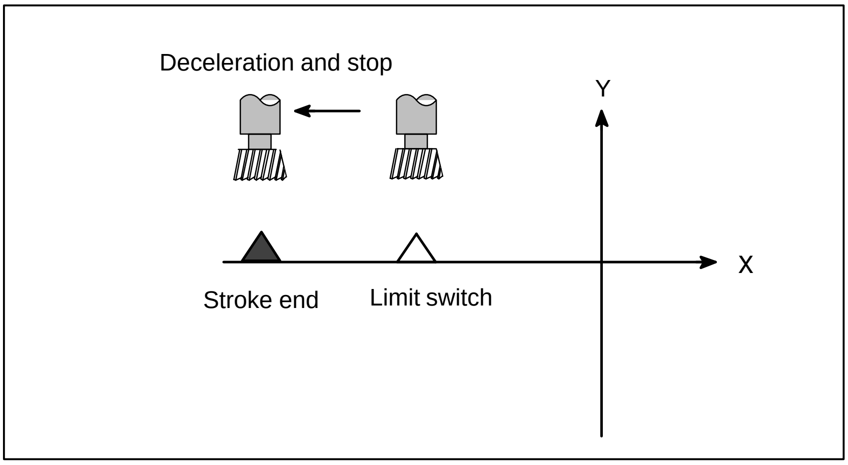


Fig. 6.2 Overtravel

Explanation

- **Overtravel during automatic operation**

When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an overtravel alarm is displayed.

- **Overtravel during manual operation**

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

- **Releasing overtravel**

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

Alarm

No.	Message	Description
506	Overtravel: +n	The tool has exceeded the hardware–specified overtravel limit along the positive nth axis (n: 1 to 8).
507	Overtravel: –n	The tool has exceeded the hardware–specified overtravel limit along the negative nth axis (n: 1 to 8).

6.3 STORED STROKE CHECK

Two areas which the tool cannot enter can be specified with stored stroke limit 1 and stored stroke check 2.

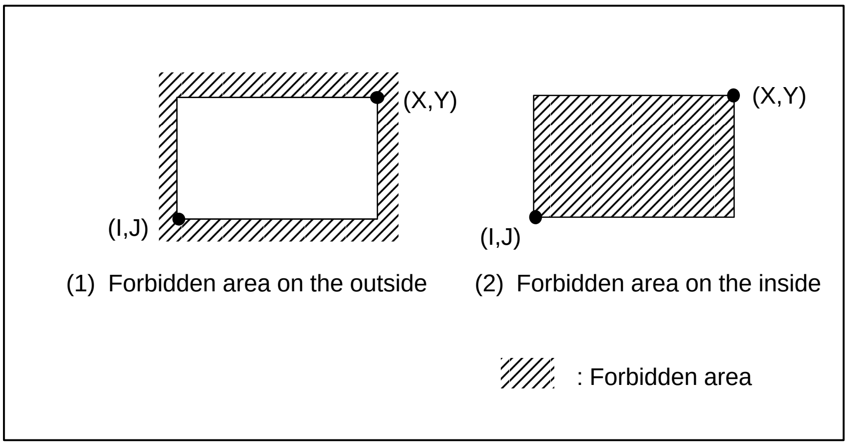


Fig. 6.3 (a) Stroke check

When the tool exceeds a stored stroke check, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

Explanation

- **Stored stroke check 1**

Parameters (Nos. 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**

Parameters (Nos. 1322, 1323) or commands set these boundaries. Inside or outside the area of the limit can be set as the forbidden area. Parameter OUT (No. 1300#0) selects either inside or outside as the forbidden area.

In case of program command a G22 command forbids the tool to enter the forbidden area, and a G23 command permits the tool to enter the forbidden area. Each of G22; and G23; should be commanded independently of another commands in a block.
The command below creates or changes the forbidden area:

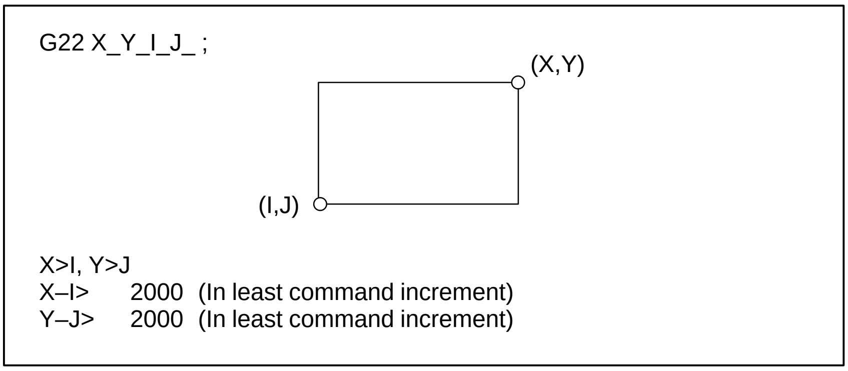


Fig. 6.3 (b) Creating or changing the forbidden area using a program

When setting the area by parameters, points A and B in the figure below must be set.

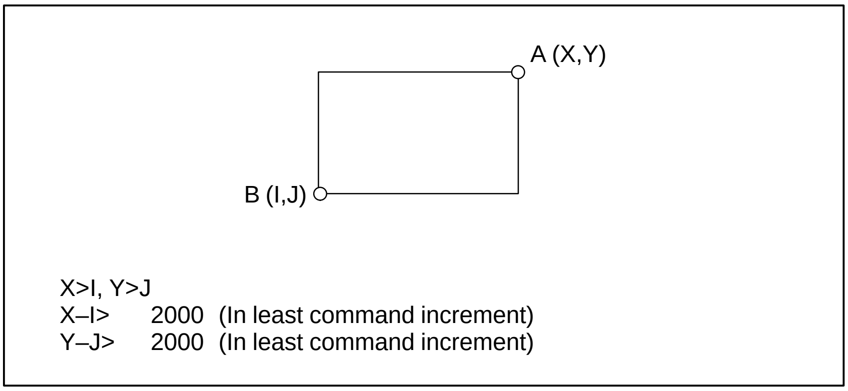


Fig. 6.3 (c) Creating or changing the forbidden area using a parameters

In limit 2, even if you mistake the order of the coordinate value of the two points, a rectangular, with the two points being the apexes, will be set as the area.
When you set the forbidden area through parameters (Nos. 1322, 1323), the data should be specified by the distance from the reference position in the least command increment. (Output increment)
If it is set by a G22 command, specify the data by the distance from the reference position in the least input increment (Input increment.) The programmed data are then converted into the numerical values in the least command increment, and the values are set as the parameters.

- **Forbidden area over–lapping**

Area can be set in piles.

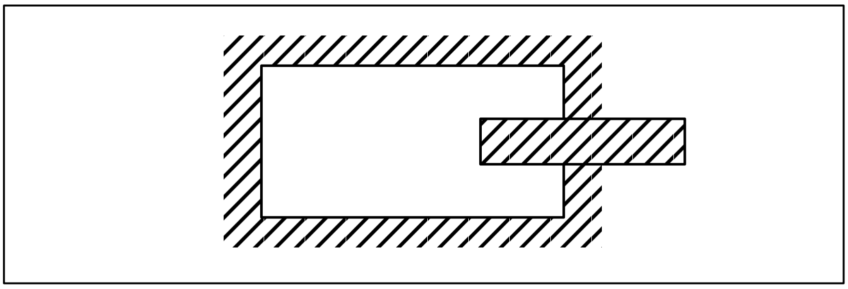


Fig. 6.3 (d) Setting the forbidden area over lapping

Unnecessary limits should be set beyond the machine stroke.

- **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 tool enters the specified inhibited area by up to L (mm). Bit 7 (BFA) of parameter No. 1300 can be used to stop the tool when it reaches a point L mm short of the specified area. In this case, the tool 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 check 2).

- **Releasing the alarms**

When the tool has become unmovable in the forbidden area, push the emergency stop button to release the forbidden condition and move the tool out of the forbidden area in the G23 mode; then, if the setting is wrong, correct it and perform the reference position return again.

- **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.

WARNING
In setting a forbidden area, if the two points to be set are the same, the area is as follows:
(1) When the forbidden area is limit 1, all areas are forbidden areas.
(2) When the forbidden area is limit 2, all areas are movable areas.
(3) Neither stored stroke check 1 nor 2 exist in T–axis and C–axis.

- **Timing for displaying an alarm**

Parameter BFA (bit 7 of No. 1300) selects whether an alarm is displayed immediately before the tool enters the forbidden area or immediately after the tool has entered the forbidden area.

Alarms

Number	Message	Contents
500	OVER TRAVEL: +n	Exceeded the n–th axis (1–8) + side stored stroke check I.
501	OVER TRAVEL: –n	Exceeded the n–th axis (1–8) – side stored stroke check I.
502	OVER TRAVEL: +n	Exceeded the n–th axis (1–8) + side stored stroke check II.
503	OVER TRAVEL: –n	Exceeded the n–th axis (1–8) – side stored stroke check II.

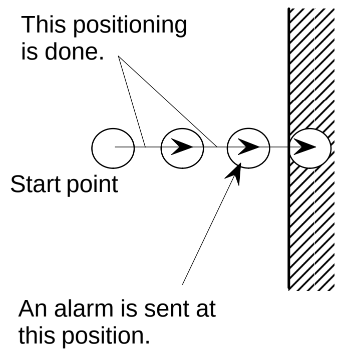
6.4 STROKE CHECK BEFORE MOVEMENT

When the tool starts to move for positioning by rapid traverse (G00) of automatic operation, this function checks the end point coordinates from the machine's current position and the specified amount of movement. It checks if the tool will enter a forbidden area of stored stroke limit 1 and if it does, it stops the tool immediately after the start of movement for that block and displays an alarm.

WARNING

1 This function checks whether the end point coordinates of the tool enter a forbidden area or not, but does not check the path of the tool during the move command.
The stored stroke check 1, or the stored stroke check 2, however, will output an alarm when the tool reaches a forbidden area.

2 The previous check in pattern function and nibbling function blocks causes an alarm, if the end point is located in a forbidden area of the stored stroke check 1 when individual positioning is started.
(Example) In case of line at angle (G76)



3 Valid only for the X and Y axes.

6.5 SAFETY ZONE CHECK

This is the safety function to set the safety zone for protecting the workpiece holder that holds the workpiece set on the carriage, and disable punching in that area or forbid the tool to approach thereinto.

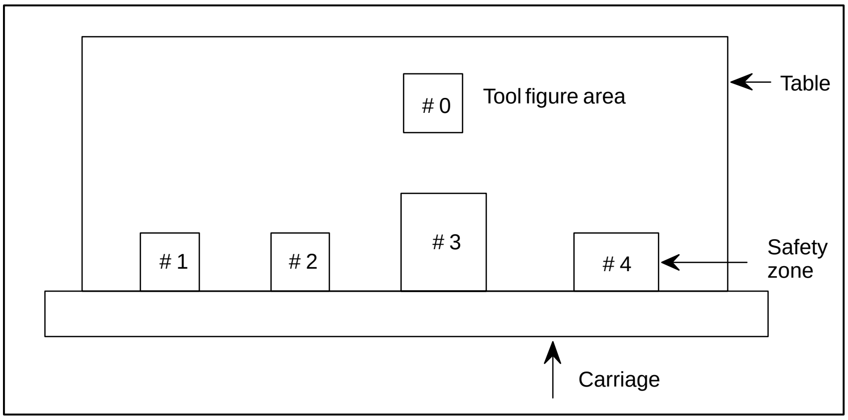


Fig. 6.5

This function permits to set tool figure area (#0) and up to four safety zones (#1 - #4), as shown above.
Type A and type B of safety zone check are prepared.

6.5.1 Punch Forbidden Area and Approach Forbidden Area (Type A)

The safety zone is settable in two types, punch forbidden area and approach forbidden area, that are set by the parameter SZ1 to SZ4 (No. 16501#0 - #3) shown below.

1) Punch forbidden area
   When the tool figure area goes into the safety zone and the punching is commanded, an alarm (Nos. 4800 to 4803) is given to disable punching. In the case of positioning & punching command in the automatic operation mode, when the end point of positioning is in the punch forbidden area, an alarm is given without moving the axis. (Previous check)
   In the case of move command without punching, the tool figure area can go into the punch forbidden area, but manual punching is impossible after going into this area.

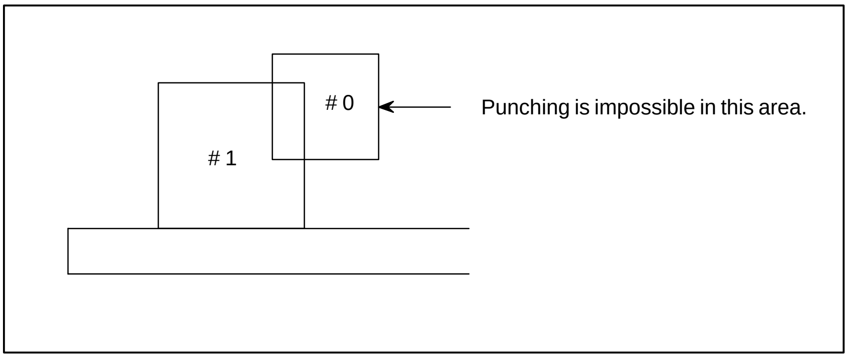


Fig. 6.5.1 (a)

2) Approach forbidden area
   The tool figure area can not go into the safety zone. When the tool figure area approaches into the safety zone by the move command, the axis is immediately stopped and an alarm (Nos. 4810 - 4837) is given. This is valid in either manual or automatic operation mode.

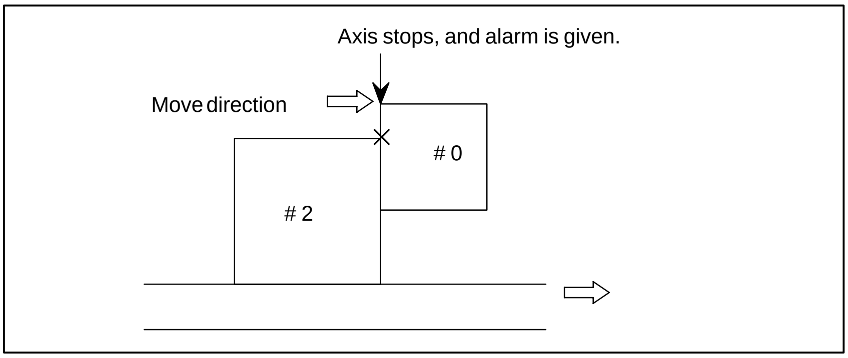


Fig. 6.5.1 (b)

6.5.2 Punch Forbidden Area and Approach Forbidden Area (Type B)

General

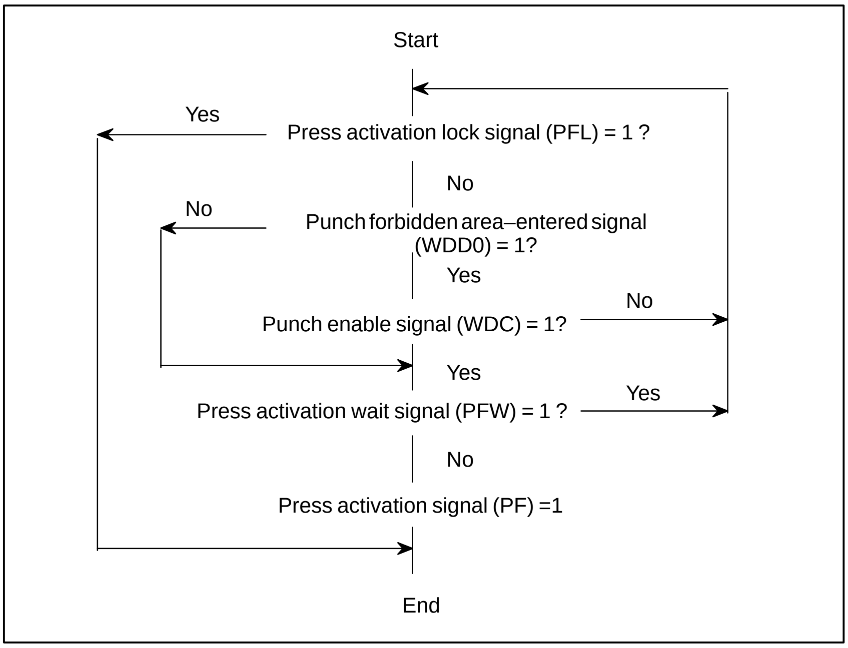
By setting bit 0 (SF0) of parameter No. 16500, the type B safety zone check can be selected. With type B, no alarm is issued even if a tool enters a safety zone; after confirming the safety of the situation, the operator can perform a punch operation, or can position the tool to the next punching position without performing punching.

With type B, all safety zones are handled as punch forbidden areas.

Punch forbidden area

If the tool enters a safety zone, the punch–forbidden area–entered signal WDD0 (F231#7), used to provide notification that the tool has entered a punch forbidden area, is set to 1 in the punch block. When punching is to be performed in a punch–forbidden area, punch–enable signal WDC (G232#4) is set to 1 after the operator confirms that punching can be performed safely. After detecting the WDC signal, the CNC sets the PF signal to 1.

When punching is not to be performed in a punch forbidden area, the press activation lock signal PFL (G230#0) is set to 1. Upon detecting the PFL signal, the CNC moves on to execute the next block.



WARNING

1 If the tool enters a punch forbidden area during nibbling, the WDD0 signal is set to 1 one punching position before the tool enters punch forbidden area.

2 With type B, the setting of SZ1 to SZ4 (bits 0 to 3 of parameter No. 16501) is ignored.

6.5.3 Setting the Safety Zone

Set the machine coordinate value when the workpiece holder is positioned at the tool center (punching position), in the parameters 16505 - 16516 in output units.

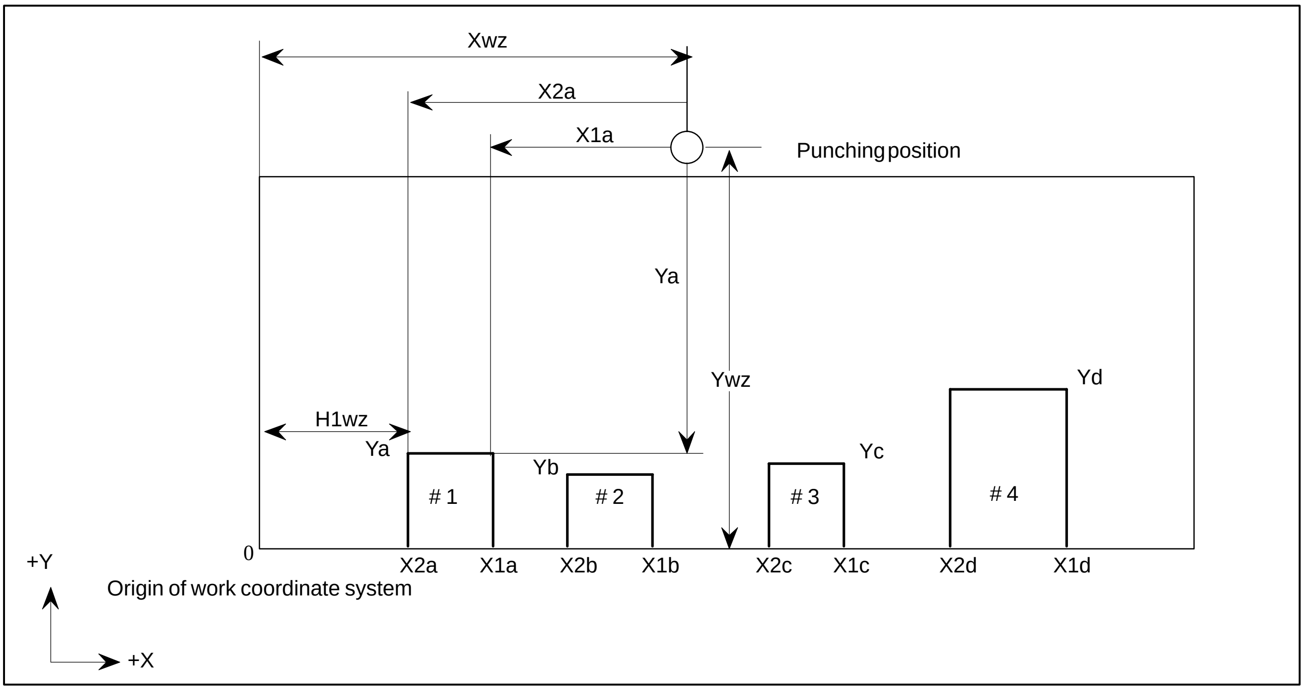


Fig. 6.5.3

Regarding #1 in Fig. 6.5.3, the safety zone is specified at both ends (X1a, X2b) for the X-axis direction, and at the forward end (Ya) of the workpiece holder for the Y-axis direction. The specifying method is the same as for #2, #3 and #4.
Considering the setting value in the work coordinate system, it is the value obtained by subtracting the set value of automatic coordinate system from the workpiece holder position in the work coordinate system.
For example, set value of X2a is as follows in Fig. 6.5.3.

Set value (X2a) = (H1wz) – (Xwz)

Set four safety zones to be arranged sequentially in the order of #1, #2, #3 and #4 from the origin to positive of the X-axis.

6.5.4 Setting the Tool Shape Area

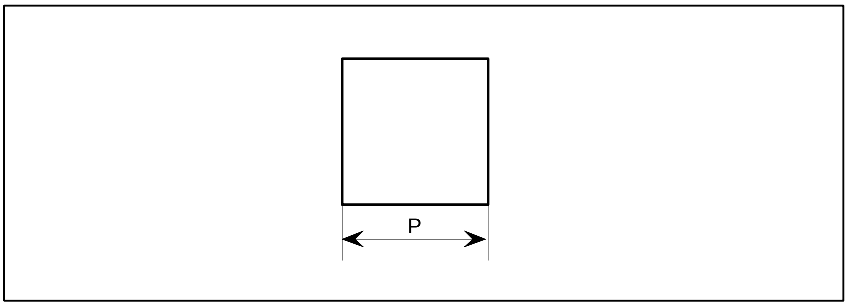


Fig. 6.5.4 (a)

The specification of the area of tool figure sets the size in the X direction and Y direction of the tool by the parameter (No.16517 to 16532 and No.16551 to 16558).
The setting unit is output unit.
12 kinds of or less tool figure can be set.

The tool shape area can be changed by using the programmable parameter input function (G10). Therefore, when multiple tools are used, it is possible to specify the tool shape area meeting the tool No. (Txx).

When there are an area of the punched tool and an area with the laser oscillator for special, first set two safety zones for the workpiece holder. Reserve the remaining two safety zones for the imaginary workpiece holder.
When the workpiece holder (a) approaches to the laser oscillator in Fig. 6.5.4 (b) below, it is judged as the approaching of the tool area to the imaginary safety zone.

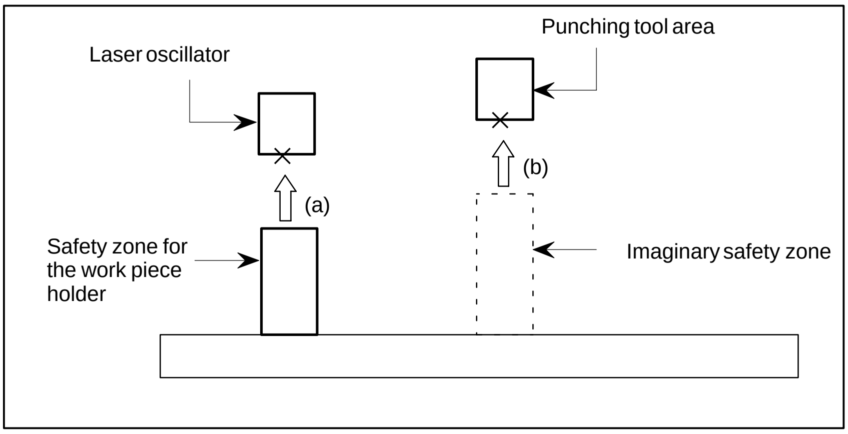


Fig. 6.5.4 (b)

6.5.5 Automatic Setting of the Safety Zone

The detector on the machine automatically detects the positions of the workpiece holders mounted on the carriage. Values representing the detected positions are then set in the safety zone parameters.

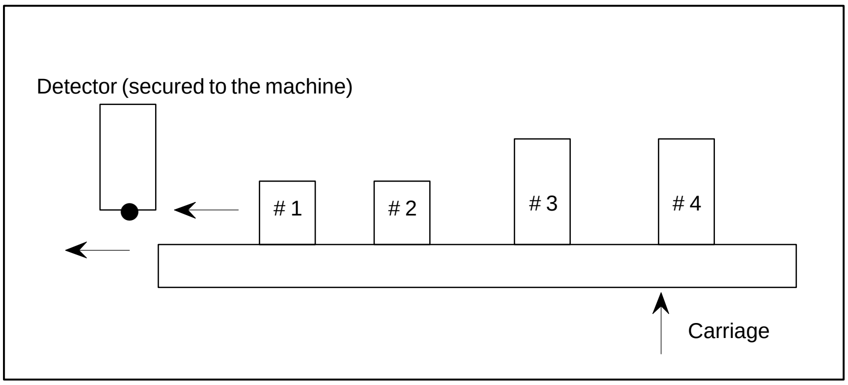


Fig. 6.5.5 (a)

The signal for detecting the position of the workpiece holders turns on and off as workpiece holders #1 to #4 pass the detector as shown in Fig. 6.5.5 (a). The safety zone is determined from the signal. The position on the X axis when the signal goes high is regarded as one end of a workpiece holder, and the position when the signal goes low is regarded as the other end.

Workpiece holder detection command format

G32X_x_ F_f_ P_p_ Q _q_ ;

G32 is specified to detect the positions of workpiece holders. Before G32 can be specified, the workpiece holder position detector must be activated.

Before detection is started, the X-axis move command must be specified following address X. Either the positive or negative move direction is allowed. If a positive move command is specified, the position at which detection starts must be in the negative side with respect to workpiece holder #1. Sufficient distance must be kept between the position at which detection starts and workpiece holder #1 to enable stability of speed before workpiece holder #1 is reached. Workpiece holders #1, #2, #3, and #4 are detected in that order. If a negative move command is specified, the workpiece holders are detected in the reverse order starting from #4. The other conditions are the same as those used for a positive move command.

The feed rate during detection is specified with F, in the same way as for ordinary interpolation.

The position of a workpiece holder is obtained from the machine positions indicated by the rising and falling edges of the workpiece holder detection signal. There is a difference between the indicated machine position and actual machine position because of an error such as a lag in the servo system. This error must be compensated with numeric values following addresses P and Q. The compensating value for the position on the rising edge of the workpiece holder detection signal is set following address P, and the compensating value for the position on the falling edge is set following address Q.

The lag in the servo system is obtained from the following equations:

$\Delta E = T1 \times F + T2 \times F$ (exponential function acceleration/deceleration)

$\Delta E = 1/2T1 \times F + T2 \times F$ (linear acceleration/deceleration)

where,

ΔE : Lag in the servo system
T1 : Time constant for automatic acceleration/deceleration
T2 : Servo time constant
F : Feed rate

The sign of the compensating value is positive when compensation is made in the opposite direction to the move direction specified by the detection command.

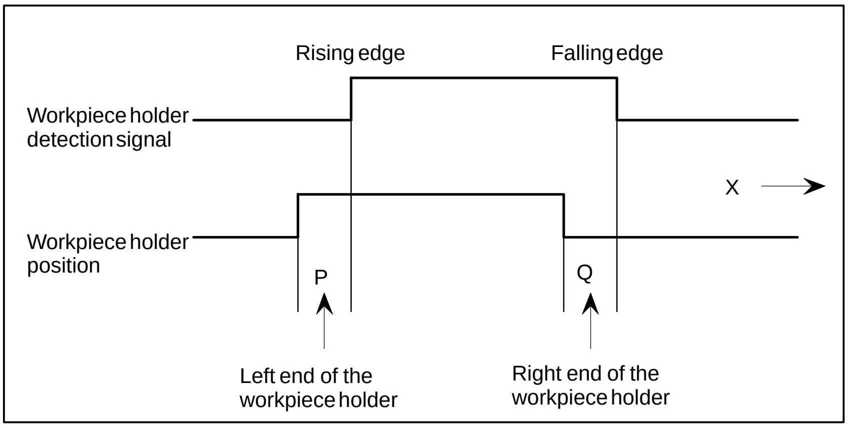


Fig. 6.5.5 (b)

6.5.6 Displaying the Safety Zones and Tool Zone

After safety zone values are set automatically, they can be displayed on the safety zone screen as shown below. With this screen, the user can check whether the set values are valid.

Screen

Type A

```
SAFETY ZONE (ABSOLUTE)                 O0017 N01234
 AREA #1                 AREA #3
  X2=   100.000           X2=  1000.000
  X1=   200.000           X1=  1150.000
  Y =   100.000           Y =   110.000
 AREA #2                 AREA #4
  X2=   500.000           X2=  1400.000
  X1=   600.000           X1=  1550.000
  Y =   100.000           Y =   110.000
 TOOL ZONE
  X =     5.000
  Y =    10.000

)_
MEM **** *** ***         11:32:41
[ TOOL ][      ][ SAFETY ][       ][ (OPRT) ]
```

Screen

Type B

```
SAFETY ZONE (ABSOLUTE)                 O0017 N01234
 AREA #1                 AREA #3
  W =   100.000           W =   100.000
  X =   200.000           X =  1150.000
  Y =   100.000           Y =   110.000
 AREA #2                 AREA #4
  W =   100.000           W =   100.000
  X =   600.000           X =  1550.000
  Y =   100.000           Y =   110.000
 TOOL ZONE               ZONE NUMBER
  X =     5.000           N = 2
  Y =    10.000

)_
MEM **** *** ***         11:32:41
[ TOOL ][      ][ SAFETY ][       ][ (OPRT) ]
```

NOTE

The display items of type B are as follows:

W : Workpiece holder width

X : Workpiece holder central position relative to the tool center

Y : Workpiece holder tip position relative to the tool center

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 50 previous alarms can be stored and displayed on the screen (alarm history display).
The system may sometimes seem to be at a halt, although no alarm is displayed. In this case, the system may be performing some processing. The state of the system can be checked using the self–diagnostic function.

7.1 ALARM DISPLAY

Explanations

- **Alarm screen**

When an alarm occurs, the alarm screen appears.

```
ALARM  MESSAGE                              O0000  N00000

     100     PARAMETER WRITE ENABLE
     510     OVER TRAVEL    :+1
     520     OVER TRAVEL    :+2
     530     OVER TRAVEL    :+3




                                              S    0  T0000
 MDI   ****   ***   ***             18 : 52 : 05
(ARALM)(  MSG  )( HISTRY )(        )(        )
```

- **Another method for alarm displays**

In some cases, the alarm screen does not appear, but an ALM is displayed at the bottom of the screen.

```
 PARAMETER (RS232C INTERFACE)          O1000  N00010

0100  ENS                        NCR         CTV
        0     0     0     0     0     0     0      0
0101  NFD                        ASI                SB2
        0     0     0     0     0     0     0      1
0102  DEVICE NUM. (CH0)                            2
0103  BAUDRATE (CH0)                              10
0111  NFD                        ASI                SB2
        0     0     0     0     0     0     0      0
0112  DEVICE NUM. (CH1)                            0
0113  BAUDRATE (CH1)                               0

)_                                       S       0 T0000
 MEM   ****   ***   ***   ALM  08 : 41 : 27
(NO.SRH)(  ON:1  )( OFF:0 )(+INPUT )( INPUT )
```

In this case, display the alarm screen as follows:

1. Press the function key MESSAGE .
2. Press the chapter selection soft key **[ALARM]**.

- **Reset of the alarm**

Error codes and messages indicate the cause of an alarm. To recover from an alarm, eliminate the cause and press the reset key.

- **Error codes**

The error codes are classified as follows:
No. 000 to 255: PS alarms (Program errors) (*)
No. 300 to 349: Absolute pulse coder (APC) alarms
No. 350 and 399: Serial pulse coder (SPC) alarms
No. 400 to 499: Servo alarms
No. 500 to 599: Overtravel alarms
No. 700 to 739: Overheat alarms
No. 900 to 999: System alarms
No. 4500 to 4599: Punch press alarm
No. 5000 and 5999: PS alarms (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 alarm list in the appendix for details of the alarms.

7.2 ALARM HISTORY DISPLAY

Up to 50 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 number

3 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 : 01
  (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, S, T function being executed
001	MOTION	Move command in automatic operation being executed
002	DWELL	Dwell being executed
003	IN–POSITION CHECK	In–position check being executed
004	FEEDRATE OVERRIDE 0%	Cutting feed override 0%
005	INTERLOCK/START–LOCK	Interlock ON
006	SPINDLE SPEED ARRIVAL CHECK	Waiting for spindle speed arrival signal to turn on
010	PUNCHING	Data being output via reader puncher interface
011	READING	Data being input via reader puncher interface
012	WAITING FOR (UN) CLAMP	Waiting for index table clamp/unclamp before B axis index table indexing start/after B axis index table indexing end to complete
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

Diagnostic numbers 020 to 025 indicate states when an automatic operation is stopped or paused.

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 indicates 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

Diagnostic numbers 990 to 998 indicate Press condition.

Table 7.3 (c) Press condition

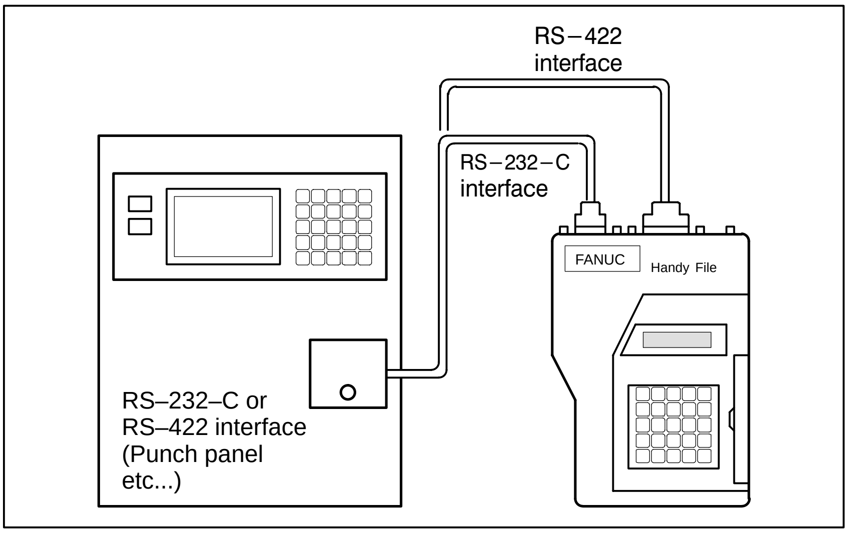
No.	Display	Internal status when 1 is displayed
990	WAITING FOR SIGNAL PFW OFF	Waiting for turning off press start wait signal PFW.
991	WAITING FOR 1–CYCLE *PE OFF	Waiting for press stop to an one–cycle press motion.
992	WAITING FOR 1–CYCLE *PFIN OFF	Waiting for completion of punching to an one–cycle press motion.
993	WAITING FOR NIBBLING *PE OFF	Waiting for press stop to a nibbling motion.
994	WAITING FOR NIBBLING *NFIN OFF	Waiting for completion of punching to a nibbling motion.
995	WAITING FOR T COMMAND	Press operation is not executable, because the punch command was given before executing the T command once after the cycle operation signal OP has been turned from off to on.
996	MANUAL PRESS	Manual press is executing.
997	WAITING FOR PF	Waiting that absolute value of X and Y axis position deviation value becomes the value of parameter No. 16010 and below.
998	REMAINING U/V MACRO CHARACTER	Remaining U/V macro character number.

8 DATA INPUT/OUTPUT

The Handy File or other external input/output devices are used to transfer data between a floppy disk or memory card and the CNC.
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, and the
In this manual, an 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.

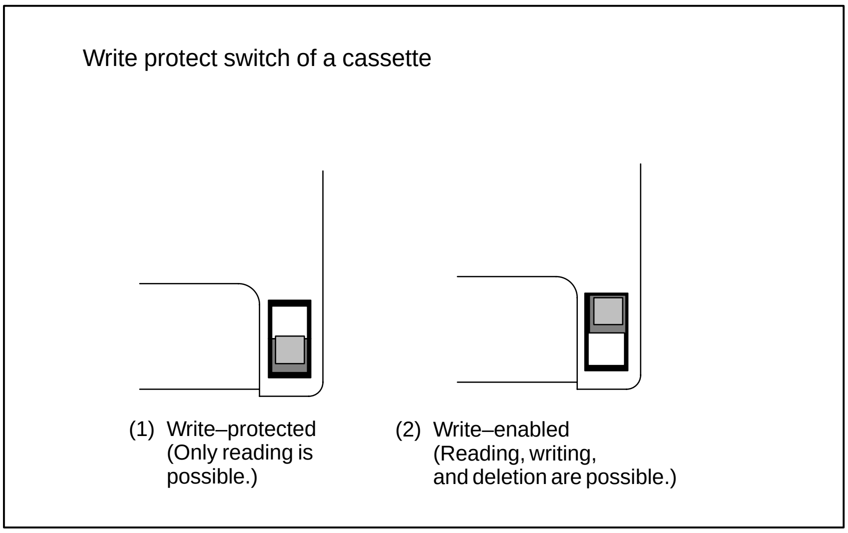


Fig. 8.1 Protect swtich

- **Writing memo**

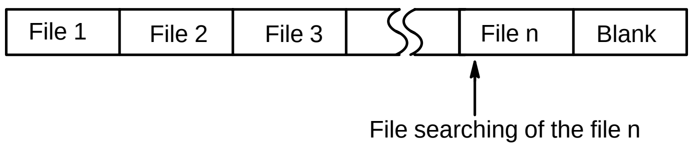
Once written in the cassette or card, data can subsequently be read out by correspondence between the data contents and file numbers. This correspondence cannot be verified, unless the data contents and file numbers are output to the CNC and displayed. The data contents can be displayed with display 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 .

3 Press soft key **[(OPRT)]**

4 Press the rightmost soft key ▷ (next–menu key).

5 Enter address N.

6 Enter the number of the file to search for.

·N0
The beginning of the cassette or card is searched.
·One of N1 to N9999
Of the file Nos. 1 to 9999, a designated file is searched.
·N–9999
The file next to that accessed just before is searched.
·N–9998
When N–9998 is designated, N–9999 is automatically inserted each time a file is input or output. This condition is reset by the designation of N1,N1 to 9999, or N–9999 or reset.

7 Press soft keys **[F SRH]** and **[EXEC]**
The specified file is searched for.

Explanation

- **File search by N–9999**

The same result is obtained both by sequentially searching the files by specifying Nos. N1 to N9999 and by first searching one of N1 to N9999 and then using the N–9999 searching method. The searching time is shorter in the latter case.

Alarm

No.	Description
86	The ready signal (DR) of an input/output device is off. An alarm is not immediately indicated in the CNC even when an alarm occurs during head searching (when a file is not found, or the like). An alarm is given when the input/output operation is performed after that. This alarm is also raised when N1 is specified for writing data to an empty floppy. (In this case, specify N0.)

8.3 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
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]**
   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 Section 8.2.

4 Press function key PROG

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.

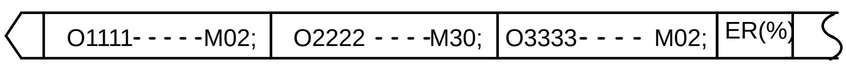
Explanations

- **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]

- **Appending a program**

An entered program can be appended to a previously registered program.

Registered program	Entered program	Resulting program
O1234 ;	O5678 ;	O1234 ;
□□□□□□□ ;	○○○○○○ ;	□□□□□□□ ;
□□□□□ ;	○○○○○ ;	□□□□□ ;
□□□□ ;	○○○○ ;	□□□□ ;
□□□ ;	○○○ ;	□□□ ;
%	%	;
		○○○○○○ ;
		○○○○○ ;
		○○○○ ;
		○○○ ;
		%

In the above example, the contents of program O5678 are appended to program O1234. O5678 is not registered as a program number.
To append a program, press the **[READ]** soft key without first entering a program number, in step 8, then press the **[CHAIN]** soft key, followed by **[EXEC]**.

- In all program input mode, the entire contents of the program are appended, with the exception of the O number .
- To cancel append mode, press the reset key, or the **[CAN]** or **[STOP]** soft key.

- Immediately after the **[CHAIN]** soft key is pressed, the cursor is positioned to the end of the registered program. After the entered program is appended, the cursor is positioned to the beginning of the resulting program.
- A program cannot be appended if no registered programs exist.

- **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

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 .

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.

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 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.

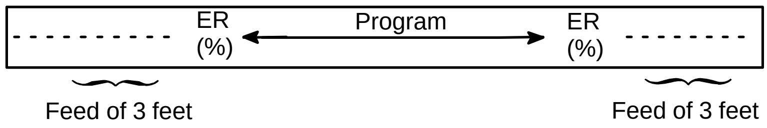
- **Punching programs in the background**

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.

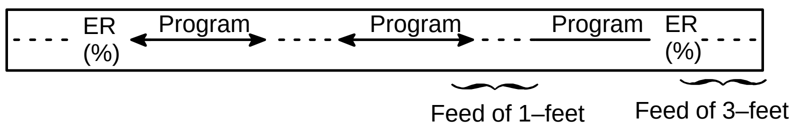
- - - - - - - - - - - - - - - LF CR CR

- **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 section 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 Section 8.2.

4 Press function key OFFSET SETTING .

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 .
5. Press soft key **[(OPRT)]**.
6. Press the rightmost soft key ▷ (next–menu key)
7. Press soft keys **[PUNCH]** and **[EXEC]**.
   Offset data is output in the output format described below.

Explanations

- **Output format**

Output format is as follows:

Format

G10 L11 P_R_;
where P_: Offset No.
R_: Tool compensation amount

The L1 command may be used instead of L11 for format compatibility of the conventional CNC.

- **Output file name**

When the floppy disk directory display function is used, the name of the output file is OFFSET.

8.6 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 Section 8.6.2. When a parameter is loaded which has the same data number as a parameter already registered in the memory, the loaded parameter replaces the existing parameter.

Inputting parameters

Procedure

1 Make sure the input device is ready for reading.

2 When using a floppy, search for the required file according to the procedure in Section 8.2.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key OFFSET SETTING .

5 Press the soft key **[SETING]** for chapter selection, 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.

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 Section 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 Section 8.2.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key OFFSET SETTING .

5 Press the soft key **[SETING]** for chapter selection.

6 Enter 1 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 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 3620 to 3624 and pitch error compensation data must be set correctly to apply pitch error compensation correctly
(See subsection 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:
N10000 . . . . P . . . . ;
N11023 . . . . . 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 Section 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 Input the program according to the procedure in Section 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 chek whether the values of the common variables have been set correctly.

Display of the macro variable screen

· Press function key OFFSET SETTING .

· Press the rightmost soft key (next–menu key).
· Press soft key **[MACRO]**.
· Select a variable with the page keys or numeric keys and soft key **[NO.SRH]**.

Explanations

- **Common variables**

The common variables (#500 to #999) can be input and output.
Common variables #100 to 199 cannot be input or output.

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**

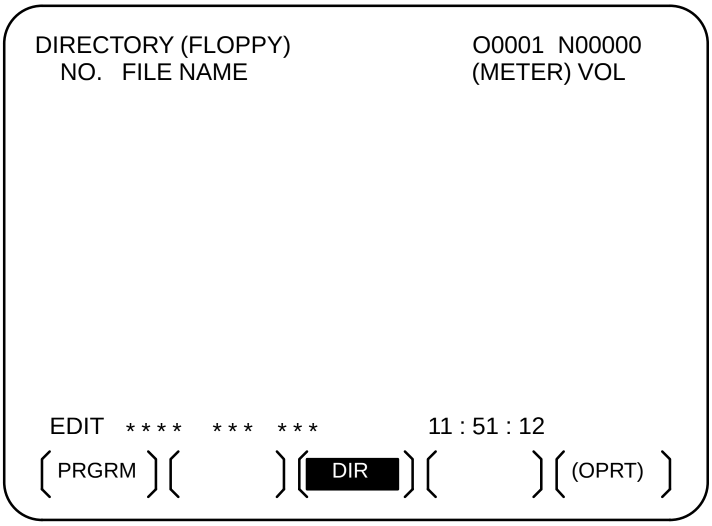
When the floppy disk directory display function is used, the name of the output file is "**MACRO VAR**".

- **Common variable**

The common variables (#500 to #999) can be input and output.
#100 to #199 can be input and output when bit 3 (PV5) of parameter No. 6001 is set to 1.

8.8 DISPLAYING DIRECTORY OF FLOPPY DISK

On the floppy directory display screen, a directory of the FANUC Handy File, FANUC Floppy Cassette, or FANUC FA Card files can be displayed. In addition, those files can be loaded, output, and deleted.



8.8.1 Displaying the Directory

Displaying the directory of floppy disk files

Procedure 1

Use the following procedure to display a directory of all the files stored in a floppy:

1 Press the EDIT switch on the machine operator’s panel.

2 Press function key PROG .

3 Press 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

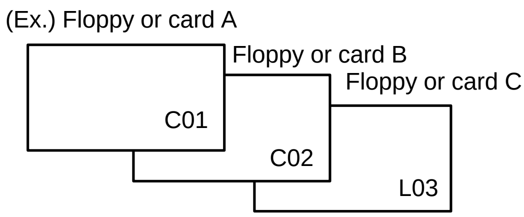
- **Screen fields and their meanings**

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.



C(number)means . . . . CONTINUE
L(number)means . . . . LAST
number . . . . number of floppies or cards

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]**.
If steps 7 and 8 are omitted, the program currently being selected is output.

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).

Limitations

- **Inputting file numbers and program numbers with keys**

If **[F SET]** or **[O SET]** is pressed without key inputting file number and program number, file number or program number shows blank. When 0 is entered for file numbers or program numbers, 1 is displayed.

- **I/O devices**

To use channel 0 ,set a device number in parameter 102.
Set the I/O device number to parameter No. 0112 when cannel 1 is used. Set it to No. 0122 when channel 2 is used.

- **Significant digits**

For the numeral input in the data input area with FILE NO. and PROGRAM NO., only lower 4 digits become valid.

- **Collation**

When the data protection key on the machine operator's panel is ON, no programs are read from the floppy. They are verified against the contents of the memory of the CNC instead.

Alarm

No.	Contents
71	An invalid file number or program number was entered. (Specified program number is not found.)
79	Verification operation found a mismatch between a program loaded into memory and the contents of the floppy
86	The dataset–ready signal (DR) for the input/output device is turned off. (The no file error or duplicate file error occurred on the input/output device because an invalid file number, program number, or file name was entered.

8.9 INPUTTING/ OUTPUTTING TOOL DATA

8.9.1 Inputting Tool Data

The value of a tool data is loaded into the memory of the CNC from a floppy or NC tape. The same format used to output tool data is used for input. See Section 8.9.2. When the value of a tool data is loaded into memory, this value replaces the value of the same tool data already existing in memory.

Inputting Tool Data

Procedure

1 Make sure the input device is ready for reading.

2 When using a floppy, search for the required file according to the procedure in Section 8.2.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key OFFSET SETTING .

5 Press the soft key **[SETTING]** for chapter selection.

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 the rightmost soft key ▷ (next–menu key) two times, then press soft key **[TOOL]**.

8 Press the rightmost soft key ▷ (next–menu key), then press soft key **[T.NUM.]**.

9 Press soft key **[(OPRT)]**.

10 Press the rightmost soft key ▷ (next–menu key).

11 Press soft keys **[READ] [EXEC]**.

12 The input tool data will be displayed on the screen after completion of input operation.

13 Press function key OFFSET SETTING .

14 Press the soft key **[SETTING]** for chapter selection.

15 Enter 0 in response to the prompt for "PARAMETER WRITE (PWE)" in setting data.

16 Release the EMERGENCY STOP button on the machine operator's panel.

8.9.2 Outputting Tool Data

All tool data are output in the defined format from the memory of the CNC to a floppy or NC tape.

Outputting Tool 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 .

5 Press the rightmost soft key ▷ (next–menu key) two times, then press soft key **[TOOL]**.

6 Press the rightmost soft key ▷ (next–menu key), then press soft key **[T.NUM.]**.

7 Press soft key **[(OPRT)]**.

8 Press the rightmost soft key ▷ (next–menu key).

9 Press soft keys **[PUNCH] [EXEC]**.
All tool data are output in the defined format.

Explanations

- **Output format**

Output format is as follows.

```
%
;
N001T   M   X   Y   S   P   C   I   J   K   B   ;
(1) (2) (3) (4) (5) (6) (7) (8) (9) (10) (11) (12)

N002T   M   X   Y   S   P   C   I   J   K   B   ;

. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . ;
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . ;

N136T   M   X   Y   S   P   C   I   J   K   B   ;

. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . ;
. . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . . ;

%
```

Items (1) to (12) are as follows :

(1) Tool registration number
(2) Tool number
(3) Turret position
(4) X–axis offset
(5) Y–axis offset
(6) Number of a tool to be substituted
(7) Number of punch operations
(8) Tool figure for graphic operation
(9) X dimension of a tool for graphic operation
(10) Y dimension of a tool for graphic operation
(11) Tool angle for graphic operation
(12) Tool life value

- **Output file name**

When the floppy disk directory display function is used, the name of the output file is “TOOL DATA”.

8.10 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.

Procedure for Outputting a Program List for a Specified Group

Procedure

1 Display the program list screen for a group of programs.

```
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–EDT] [O–SRH] [     ] [     ] [GROUP]
[     ] [READ] [PUNCH] [     ] [     ]
[AL–GRP] [     ] [STOP] [CAN] [EXEC]
```

3 Press the right–most soft key (continuous menu key).

4 Press the **[PUNCH]** operation soft key.

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.

8.11 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. 8.11 ALL IO screen (when channel 3 is being used for input/output)

8.11.1 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.)

8.11.2 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 8.11.1.

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 8.11.1.

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 8.11.1.

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 8.11.1.

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.

8.11.3 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 8.11.1.

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 8.11.1.

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]**.

8.11.4 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 8.11.1.

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 8.11.1.

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]**.

8.11.5 Outputting Custom Macro Common Variables

Custom macro common variables can be output using the ALL IO screen.

Outputting custom macro common variables

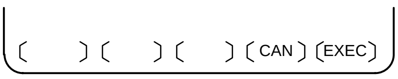
Procedure

1 Press soft key **[MACRO]** on the ALL IO screen, described in Section 8.11.1.

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 ][        ][        ]
```



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.

8.11.6 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 8.11.1.

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 **[F SRH]**.

```
(F SET)(    )(    )(CAN)(EXEC)
```

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 8.11.1.

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 8.11.1.

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 8.11.1.

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.

8.12 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

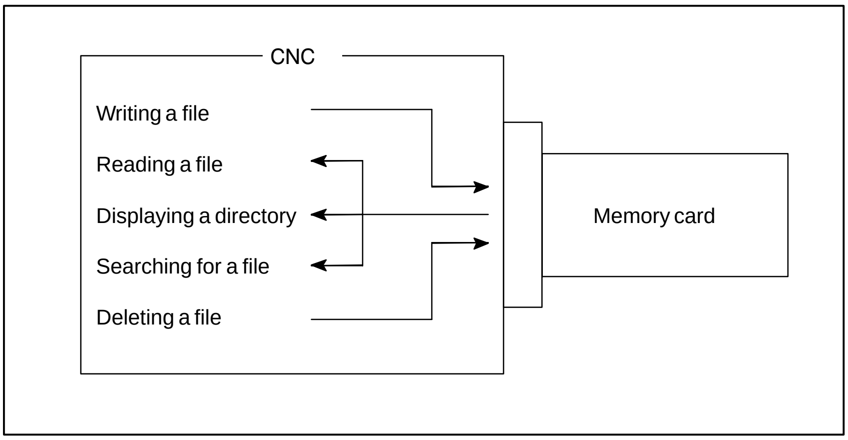
  Text–format files can be read from a memory card.

· 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	01/07/10
0002	O1001	8458	01/07/30
0003	O0002	3250	01/07/30
0004	O2000	73456	01/07/31
0005	O2001	3444	01/07/31
0006	O3001	8483	01/08/02
0007	O3300	406	01/08/05
0008	O3400	2420	01/07/31
0009	O3500	7460	01/07/31

(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         01/07/10
  0002    O1001               8458         01/07/30
  0003    O0002               3250         01/07/30
  0004    O2000              73456         01/07/31
  0005    O2001               3444         01/07/31
  0006    O3001               8483         01/08/02
  0007    O3300                406         01/08/05
  0008    O3400               2420         01/07/31
  0009    O3500               7460         01/07/31

 ( 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         01/07/10
  0002     O1001              8458         01/07/30
  0003     O0002              3250         01/07/30
  0004     O2000             73456         01/07/31
  0005     O2001              3444         01/07/31
  0006     O3001              8483         01/08/02
  0007     O3300               406         01/08/05
  0008     O3400              2420         01/07/31
  0009     O3500              7460         01/07/31
~                                                        ~
 ( 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              01/07/10
  0002   O1001                    8458              01/07/30
  0003   O0002                    3250              01/07/30
  0004   O2000                   73456              01/07/31
  0005   O2001                    3444              01/07/31
  0006   O3001                    8483              01/08/02
  0007   O3300                     406              01/08/05
  0008   O3400                    2420              01/07/31
  0009   O3500                    7460              01/07/31

( 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        01/07/10
  0002   O1001                8458        01/07/30
  0003   O0002                3250        01/07/30
  0004   O2000               73456        01/07/31
  0005   O2001                3444        01/07/31
  0006   O3001                8483        01/08/02
  0007   O3300                 406        01/08/05
  0008   O3400                2420        01/07/31
  0009   O3500                7460        01/07/31

[ 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       )
```

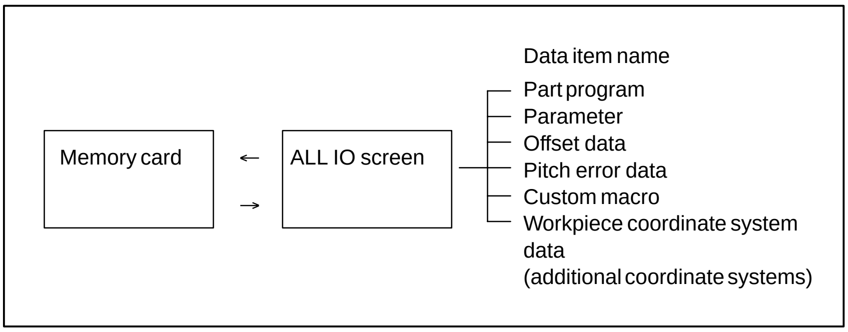
File name O1020 is deleted.

```
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        01–04–06
  *0002   O1003                  334450        01–05–04
  *0003   MACROVAR.DAT           653400        01–05–12
  *0004   O0002                  341205        01–05–13
     [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 Sections NO TAG and Section 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   01/04/06
 0002   O1003                       4450   01/05/04
 0003   MACROVAR.DAT              653400   01/05/12
 0004   O0003                       4610   01/05/04
 0005   O0001                       4254   01/06/04
 0006   O0002                        750   01/06/04
 0007   CNCPARAM.DAT               34453   01/06/04
```

- **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  01–06–14

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.

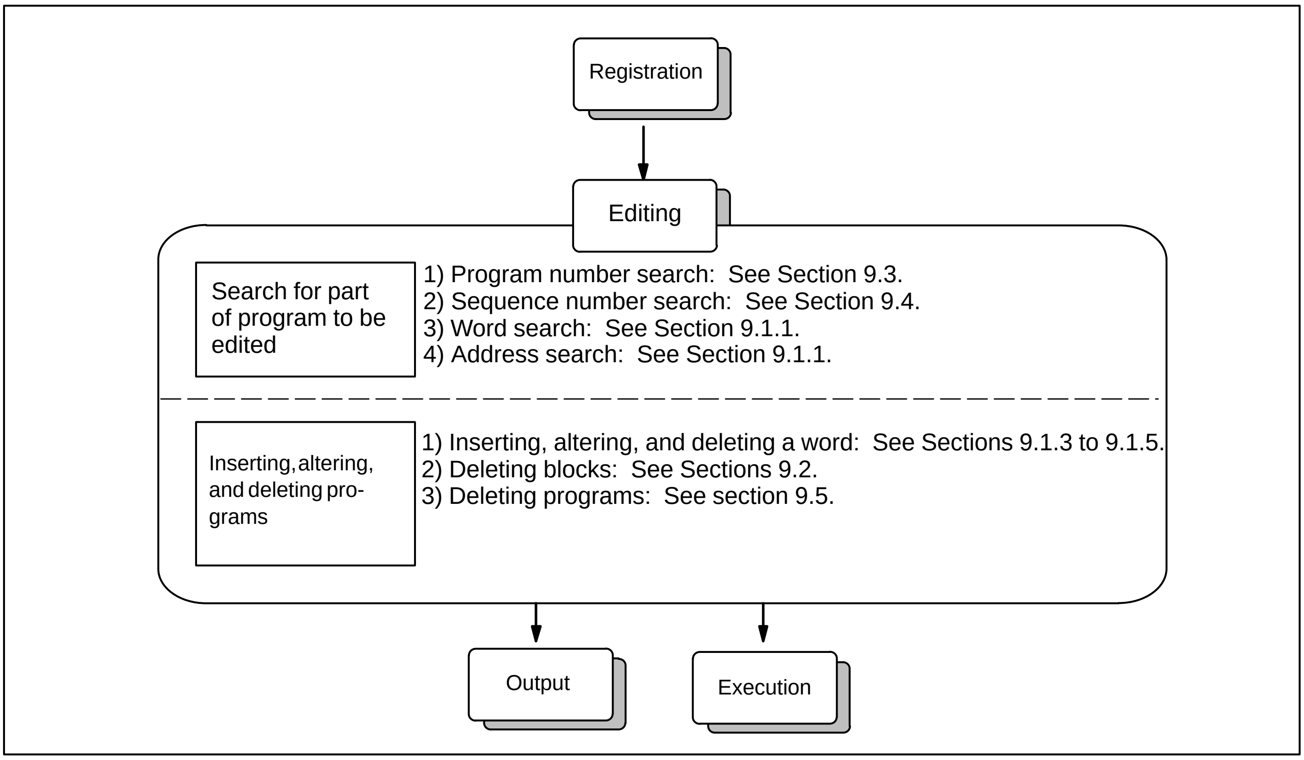
Memory Card Error Codes

Code	Meaning
99	A portion that precedes the FAT area on the memory card is disrupted.
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	There is not enough free space on the memory card.
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.

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

Procedure

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

Procedure

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  Y1250.0   ;
 T12  ;
 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

Procedure

Example) of Searching for T12

```
PROGRAM                          O0050  N01234
 O0050  ;
 N01234   X100.0  Y1250.0  ;  <---- N01234 is being searched for/ scanned currently.
 T12  ;  <------------------------- T12 is searched for.
 N56789  M03  ;
 M02  ;
 %
```

1 Key in address [T] .

2 Key in [1] [2] .

·T12 cannot be searched for if only T1 is keyed in.
·T09 cannot be searched for by keying in only T9.
To search for T09, be sure to key in T09.

3 Pressing the **[SRH↓]** key starts search operation.
Upon completion of search operation, the cursor is displayed at T12.
Pressing the **[SRH↑]** key rather than the **[SRH↓]** key performs search operation in the reverse direction.

Procedure for searching an address

Procedure

Example) of Searching for M03

```
PROGRAM                          O0050  N01234
 O0050  ;
 N01234   X100.0  Y1250.0  ;  <---- N01234 is being searched for/ scanned currently.
 T12  ;
 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

Procedure

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

Procedure

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 Y1250.

```
Program                              O0050  N01234
 O0050  ;
 N01234  X100.0  Y1250.0  ;
 T12  ;
 N56789  M03  ;
 M02  ;
 %
```

Y1250.0 is searched for/ scanned.

2. Key in [T] [1] [5] .
3. Press the [INSERT] key.

```
Program                              O0050  N01234
 O0050  ;
 N01234  X100.0  Y1250.0  T15  ;
 T12  ;
 N56789  M03  ;
 M02  ;
 %
```

T15 is inserted.

9.1.4 Altering a Word

Procedure for altering a word

Procedure

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 Y1250.0 T15 ;
 T12 ;
 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 Y1250.0 M15 ;
 T12 ;
 N5678 M03 ;
 M02 ;
 %
```

T15 is changed to M15.

9.1.5 Deleting a Word

Procedure for deleting a word

Procedure

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   Y1250.0  M15  ;
 T12  ;
 N56789  M03  ;
 M02  ;
 %
```

X100.0 is searched for/ scanned.

2 Press the DELETE key.

```
Program                              O0050  N01234
 O0050  ;
 N01234  Y1250.0  M15  ;
 T12  ;
 N56789  M03  ;
 M02  ;
 %
```

X100.0 is deleted.

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

Procedure

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   Y1250.0  M15  ;   <----------- N01234 is searched for/ scanned.
 T12  ;
 N56789  M03  ;
 M02  ;
 %
```

2 Key in [EOB].

3 Press the [DELETE] key.

```
Program                                  O0050  N01234
 O0050  ;   <--------------------------- Block containing N01234 has been deleted.
 T12  ;
 N56789  M03  ;
 M02  ;
 %
```

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

Procedure

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  Y1250.0  M15 ;  <----
 T12 ;
 N56789  M03 ;
 M02 ;
 %
```

N01234 is searched for/ scanned.

2 Key in N 5 6 7 8 9 .

```
Program                                   O0050  N01234
 O0050 ;
 N01234  Y1250.0  M15 ; }
 T12 ;                  } <----
 N56789  M03 ;          }
 M02 ;
 %
```

Underlined part is deleted.

3 Press the DELETE key.

```
Program                                   O0050  N01234
 O0050 ;
 M02 ;  <----
 %
```

Blocks from block containing N01234 to block containing N56789 have been deleted.

9.3 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

Procedure

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 tool side to start automatic operation. Refer to the relevant manual prepared by the machine tool builder for detailed information on operation.

1 Select **MEMORY** mode.

2 Set the reset state(*1)

· The reset state is the state where the LED for indicating that automatic operation is in progress is off. (Refer to the relevant manual of the machine tool builder.)

3 Set the program number selection signal on the machine tool side to a number from 01 to 15.

· If the program corresponding to a signal on the machine tool side is not registered, P/S alarm (No. 59) is raised.

4 Press the cycle start button.

· When the signal on the machine tool side represents 00, program number search operation is not performed.

Alarm

No.	Contents
59	The program with the selected number cannot be searched during external program number search.
71	The specified program number was not found during program number search.

9.4 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 Y100.0 ;
                          T12 ;
                            :
Selected program    ──►   O0002 ;                  ⎫ This section is
                          N02345 X20.0 Y20.0 ;     ⎬ searched starting at
Target sequence     ──►   N02346 X10.0 Y10.0 ;     ⎭ the beginning.
number is found.            :                        (Search operation is
                          O0003 ;                    performed only within a
                            :                        program.)
```

Procedure for sequence number search

Procedure

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 60 occurs.

Explanations

- **Operation during Search**

Those blocks that are skipped do not affect the CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values.
So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify M, S, and T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and NC states at that point.

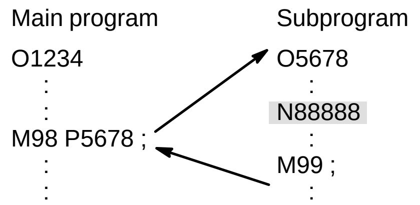
- **Checking during search**

During search operation, the following checks are made:
·Optional block skip
·P/S alarm (No. 003 to 010)

Limitations

- **Searching in sub–program**

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm (No.060) is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.



If an attempt is made to search for N8888 in the example above, an alarm is raised.

Alarm

Number	Contents
60	Command sequence number was not found in the sequence number search.

9.5 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

Procedure

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

Procedure

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

Procedure

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 EDITING OF CUSTOM MACROS

Unlike ordinary programs, custom macro programs are modified, inserted, or deleted based on editing units.
Custom macro words can be entered in abbreviated form.
Comments can be entered in a program.
Refer to the section 10.1 for the comments of a program.

Explanations

- **Editing unit**

When editing a custom macro already entered, the user can move the cursor to each editing unit that starts with any of the following characters and symbols:

(a) Address
(b) # located at the start of the left side of a substitution statement
(c) /, (,=, and ;
(d) First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT and PCLOS

On the CRT screen, a blank is placed before each of the above characters and symbols.
(Example) Head positions where the cursor is placed
N001 X–#100 ;
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X–SQRT[#3/3*[#4+1]] ;
N004 X–#2 Z#1 ;
N005 #5 =1+2–#10 ;
IF[#1NE0] GOTO10 ;
WHILE[#2LE5] DO1 ;
#[200+#2] =#2*10 ;
#2 =#2+1 ;
END1 ;

- **Abbreviations of custom macro word**

When a custom macro word is altered or inserted, the first two characters or more can replace the entire word.
Namely,

WHILE → WH	**GOTO → GO**	**XOR → XO**	**AND → AN**
SIN → SI	**COS → CO**	**TAN → TA**	**ATAN → AT**
SQRT → SQ	**ABS → AB**	**BCD → BC**	**BIN → BI**
FIX → FI	**FUP → FU**	**ROUND → RO**	**END → EN**
POPEN → PO	**BPRNT → BP**	**DPRNT → DP**	**PCLOS→PC**

(Example) Keying in
WH [AB [#2] LE RO [#3]]
has the same effect as
WHILE [ABS [#2] LE ROUND [#3]]
The program is also displayed in this way.

9.7 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

Procedure

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.

9.8 PASSWORD FUNCTION

The password function (bit 4 (NE9) of parameter No. 3202) can be locked using parameter No. 3210 (PASSWD) and parameter No. 3211 (KEYWD) to protect program Nos. 9000 to 9999. In the locked state, parameter NE9 cannot be set to 0. In this state, program Nos. 9000 to 9999 cannot be modified unless the correct keyword is set.
A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

Procedure for locking and unlocking

Locking

1 Set the MDI mode.

2 Enable parameter writing. At this time, P/S alarm No. 100 is issued on the CNC.

3 Set parameter No. 3210 (PASSWD). At this time, the locked state is set.

4 Disable parameter writing.

5 Press the RESET key to release the alarm state.

Unlocking

1 Set the MDI mode.

2 Enable parameter writing. At this time, P/S alarm No. 100 is issued on the CNC.

3 In parameter No. 3211 (KEYWD), set the same value as set in parameter No. 3210 (PASSWD) for locking. At this time, the locked state is released.

4 Set bit 4 (NE9) of parameter No. 3202 to 0.

5 Disable parameter writing.

6 Press the RESET key to release the alarm state.

7 Subprograms from program Nos. 9000 to 9999 can now be edited.

Explanations

- **Setting parameter PASSWD**

The locked state is set when a value is set in the parameter PASSWD. However, note that parameter PASSWD can be set only when the locked state is not set (when PASSWD = 0, or PASSWD = KEYWD). If an attempt is made to set parameter PASSWD in other cases, a warning is given to indicate that writing is disabled. When the locked state is set (when PASSWD = 0 and PASSWD = KEYWD), parameter NE9 is automatically set to 1. If an attempt is made to set NE9 to 0, a warning is given to indicate that writing is disabled.

- **Changing parameter PASSWD**

Parameter PASSWD can be changed when the locked state is released (when PASSWD = 0, or PASSWD = KEYWD). After step 3 in the procedure for unlocking, a new value can be set in the parameter PASSWD. From that time on, this new value must be set in parameter KEYWD to release the locked state.

- **Setting 0 in parameter PASSWD**

When 0 is set in the parameter PASSWD, the number 0 is displayed, and the password function is disabled. In other words, the password function can be disabled by either not setting parameter PASSWD at all, or by setting 0 in parameter PASSWD after step 3 of the procedure for unlocking. To ensure that the locked state is not entered, care must be taken not to set a value other than 0 in parameter PASSWD.

- **Re–locking**

After the locked state has been released, it can be set again by setting a different value in parameter PASSWD, or by turning the power to the NC off then on again to reset parameter KEYWD.

WARNING

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.

9.9 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.

9.9.1 Copying an Entire Program

A new program can be created by copying a program.

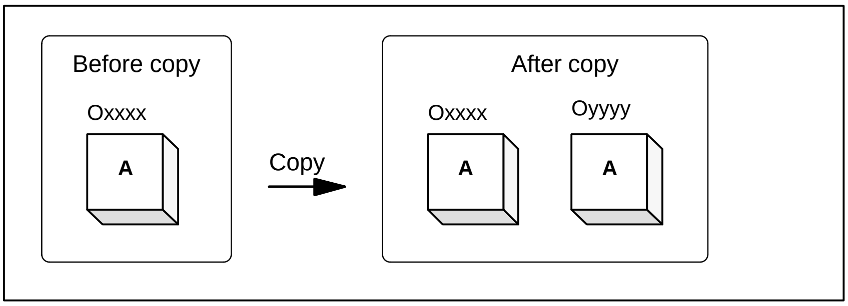


Fig. 9.9.1 Copying an entire program

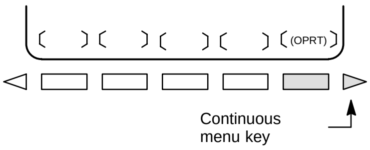
In Fig. 9.9.1, the program with program number xxxx is copied to a newly created program with program number yyyy. The program created by copy operation is the same as the original program except the program number.

Procedure of copying an entire program

Procedure

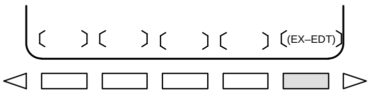
1 Enter the **EDIT** mode.

2 Press function key PROG .

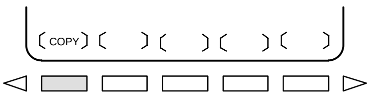


3 Press soft key **[(OPRT)]**.

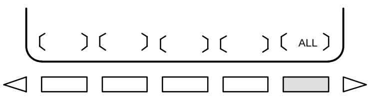
4 Press the continuous menu key.



5 Press soft key **[EX–EDT]**.



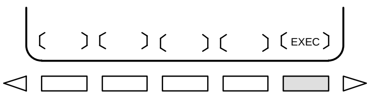
6 Check that the screen for the program to be copied is selected and press soft key **[COPY]**.



7 Press soft key **[ALL]**.

Numeric keys 0 ~ 9

8 Enter the number of the new program (with only numeric keys) and press the INPUT key.



9 Press soft key **[EXEC]**.

9.9.2 Copying Part of a Program

A new program can be created by copying part of a program.

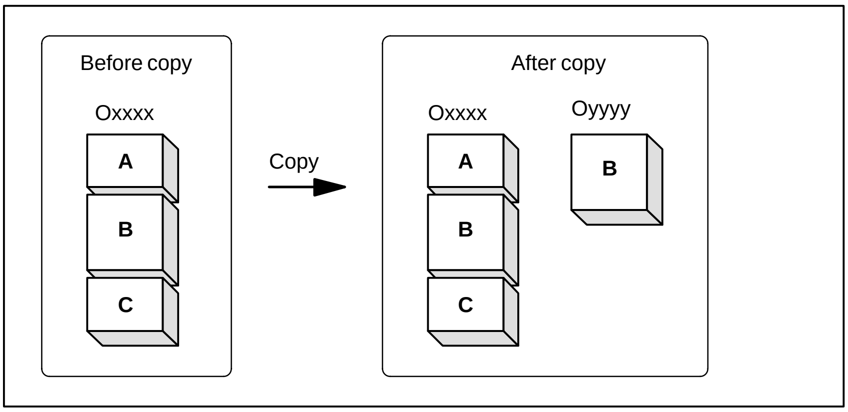


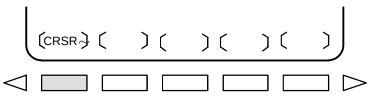
Fig. 9.9.2 Copying part of a program

In Fig. 9.9.2, 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.

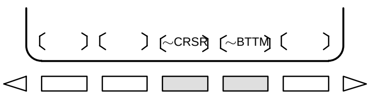
Procedure for copying part of a program

Procedure

1 Perform steps 1 to 6 in subsection 9.9.1.



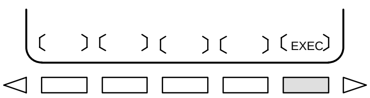
2 Move the cursor to the start of the range to be copied and press soft key **[CRSR∼]**.



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]**.

9.9.3 Moving Part of a Program

A new program can be created by moving part of a program.

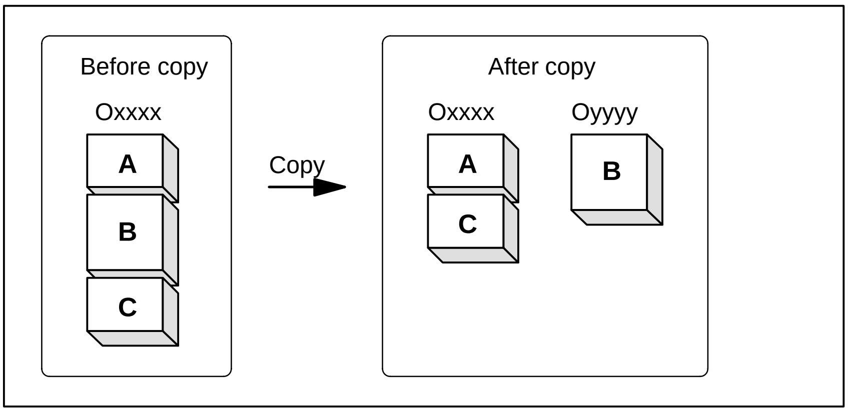


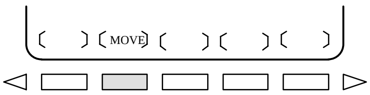
Fig. 9.9.3 Moving part of a program

In Fig. 9.9.3, 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.

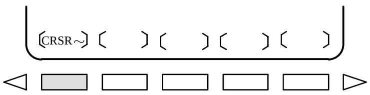
Procedure for moving part of a program

Procedure

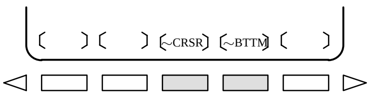
1 Perform steps 1 to 5 in subsection 9.9.1.



2 Check that the screen for the program to be moved is selected and press soft key **[MOVE]**.



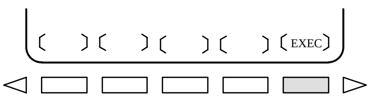
3 Move the cursor to the start of the range to be moved and press soft key **[CRSR∼]**.



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]**.

9.9.4 Merging a Program

Another program can be inserted at an arbitrary position in the current program.

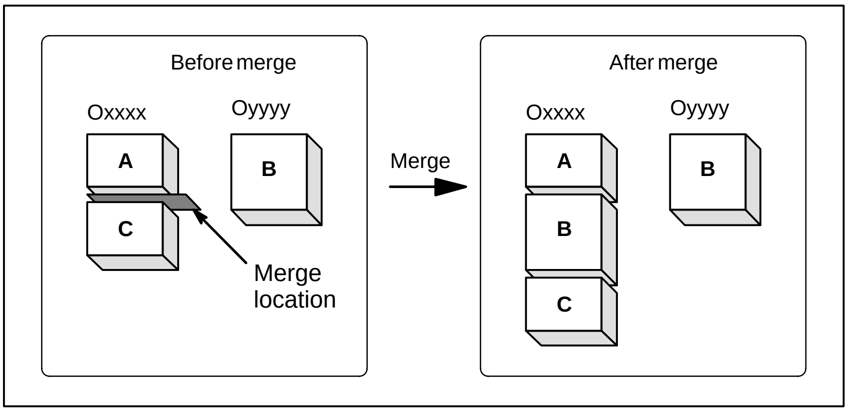
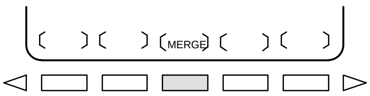


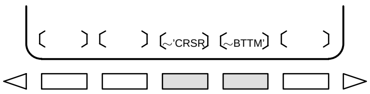
Fig. 9.9.4 Merging a program at a specified location

In Fig. 9.9.4, the program with program number XXXX is merged with the program with program number YYYY. The OYYYY program remains unchanged after merge operation.

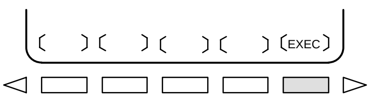
Procedure for merging a program

Procedure





Numeric keys 0 ~ 9



1 Perform steps 1 to 5 in subsection 9.9.1.

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.

9.9.5 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/S140 alarm 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.

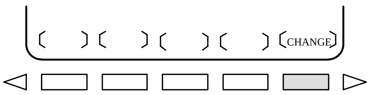
9.9.6 Replacement of Words and Addresses

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 hange of words or addresses

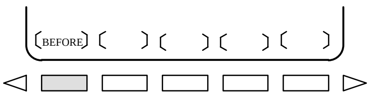
Procedure

1 Perform steps 1 to 5 in subsection 9.9.1.



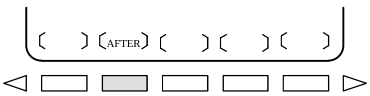
2 Press soft key **[CHANGE]**.

3 Enter the word or address to be replaced.

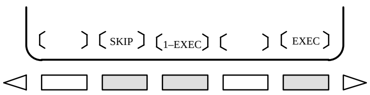


4 Press soft key **[BEFORE]**.

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 **[BEFOR]** , 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.)

10 CREATING PROGRAMS

Programs can be created using any of the following methods:

- MDI keyboard
- AUTOMATIC PROGRAM PREPARATION DEVICE (FANUC SYSTEM P)

This chapter describes creating programs using the MDI panel. 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 Chapter 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 Chapter 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 9.6).
- 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 subjection 11.4.3).

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 ][  LIB  ][        ][  C.A.P  ][ (OPRT) ]
```

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 CONVERSATIONAL PROGRAMMING WITH GRAPHIC FUNCTION

Programs can be created block after block on the conversational screen while displaying the G code menu.
Blocks in a program can be modified, inserted, or deleted using the G code menu and conversational screen.

Procedure for Conversational Programming with Graphic Function

Procedure 1
Creating a program

1 Enter the **EDIT** mode.

2 Press PROG . If no program is registered, the following screen is displayed. If a program is registered, the program currently selected is displayed.

```
PROGRAM                                          O0000  N0000









>_
EDIT  ****  ***  ***                 11 : 59 : 46
[ PRGRM ][  LIB  ][        ][ (C.A.P) ][ (OPRT) ]
```

3 Key in the program number of a program to be registered after keying in address O, then press INSERT . For example, when a program with program number 10 is to be registered, key in O 1 0 , then press INSERT . This registers a new program O0010.

4 Press the **[C.A.P]** soft key. The following G code menu is displayed on the screen.
If soft keys different from those shown in step 2 are displayed, press the menu return key to display the correct soft keys.

```
PROGRAM                              O0010 N00000
G00 : POSITIONING
G01 : LINEAR IPL.
G02 : CIRCULAR IPL.  CW
G03 : CIRCULAR IPL.  CCW
G04 : DWELL
G09 : EXACT STOP CHECK
G10 : OFFSET VALUE SETTING                    (O)
G20 : INCH                                    (O)
G21 : METRIC
G22 : STORED STROKE CHECK ON                  (O)
G23 : STORED STROKE CHECK OFF
G26 : BOLT HOLE CIRCLE
)_
EDIT **** *** ***        11:54:42
[ PRGRM ] [       ] [ G.MENU ] [ BLOCK ] [        ]
```

5 Key in the G code corresponding to a function to be programmed. When the positioning function is desired, for example, the G code menu lists the function with the G code G00. So key in G00. If the screen does not indicate a function to be programmed, press the page key to display the next G code menu screen. Repeat this operation until a desired function appears. If a desired function is not a G code, key in no data.

6 Press the soft key **[BLOCK]** to display a detailed screen for a keyed in G code. The figure below shows an example of detailed screen for G00.

```
PROGRAM                              O0010 N00200
G00 : POSITIONING

 G00  G    G    G
 X     100.   Y     50.
 C
 T                                  (x,y,c)
 M
 S
 D
 ;

EDIT **** *** ***        11:56:22
[ PRGRM ] [       ] [ G.MENU ] [ BLOCK ] [ (OPRT) ]
```

When no keys are pressed, the standard details screen is displayed.

```
PROGRAM                                 O0010 N00000
STANDARD FORMAT

 G    G    G    G
 X            Y
 C
 I            J
 K
 P            Q
 R            F
 M            S
 T            D
 L            H
 ;
EDIT **** *** ***          11:55:13
[ PRGRM ][      ][ G.MENU ][ BLOCK ][ (OPRT) ]
```

7 Move the cursor to the block to be modified on the program screen. At this time, a data address with the cursor blinks.

8 Enter numeric data by pressing the numeric keys and press the **[INPUT]** soft key or INPUT key. This completes the input of one data item.

9 Repeat this operation until all data required for the entered G code is entered.

10 Press the INSERT key. This completes the registration of data of one block in program memory. On the screen, the G code menu screen is displayed, allowing the user to enter data for another block. Repeat the procedure starting with **5** as required.

11 After registering all programs, press the **[PRGRM]** soft key. The registered programs are converted to the conversational format and displayed.

12 Press the RESET key to return to the program head.

Procedure2
Modifying a block

1 Move the cursor to the block to be modified on the program screen and press the **[C.A.P]** soft key. Or, press the **[C.A.P]** soft key first to display the conversational screen, then press the ⬇ or ⬆ page key until the block to be modified is displayed.

2 When data other than a G code is to be altered, just move the cursor to the data and key in a desired value, then press the **[INPUT]** soft key or INPUT key.

3 When a G code is to be altered, press the menu return key ◁ and the soft key **[G.MENU]**. Then the G code menu appears. Select a desired G code, then key in the value. For example, to specify a cutting feed, since the G code menu indicates G01, key in G01. Then press the soft key **[BLOCK]**. The detailed screen of the G code is displayed, so enter the data.

4 After data is changed completely, press the ALTER key. This operation replaces an entire block of a program.

Procedure3
Inserting a block

1 On the conversational screen, display the block immediately before a new block is to be inserted, by using the page keys. On the program screen, move the cursor with the page keys and cursor keys to immediately before the point where a new block is to be inserted.

2 Press the soft key **[G.MENU]** to display the G code menu. Then enter new block data.

3 When input of one block of data is completed in step **2**, press the INSERT key. This operation inserts a block of data.

Procedure4
Deleting a block

1 On the conversational screen, display the contents of a block to be deleted, then press the DELETE key.

2 The contents of the block displayed are deleted from program memory. Then the contents of the next block are displayed on the conversational screen.

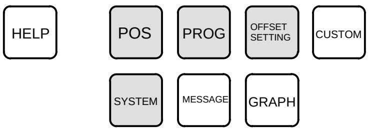
11 SETTING AND DISPLAYING DATA

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



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, tool compensation values, setting data, and custom macro variables. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.

POSITION DISPLAY SCREEN

Screen transition triggered by the function key POS

POS

Current position screen

[ABS] [REL] [ALL] [HNDL] [(OPRT)] [▷]

ABS

- Position display of work coordinate system ⇒See subsec. 11.1.1.
- Display of part count and run time ⇒See subsec. 11.1.6.
- Display of actual speed ⇒See subsec. 11.1.5.

REL

- Position displays relative coordinate system ⇒See subsec. 11.1.2.
- Display of part count and run time ⇒See subsec. 11.1.6.
- Display of actual speed ⇒See subsec. 11.1.5.
- Setting of relative coordinate values ⇒See subsec. 11.1.2.

ALL

- Total position display of each coordinate system ⇒See subsec.11.1.3.
- Display of part count and run time ⇒See subsec. 11.1.6.
- Display of actual speed ⇒See subsec. 11.1.5.
- Setting of relative coordinate values ⇒See subsec. 11.1.2.

HNDL

- Manual handle interruption ⇒See sec. 4.6.

Current position screen

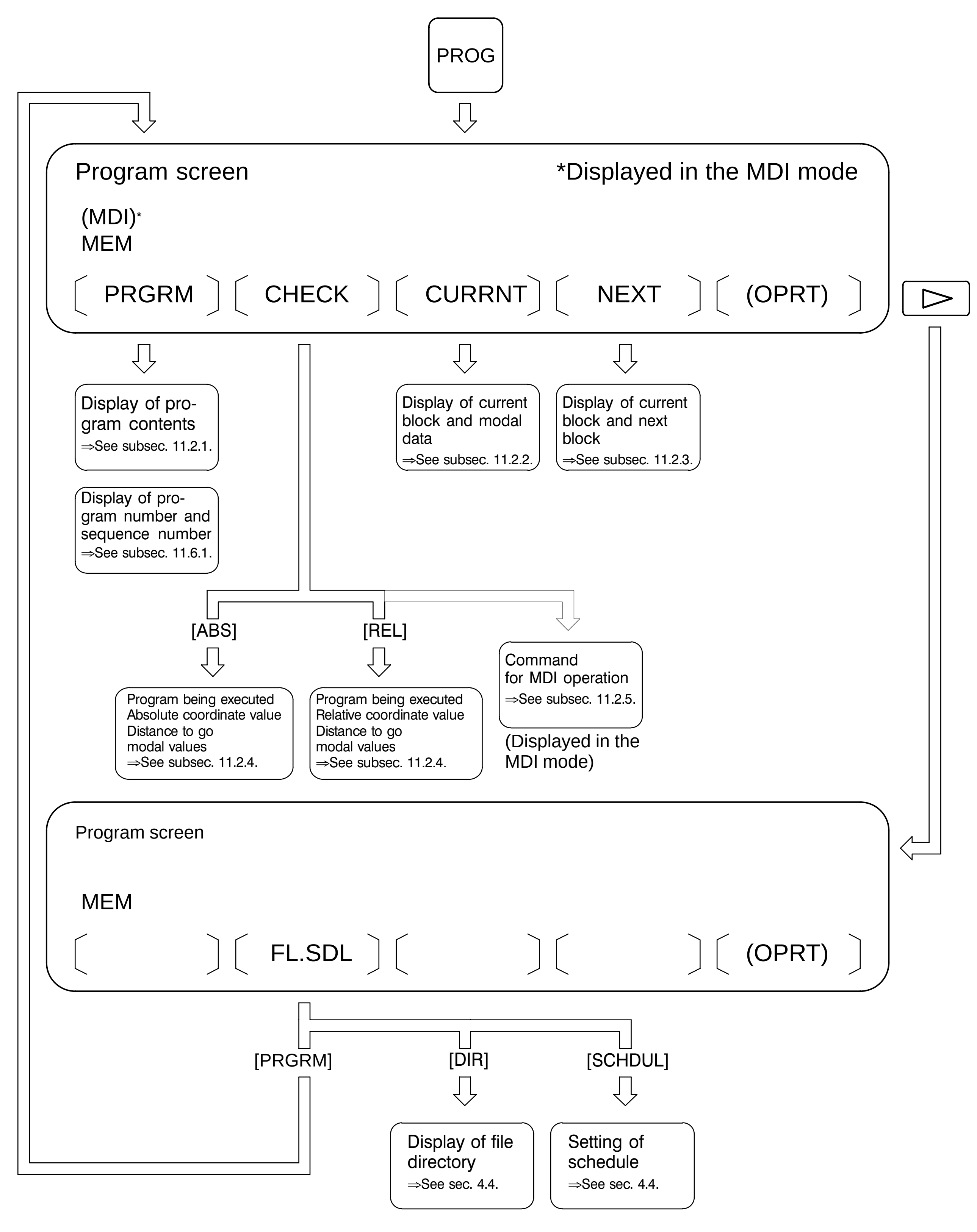
[MONI] [ ] [ ] [ ] [(OPRT)]

MONI

- Display of operating monitor ⇒See subsec. 11.1.7.

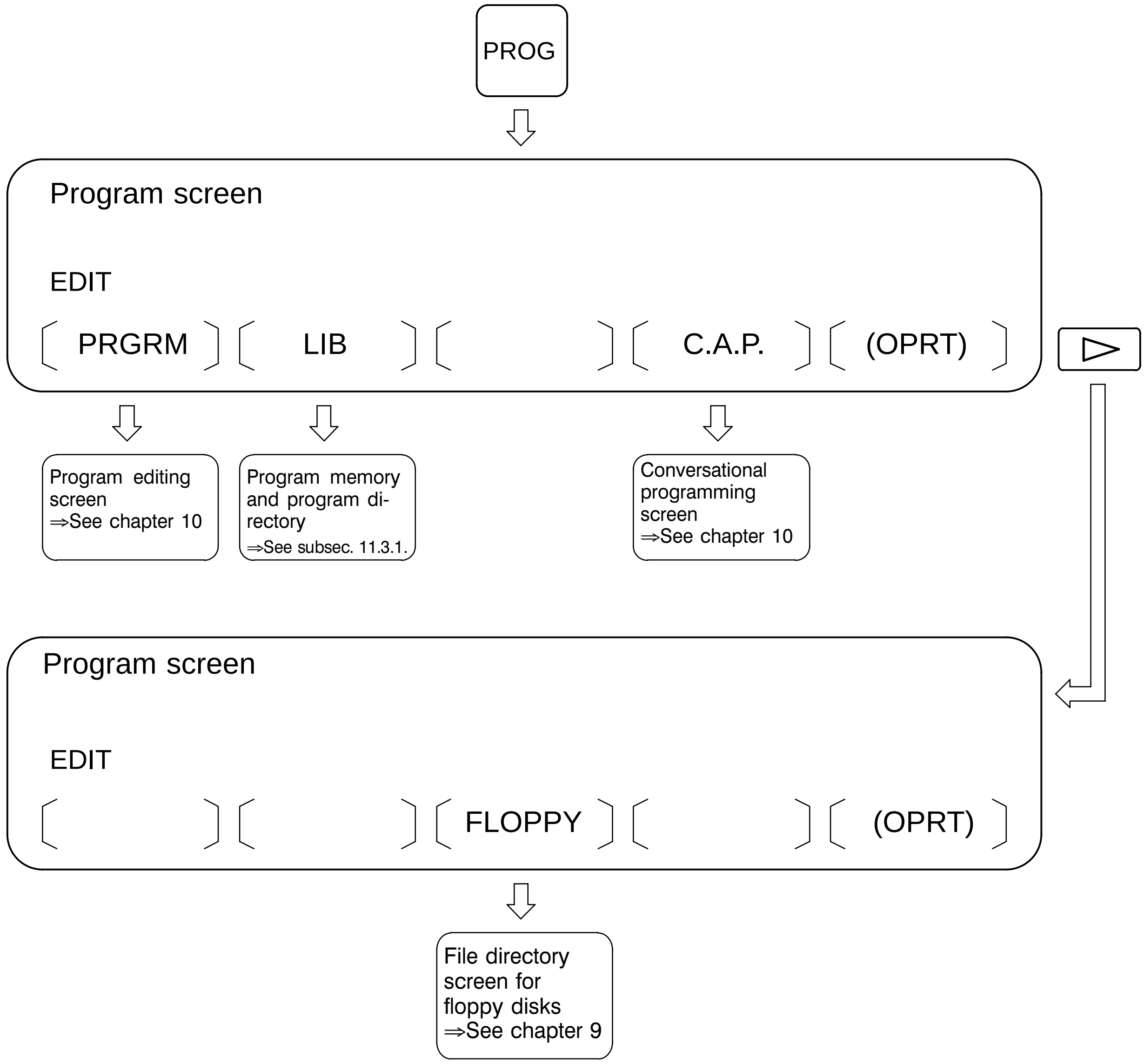
PROGRAM SCREEN

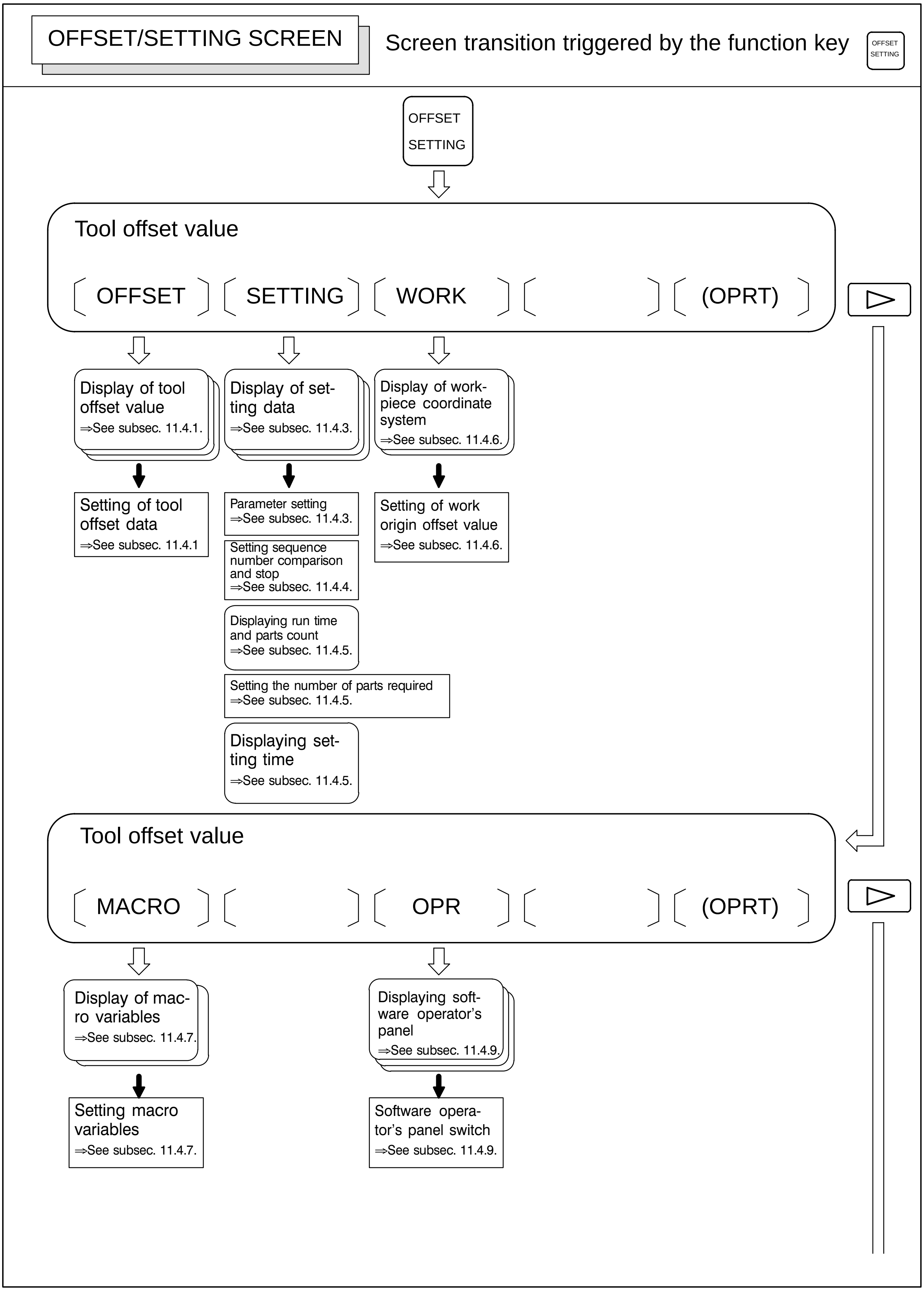
Screen transition triggered by the function key PROG in the MEMORY or MDI mode

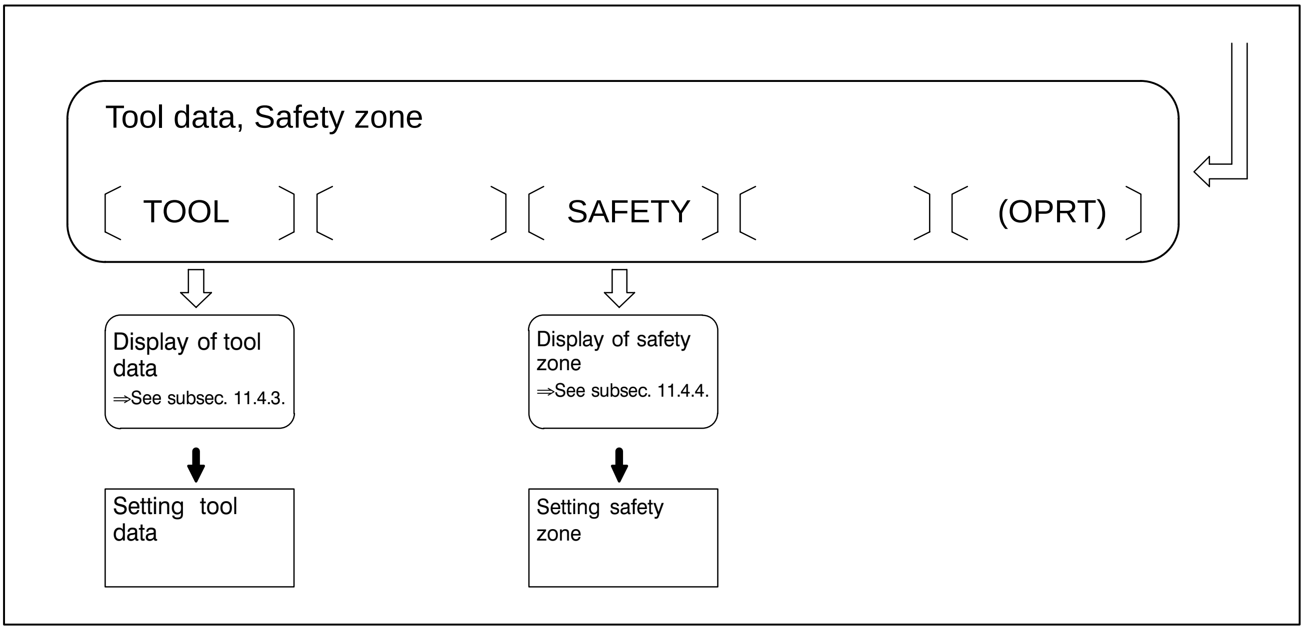


PROGRAM SCREEN

Screen transition triggered by the function key PROG in the EDIT mode

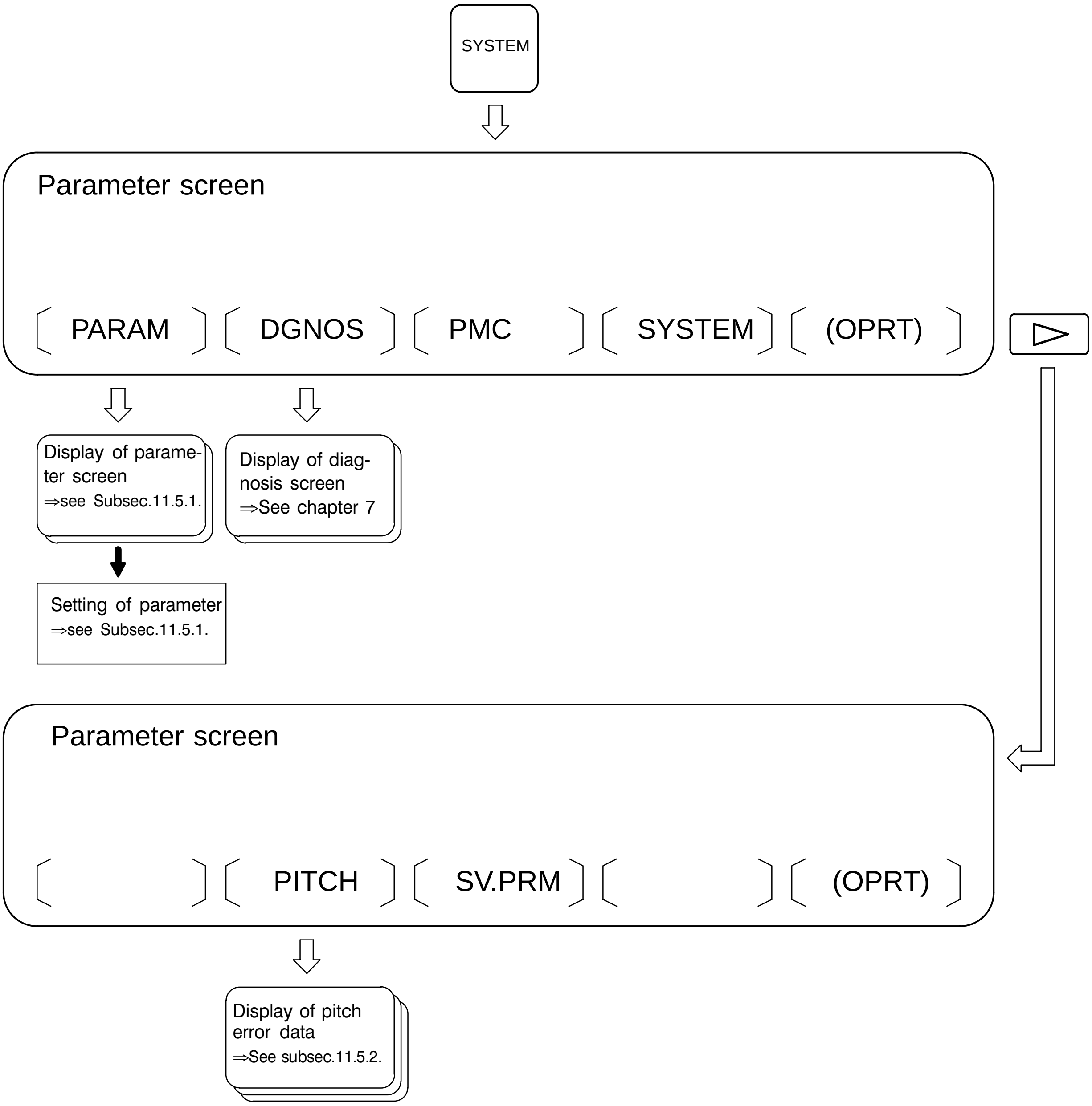



OFFSET/SETTING SCREEN
Screen transition triggered by the function key
OFFSET SETTING
OFFSET SETTING
Tool offset value
OFFSET
SETTING
WORK
(OPRT)
Display of tool offset value
⇒See subsec. 11.4.1.
Display of setting data
⇒See subsec. 11.4.3.
Display of workpiece coordinate system
⇒See subsec. 11.4.6.
Setting of tool offset data
⇒See subsec. 11.4.1
Parameter setting
⇒See subsec. 11.4.3.
Setting sequence number comparison and stop
⇒See subsec. 11.4.4.
Displaying run time and parts count
⇒See subsec. 11.4.5.
Setting the number of parts required
⇒See subsec. 11.4.5.
Displaying setting time
⇒See subsec. 11.4.5.
Setting of work origin offset value
⇒See subsec. 11.4.6.
Tool offset value
MACRO
OPR
(OPRT)
Display of macro variables
⇒See subsec. 11.4.7.
Displaying software operator's panel
⇒See subsec. 11.4.9.
Setting macro variables
⇒See subsec. 11.4.7.
Software operator's panel switch
⇒See subsec. 11.4.9.


Tool data, Safety zone
TOOL
SAFETY
(OPRT)
Display of tool data
⇒See subsec. 11.4.3.
Display of safety zone
⇒See subsec. 11.4.4.
Setting tool data
Setting safety zone

SYSTEM SCREEN

Screen transition triggered by the function key SYSTEM



• Setting screens

The table below lists the data set on each screen.

Table11 Setting screens and data on them

No.	Setting screen	Contents of setting	Reference item
1	Tool offset value	Cutter compensation value	Subsec. 11.4.1
2	Setting data(handy)	Parameter write TV check Punch code Input unit (mm/inch) I/O channel Automatic insert of Sequence No.	Subsec. 11.4.2
		Sequence number comparison and stop	Subsec. 11.4.5
3	Setting data (mirror image)	Mirror image	Subsec. 11.4.2
4	Setting data (timer)	Parts required	Subsec. 11.4.6
5	Macro variables	Custom macro common variables (#100 to #199) (#500 to #999)	Subsec. 11.4.9
6	Parameter	Parameter	Subsec. 11.5.1
7	Pitch error	Pitch error compensation data	Subsec. 11.5.2
8	software operator's panel	Mode selection Jog feed axis selection Jog rapid traverse Axis selection for Manual pulse generator Multiplication for manual pulse generator Jog feedrate Feedrate override Rapid traverse override Optional block skip Single block Machine lock Dry run Protect key Feed hold	Subsec. 11.4.10
9	Tool data	Number of using tools Number of indexing tools Zero position tool Turret per rotation Total punch count Tool number Turret position X, Y axis offset Tool change Punch count Tool shape for graphic Tool life data	Subsec. 11.4.3
10	Sefety zone data	Safety zone area Tool zone	Subsec. 11.4.4
11	Work coordinate system setting	Work origin offset value	Subsec. 11.4.7

11.1 SCREENS DISPLAYED BY FUNCTION KEY POS

Press function key POS to display the current position of the tool.

The following three screens are used to display the current position of the tool:

- Position display screen for the work coordinate system.
- Position display screen for the relative coordinate system.
- Overall position display screen.

The above screens can also display the feedrate, run time, and the number of parts.

Function key POS can also be used to display the load on the servo motor.

Function key POS can also be used to display the screen for displaying the distance moved by handle interruption. See Section 4.6 for details on this screen.

11.1.1 Position Display in the Work Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate system

Procedure

1 Press function key POS .

2 Press soft key **[ABS]**.

```
ACTUAL POSITION(ABSOLUTE)        O1000 N00010

   X          123.456
   Y          363.233
   C            0.000

                        PART COUNT        5
 RUN TIME     0H15M     CYCLE TIME    0H 0M38S
 ACT.F     3000 MM/M            S    0 T0000

MEM STRT MTN ***        09:06:35
[ ABS ] [ REL ] [  ALL ] [ HNDL ] [ OPRT ]
```

Explanations

- **Display including compensation values**

Bits 6 and 7 of parameter 3104 can be used to select whether the displayed values include cutter compensation.

11.1.2 Position Display in the Relative Coordinate System

Displays the current position of the tool in a relative coordinate system based on the coordinates set by the operator. The current position changes as the tool moves. The increment system is used as the unit for numeric values. The title at the top of the screen indicates that relative coordinates are used.

Display procedure for the current position screen with the relative coordinate system

Procedure

1 Press function key POS .

2 Press soft key **[REL]**.

```
ACTUAL POSITION(RELATIVE)          O1000 N00010
     X          123.456
     Y          363.233
     C            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.

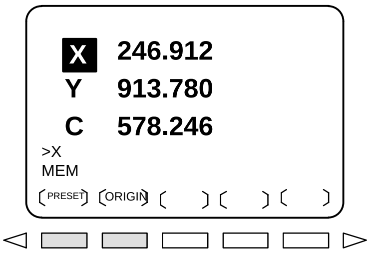
Explanations

- **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



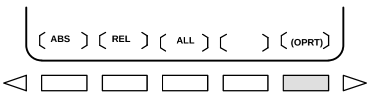
1 Enter an axis address (such as X or Y) on the screen for the relative coordinates. The indication for the specified axis blinks and the soft keys change as shown on the left.

2 · To reset the coordinate to 0, press soft key **[ORGIN]**. The relative coordinate for the blinking axis is reset to 0.

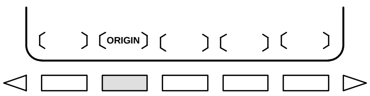
· To preset the coordinate to a specified value, enter the value and press soft key **[PRESET]**. The relative coordinate for the blinking axis is set to the entered value.

Procedure to reset all axes

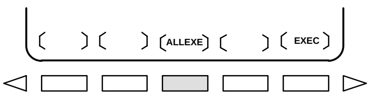
Procedure



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 6 and 7 of parameter 3104 can be used to select whether the displayed values include cutter compensation.

- **Presetting by setting a coordinate system**

Bit 3 of parameter 3104 is used to specify whether the displayed positions in the relative coordinate system are preset to the same values as in the workpiece coordinate system when a coordinate system is set by a G92 command or when the manual reference position return is made.

11.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance. The relative coordinates can also be set on this screen. See subsection 11.1.2 for the procedure.

Procedure for displaying overall position display screen

Procedure

1 Press function key POS .

2 Press soft key **[ALL]**.

```
ACTUAL POSITION                          O1000 N00010
   (RELATIVE)                  (ABSOLUTE)
   X   246.912                 X   123.456
   Y   913.780                 Y   456.890
   C  1578.246                 C   789.123
   (MACHINE)                   (DISTANCE TO GO)
   X     0.000                 X     0.000
   Y     0.000                 Y     0.000
   C     0.000                 C     0.000
                             PART COUNT       5
RUN TIME    0H15M            CYCLE TIME   0H 0M38S
ACT.F     3000 MM/M                 S    0 T0000

MEM **** *** ***             09:06:35
[  ABS  ][  REL  ][  ALL  ][ HNDL ][ OPRT ]
```

Explanations

• Coordinate display

The current positions of the tool in the following coordinate systems are displayed at the same time:

- Current position in the relative coordinate system (relative coordinate)
- Current position in the workpiece coordinate system (absolute coordinate)
- Current position in the machine coordinate system (machine coordinate)
- Distance to go (distance to go)

• Distance to go

The distance remaining is displayed in the MEMORY or MDI mode. The distance the tool is yet to be moved in the current block is displayed.

• Machine coordinate system

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter 3104.

• 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

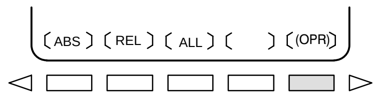
11.1.4 Presetting the Workpiece Coordinate System

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.

Procedure for Presetting the Workpiece Coordinate System

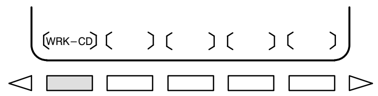
Procedure

1 Press function key POS .

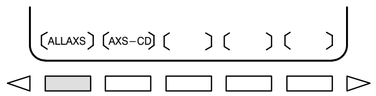


2 Press soft key **[(OPRT)]**.

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 3015.

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 POSITION(ABSOLUTE)          O1000 N00010
   X          123.456
   Y          363.233
   C            0.000

                        PART COUNT       5
RUN TIME     0H15M   CYCLE TIME    0H 0M38S
ACT.F     3000 MM/M        S    0 T0000

MEM STRT MTN ***      09:06:35
[  ABS  ][  REL  ][  ALL  ][  HNDL ][ OPRT ]
```

Actual feedrate is displayed after ACT.F.

Explanations

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

- **Actual feedrate value**

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n}(fi)^2}$$

where

n : Number of axes

fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate

Fact : Actual feedrate displayed

The display unit: mm/min (metric input).
inch/min (Inch input, Two digits below the decimal point are displayed.)

The feedrate along the PMC axis can be omitted by setting bit 1 (PCF) of parameter 3015.

- **Actual feedrate display 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
     C                0.000

                         PART COUNT        5
 RUN TIME     0H15M      CYCLE TIME    0H 0M38S
 ACT.F     3000 MM/M            S    0 T0000

MEM STRT MTN ***         09:06:35
[  ABS   ][  REL  ][  ALL   ][  HNDL ][ OPRT ]
```

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanations

- **PART COUNT**

Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter 6710 is executed.

- **RUN TIME**

Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- **CYCLE TIME**

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- **Display on the other screen**

Details of the run time and the number of machined parts are displayed on the setting screen. See subsection 11.4.5.

- **Parameter setting**

The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters 6711, 6751, and 6752 or on the setting screen.

- **Incrementing the number of machined parts**

Bit 0 (PCM) of parameter 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter 6710 is executed, or only each time an M code specified by parameter 6710 is executed.

11.1.7 Operating Monitor Display

The reading on the load meter can be displayed for each servo axis by setting bit 5 (OPM) of parameter 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 :                  201%

 Y :  * * * * *  0%    (SPEED METER RPM)

 C :  * * * * *  0%  S1 :       * * *    1500

                       PART COUNT       5
  RUN TIME    0H15M    CYCLE TIME    0H 0M38S
  ACT.F     3000 MM/M

MEM STRT MTN ***        09:06:35
[  ABS  ][  REL  ][  ALL  ][  HNDL ][ OPRT ]
```

Explanations

- **Display of the servo axes**

The reading on the load meter can be displayed for up to three servo axes by setting parameters 3151 to 3153.

- **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 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

Function key PROG can also be pressed in MEMORY mode to display the program restart screen and scheduling screen.
See Section 4.4 for the scheduling screen.

11.2.1 Program Contents Display Screen

Displays the program currently being executed in MEMORY or MDI mode.

Procedure for displaying the program contents

Procedure

1 Press function key PROG to display a current position display screen.

2 Press chapter selection soft key **[PRGRM]**.
The cursor is positioned at the block currently being executed.

```
PROGRAM                                 O1000 N00210
N130 X1085. Y635. ;
N140 X15. Y635. ;
N200 G72 X150. Y80. ;
N210 G76 I40.J0 K4 T03 ;
N300 G72 X400. Y80. ;
N310 G76 I40. J0 K7 ;
N400 G72 X770. Y80. ;
N410 G76 I40. J0. K4 ;
N500 G72 X50. Y200. ;
N510 G76 I40. J90. K8 T02 ;
N600 G72 X1050. Y200. ;


 MEM STRT *** FIN        15:08:47
 [ PRGRM ][ CHECK ][CURRNT][NEXT][ (OPRT) ]
```

Explanations

- **12–soft key type LCD**

On the 12–soft key type LCD, 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).

```
                                          O0006  N00000
PROGRAM

O0003 ;                              G65 H01 P#100 O#3901 ;
G65 H01 P#2001 O0 ;                  G65 H01 P#101 O#3902 ;
G65 H01 P#2014 O0 ;                  G65 H01 P#3901 O#102 ;
G65 H01 P#2110 O0 ;                  G65 H01 P#3902 O#103 ;
G04 P2000 ;                          G04 P5000 ;
G04 P2000 ;                          G04 P5000 ;
G04 P2000 ;                          G04 ;
G65 H01 P#2001 O50000 ;              G65 H01 P#100 O#4001 ;
G65 H01 P#2014 O60000 ;              G65 H01 P#101 O#4002 ;
G65 H01 P#2110 O30000 ;              / G65 H01 P#102 O#4003 ;
G04 P2000 ;                          G65 H01 P#103 O#4004 ;
G04 P2000 ;                          G65 H01 P#104 O#4005 ;
G04 P2000 ;                          G65 H01 P#105 O#4006 ;
G65 H02 P#2001 O#2001 R3 ;           G65 H01 P#106 O#4007 ;
G65 H03 P#2014 O15000 R#2014         G65 H01 P#107 O#4008 ;
;                                    G65 H01 P#108 O#4009 ;
G65 H04 P#2110 O3 R#2110 ;

                                     EDIT **** *** ***
                                     07:12:55
 |  |  |  |  |  |  |          |  |O SRH|SRH=|SRHO|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                                   O1000 N00110
    (CURRENT)                 (MODAL)
G00   .X     15.000            G00    F
G90   .Y     15.000            G17
       T          2            G90    M
                               G22
                               G21
                               G40    H    D
                               G50
                               G67    T         2
                               G54
                               G64    S
                               G85

MEM STRT MTN FIN          15:06:52
[ PRGRM ] [ CHECK ] [ CURRNT ] [ NEXT ] [        ]
```

Explanations

- **12–soft key type LCD**

The current block display screen is not provided for 12–soft key type LCD. 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
 C      30.000                      PROGRAM
                                    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                                      O1000 N00120
    (CURRENT)                (NEXT)
      .X  1085.000             .X  1085.000
      .Y    15.000             .Y   635.000




MEM STRT MTN ***         15:07:29
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][        ]
```

11.2.4 Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data in the MEMORY mode.

Procedure for displaying the program check screen

Procedure

1 Press function key PROG .

2 Press chapter selection soft key **[CHECK]**.
The program currently being executed, current position of the tool, and modal data are displayed.

```
PROGRAM CHECK                            O1000 N00210
N210 G76 I40. JO K4  T03 ;
N300 G72 X400. Y80. ;
N310 G76 I40. J0 K7 ;
N400 G72 X770. Y80. ;
 (ABSOLUTE)   (DIST TO GO) G00  G21  G54
X   190.000  X     0.000  G17  G40  G64
Y    80.000  Y     0.000  G90  G50  G85
T    10.000  T     0.000  G22  G67
C     0.000  C     0.000
                          H    M
     T       3            D
     F             S
               SACT        0
MEM STRT *** FIN           15:09:40
[  ABS  ][  REL  ][       ][       ][ (OPRT) ]
```

Explanations

- **Program display**

The screen displays up to four blocks of the current program, starting from the block currently being executed. The block currently being executed is displayed in reverse video. During DNC operation, however, only three blocks can be displayed.

- **Current position display**

The position in the workpiece coordinate system or relative coordinate system and the remaining distance are displayed. The absolute positions and relative positions are switched by soft keys **[ABS]** and **[REL]**.

- **Modal G codes**

Up to 12 modal G codes are displayed.

- **Display during automatic operation**

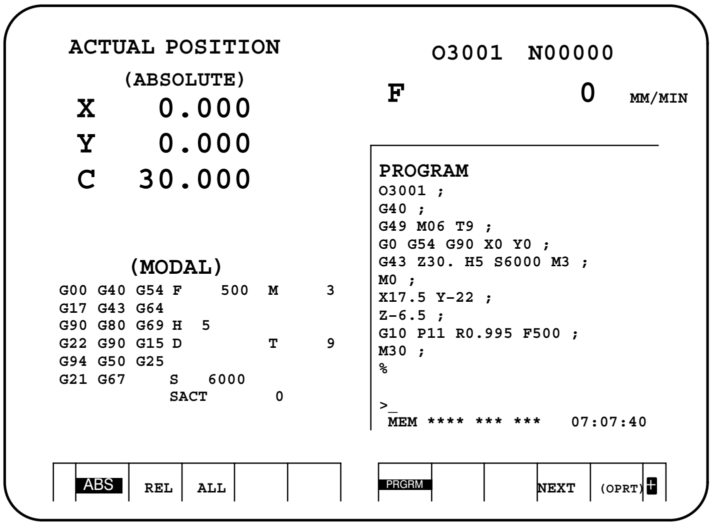
During automatic operation, the actual speed, SACT, and repeat count are displayed. The key input prompt (>_) is displayed otherwise.

- **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 key type LCD**

The program check screen is not provided for 12–soft key type LCD. 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 tool 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 (MDI)                           O1000 N00000
O0000 G00 X100. Y100. ;
G72 X200. Y200. ;
G26 I100. J0 K10 T10 ;
%



G00  G90  G21  G50  G54  G85
G17  G22  G40  G67  G64
                         H    M
     T                   D
     F              S
)_
MDI **** *** ***        15:11:15
[      ][      ][ SRH ↓ ][ SRH ↑ ][ REWIND ]
```

Explanations

- **MDI operation**

See Section 4.2 for MDI operation.

- **Modal information**

The modal data is displayed when bit 7 (MDL) of parameter 3107 is set to 1. Up to 16 modal G codes are displayed. On the 9.5″/10.4″ LCD, 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.

11.3 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 library 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 Chapter 9 for the program editing screen and conversational graphics programming screen. See Chapter 8 for the floppy file directory screen.

11.3.1 Displaying Memory Used and a List of Programs

Displays the number of registered programs, memory used, and a list of registered programs.

Procedure for displaying memory used and a list of programs

Procedure

1 Select the **EDIT** mode.

2 Press function key PROG.

3 Press chapter selection soft key **[LIB]**.

```
 PROGRAM                                O2000 N00130

    SYSTEM EDITION     B001 - 02
  PROGRAM NO.  USED :     11 FREE :     52
  MEMORY AREA  USED :   1200 FREE :   4320
PROGRAM LIBRARY LIST
  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 ][  LIB  ][       ][ C.A.P. ][ (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.
The soft key [DIR] can be used to switch between the program name screen (Fig. 11.3.1 (a)) and the program size and program update date screen (Fig. 11.3.1 (b)). 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:       17              4320
     FREE:       46              3960
   O0001   360  1966-06-12  14:40
   O0002   240  1966-06-12  14:55
   O0010   420  1966-07-01  11:02
   O0020   180  1966-08-14  09:40
   O0040  1140  1966-03-25  28:40
   O0050    60  1966-08-26  16:40
   O0100   120  1966-04-30  13:11

> _
EDIT **** *** ***     16:52:13
[ PRGRM ][  DIR  ][      ][       ][ (OPRT) ]
```

Fig. 11.3.1 (b)

If the parameter NAM (No. 3107#0) is set to 0, only program numbers are indicated.

- **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): (ΔΔΔ…Δ)

- **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**

Immediately after all programs are cleared (by turning on the power while pressing the DELETE key), each program is registered after the last program in the list.
If some programs in the list were deleted, then a new program is registered, the new program is inserted in the empty location in the list created by the deleted programs.

Example) When bit 4 (SOR) of parameter 3107 is 0

1. **After clearing all programs, register programs O0001, O0002, O0003, O0004, and O0005 in this order. The program library list displays the programs in the following order:**
   O0001, O0002, O0003, O0004, O0005

2. **Delete O0002 and O0004. The program library list displays the programs in the following order:**
   O0001, O0003, O0005

3. **Register O0009. The program library list displays the programs in the following order:**
   O0001, O0009, O0003, O0005

11.3.2 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) (      )
```

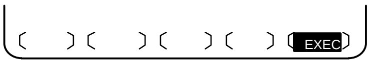
4 Press the **[(OPRT)]** operation soft key.

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 tool compensation values and other data.
This section describes how to display or set the following data:

1. Tool offset value
2. Settings
3. Run time and part count
4. Workpiece origin offset value
5. Custom macro common variables
6. Software operator's panel
7. Tool registration data
8. Safety zone data

This section also describes measurement of tool length and the sequence number comparison and stop function.
The software operator's panel 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

Cutter compensation values are specified by D codes in a program. Compensation values corresponding to D codes are displayed or set on the screen.

Procedure for setting and displaying the cutter compensation value

Procedure

1 Press function key OFFSET SETTING .

2 Press chapter selection soft key **[OFFSET]** or press OFFSET SETTING several times until the tool compensation screen is displayed.
The screen varies according to the type of tool offset memory.

```
OFFSET                                        O0001 N00000
   NO.        DATA          NO.      DATA
   001        1.000           009         0.000
   002       -2.000           010        -7.500
   003        0.000           011        12.000
   004        5.000           012       -20.000
   005        0.000           013         0.000
   006        0.000           014         0.000
   007        0.000           015         0.000
   008        0.000           016         0.000
ACTUAL POSITION (RELATIVE)
      X     0.000             Y           0.000
      C     0.000
> _
MDI **** *** ***       16:05:59
[ OFFSET ][SETING][ WORK ][      ][ (OPRT) ]
```

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key **[NO.SRH]**.

4 To set a compensation value, enter a value and press soft key **[INPUT]**. To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key **[+INPUT]**. Or, enter a new value and press soft key **[INPUT]**.

Explanations

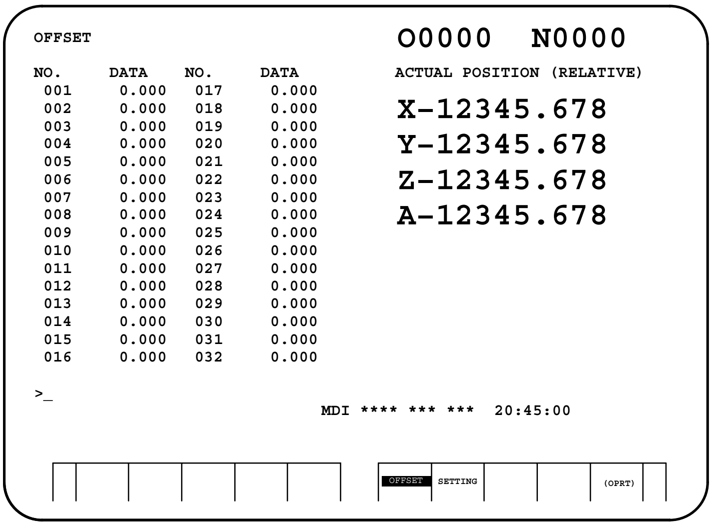
- **Decimal point input**

A decimal point can be used when entering a compensation value.

- **Other method**

An external input/output device can be used to input or output a cutter compensation value. See Chapter 8.

- **12–soft key type LCD**



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 Chapter 9 for automatic insertion of sequence numbers.
See subsection 11.4.5 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= 0 (0:OFF   1:ON)
  MIRROR IMAGE Y= 0 (0:OFF   1:ON)
  MIRROR IMAGE Z= 0 (0:OFF   1:ON)


>  _
MDI **** *** ***      16:05:59
[ OFFSET ] [SETING] [ WORK ] [     ] [ (OPRT) ]
```

4 Move the cursor to the item to be changed by pressing cursor keys ↑, ↓, ←, or →.

5 Enter a new value and press soft key **[INPUT]**.

Contents of settings

- **PARAMETER WRITE**

Setting whether parameter writing is enabled or disabled.
0 : Disabled
1 : Enabled

- **TV CHECK**

Setting to perform TV check.
0 : No TV check
1 : Perform TV check

- **PUNCH CODE**

Setting code when data is output through reader puncher interface.
0 : EIA code output
1 : ISO code output

- **INPUT UNIT**

Setting a program input unit, inch or metric system
0 : Metric
1 : Inch

- **I/O CHANNEL**

Using channel of reader/puncher interface.
0 : Channel 0
1 : Channel 1
2 : Channel 2
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.

- **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 subsection 11.4.6 for this screen.

11.4.3 Displaying and Setting Items on the Tool Registration Screens

Items concerning tools, such as the number of a tool to be used in machining, the position at which the turret is indexed for a tool, and tool position compensation, can be displayed or specified on the tool registration screens. Refer to the manual prepared by the machine tool builder for details, as the builder sets these items first.

11.4.3.1 Displaying and setting items on the initial tool registration screen

(1) Displaying the screen

1 Press the OFFSET SETTING function key.

2 Press the menu key several times until the **[TOOL]** soft key appears.

3 Press the **[TOOL]** soft key to display the initial tool registration screen.

```
TOOL SET                                    O0010 N00000

  REGISTERED    =          45

  TOOL INDEX    =          40

  ZERO POINT    =           1

  MOVEMENT      =      200000

  PUNCH COUNT   =         351


>_
MDI **** *** ***          13:38:29
[ TOOL ][       ][       ][       ][ (OPRT) ]
```

Fig. 11.4.3.1 Initial Tool Registration Screen

(2) Setting items from the MDI

1 Set the mode to MDI.

2 Press the OFFSET SETTING function key. Then press the **[SETING]** soft key to enable the parameter write operation. The CNC indicates alarm No. 100.

3 Display the initial tool registration screen by following the steps described in (1). Move the cursor to an item to be changed with the cursor keys.

4 For absolute programming, enter the data and press the **[INPUT]** soft key.
For incremental programming, enter an increment or decrement and press the **[+INPUT]** soft key.

Data items to be entered are as follows:

(a) Number of tools to be used (parameter No. 16265)
Specify the total number of tools to be used. The maximum setting is 136.

(b) Number of tools for which the turret is indexed (parameter No. 16266)
When T-axis control is specified (TCL, bit 4 of parameter No. 16260, is set to 1), specify the total number of tools for which the turret is indexed. The setting must not be greater than the number of tools to be used.
(Example)
When the number of tools to be used is 50 and the number of tools for which the turret is indexed is 45, the T-axis control applies to tool Nos. 001 to 045 and does not apply to tool Nos. 046 to 050. For the tool numbers, see Item 11.4.3.2.

(c) Number of a tool to be initially selected (parameter No. 16267)
When T-axis control is specified, specify the number of the tool to be selected when reference position return is completed after power-on. Settings range from 0 to 9999.
The tool must be mounted at the 0-position of the turret. The tool also needs to be registered on the tool number registration screen (see Item 11.4.3.2) before the number of the tool is specified.

(d) Number of pulses sent to the turret per rotation (parameter No. 16268)
When T-axis control is specified, specify the number of pulses sent to the turret per rotation.

(e) Number of punch operations (parameter No. 16269)
Specify the number of punch operations for the tool used. Settings range from 0 to 99999999.

NOTE
Data items (a) to (e) can also be specified on the parameter screen.

11.4.3.2 Displaying and setting items on the tool number registration screen

The numbers of the tools to be used, tool position compensation, and turret positions (mechanical positions around the T-axis) indexed for tools can be displayed and specified.

(1) Displaying the screen

1 Press the OFFSET SETTING function key.

2 Press the ▷ menu key several times until the **[TOOL]** soft key appears.

3 Press the **[TOOL]** soft key to display the initial tool registration screen (Fig. 11.4.3.1).

4 Press the ▷ menu key. Then press the **[T.NO]** soft key to display the tool number registration screen.

```
TOOL SET  (NUMBER)                        O0010 N00000
 NO.  TOOL    POSITION    X-OFFSET    Y-OFFSET
 001  0001           0           0           0
 002  0002        5000           0           0
 003  0003       10000           0           0
 004  0004       15000           0           0
 005  0005       20000           0       50000
 006  0006       25000           0           0
 007  0007       30000           0           0
 008  0008       35000           0           0
 009  0009       40000           0       50000
 010  0010       45000           0           0

>_
MDI **** *** ***        13:38:58
[ T.NUM. ][ T.CHG.][ T.CNT.][ SHAPE ][(OPRT)]
```

Fig. 11.4.3.2 Tool number registration screen

The first line indicates the title and the second line contains the tool data. The leftmost number is a tool registration number.

(2) Setting items from the MDI

1 Set the mode to MDI.

2 Press the OFFSET SETTING function key. Then press the **[SETING]** soft key to enable the parameter write operation. The CNC indicates alarm No. 100.

3 Display the tool number registration screen by following the steps described in (1).

4 Move the cursor to the item to be changed.

<u>Method 1</u>
Move the cursor to the item to be changed with the page keys and cursor keys.

<u>Method 2</u>
Change the mode of the soft keys to the operation selection mode using the **[(OPRT)]** soft key. Enter the registration number of the tool for which data is to be changed, then press the **[NO.SRH]** soft key. Move the cursor to the item to be changed with the cursor keys.

5 For absolute programming, enter the data and press the **[INPUT]** soft key.
For incremental programming, enter an increment or decrement and press the **[+INPUT]** soft key.

Data items to be entered are as follows:

(a) Tool number
Specify the numbers of tools to be used. Up to 136 numbers can be entered. Settings range from 0 to 9999.

(b) Turret position
When T-axis control is specified (TCL, bit 4 of parameter No. 16260, is set to 1), specify the positions at which the turret is indexed for tools. Settings range from 0 to 99999999.

(c) X-axis offset
When the tool position compensation function is used (TOF, bit 2 of parameter No. 16263 is set to 1), specify X-axis offset for tool positions in the machine coordinate system. Settings range from –99999999 to +99999999.

(d) Y-axis offset
When the tool position compensation function is usedspecify Y-axis offset for tool positions in the machine coordinate system. Settings range from –99999999 to +99999999.

CAUTION
The X- and Y-axis offset must be 0 for the tool to be initially selected, described in Item 11.4.3.1.

11.4.3.3 Displaying and setting items on the screen for entering the numbers of tools used for replacement

When the tool change function is used, the numbers of tools to be substituted for tools registered on the tool number registration screen (Item 11.4.3.2) can be displayed and specified.

(1) Displaying the screen

1 Press the OFFSET SETTING function key.

2 Press the ▷ menu key several times until the **[TOOL]** soft key appears.

3 Press the **[TOOL]** soft key to display the initial tool registration screen (Fig. 11.4.3.1).

4 Press the ▷ menu key. Then press the **[T.CHG.]** soft key to display the screen for entering the numbers of tools used for replacement.

```
TOOL SET    (CHANGE)                         O0010 N00000
 NO. DATA   NO.  DATA   NO.  DATA   NO.  DATA
 001     9  011    19   021    29   031    39
 002    10  012    20   022    30   032    40
 003    11  013    21   023    31   033     1
 004    12  014    22   024    32   034     2
 005    13  015    23   025    33   035     3
 006    14  016    24   026    34   036     4
 007    15  017    25   027    35   037     5
 008    16  018    26   028    36   038     6
 009    17  019    27   029    37   039     7
 010    18  020    28   030    38   040     8


>_
MDI **** *** ***          13:39:33
[ T.NUM. ][ T.CHG. ][ T.CNT.][ SHAPE ][(OPRT)]
```

Fig. 11.4.3.3 Screen for Entering the Numbers of Tools Used for Replacement

A number indicated on the screen corresponds to the tool registration number on the tool registration screen (see Item 11.4.3.2).

(2) Setting items from the MDI

1 Set the mode to MDI.

2 Press the OFFSET SETTING function key. Then press the **[SETING]** soft key to enable the parameter write operation. The CNC indicates alarm No. 100.

3 Display the screen for entering the numbers of tools used for replacement by following the steps described in (1).

4 Move the cursor to the item to be changed.

<u>Method 1</u>
Move the cursor to the item to be changed with the page keys and cursor keys.

Method 2
Change the mode of the soft keys to the operation selection mode using the **[(OPRT)]** soft key. Enter the registration number of the tool for which data is to be changed, then press the **[NO.SRH]** soft key.

5 For absolute programming, enter the data and press the **[INPUT]** soft key.
For incremental programming, enter an increment or decrement and press the **[+INPUT]** soft key.
Tool numbers to be specified must have been registered on the tool registration screen. Settings range from 0 to 9999.

11.4.3.4 Displaying and setting items on the screen for the number of press operations

The number of punch operations using each tool registered on the tool number registration screen (see Item 11.4.3.2) can be displayed and specified.

(1) Displaying the screen

1 Press the OFFSET SETTING function key.

2 Press the ▷ menu key several times until the **[TOOL]** soft key appears.

3 Press the **[TOOL]** soft key to display the tool registration initial screen (Fig. 11.4.3.1).

4 Press the ▷ menu key. Then press the **[T.CNT.]** soft key to display the screen for the number of punch operations.

```
TOOL SET  (COUNT)                         O0010 N00000
 NO.     COUNT   NO.    COUNT   NO.    COUNT
 001        23   011        0   021        0
 002        23   012      110   022       36
 003        15   013        0   023        0
 004        20   014        2   024        0
 005         2   015       13   025       24
 006         8   016        0   026        0
 007         0   017        0   027        0
 008         0   018       42   028        0
 009        40   019        0   029        0
 010         0   020        0   030        0

>_
MDI **** *** ***        13:39:58
[ T.NUM. ][ T.CHG.][ T.CNT. ][ SHAPE ][(OPRT)]
```

Fig. 11.4.3.4 Screen for the Number of Punch Operations

The number indicated on the screen corresponds to the tool registration number on the tool registration screen (see Item 11.4.3.2).

(2) Setting items from the MDI

1 Set the mode to MDI.

2 Press the OFFSET SETTING function key. Then press the **[SETING]** soft key to enable the parameter write operation. The CNC indicates alarm No. 100.

3 Display the screen for the number of punch operations by following the steps described in (1).

4 Move the cursor to an item to be changed.

<u>Method 1</u>
Move the cursor to the item to be changed with the page keys and cursor keys.

<u>Method 2</u>
Change the mode of the soft keys to the operation selection mode using the **[(OPRT)]** soft key. Enter the registration number of the tool for which data is to be changed, then press the **[NO.SRH]** soft key.

5 For absolute programming, enter the data and press the **[INPUT]** soft key.
For incremental programming, enter an increment or decrement and press the **[+INPUT]** soft key.
Settings range from 0 to 99999999.

11.4.3.5 Displaying and setting items on the tool figure registration screen (for drawing figures)

When the optional graphics function is used, the figures of tools registered on the tool number registration screen (see Item 11.4.3.2) can be specified. These figures are drawn with the graphics function.

(1) Displaying the screen

1 Press the OFFSET SETTING function key.

2 Press the menu key several times until the **[TOOL]** soft key appears.

3 Press the **[TOOL]** soft key to display the initial tool registration screen (Fig. 11.4.3.1).

4 Press the menu key. Then press the **[SHAPE]** soft key to display the tool figure registration screen.

```
TOOL SET  (SHAPE)                       O0010 N00000
 NO. SHAPE(C)    SIZE(I)   SIZE(J)   ANGLE(K)
 001       01      20000         0          0
 002       02      50000     50000          0
 003       02      50000     20000          0
 004       01      15500         0          0
 005       01      30000         0          0
 006       01       5000         0          0
 007       03      12000      5000      45000
 008       03      10000      5000          0
 009       02     110000    510000      20000
 010       01       1000         0          0

>_
MDI **** *** ***         13:40:24
[ T.NUM. ][ T.CHG.][ T.CNT.][ SHAPE  ][(OPRT)]
```

Fig. 11.4.3.5 Tool figure registration screen

The number indicated on the screen corresponds to the tool registration number on the tool registration screen (see Item 11.4.3.2).

(2) Setting items from the MDI

1 Set the mode to MDI.

2 Press the OFFSET SETTING function key. Then press the **[SETING]** soft key to enable the parameter write operation. The CNC indicates alarm No. 100.

3 Display the tool figure registration screen for drawing figures by following the steps described in (1).

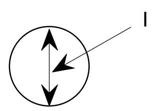
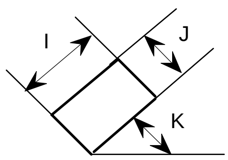
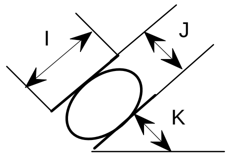
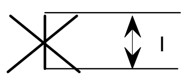
4 Move the cursor to an item to be changed.

<u>Method 1</u>
Move the cursor to the item to be changed with the page keys and cursor keys.

<u>Method 2</u>
Change the mode of the soft keys to the operation selection mode using the **[(OPRT)]** soft key. Enter the registration number of the tool for which data is to be changed, then press the **[NO.SRH]** soft key. Move the cursor to the item to be changed with the cursor keys.

5 For absolute programming, enter the data and press the **[INPUT]** soft key.
For incremental programming, enter an increment or decrement and press the **[+INPUT]** soft key.

The following table shows tool figure data.

	Figure			
	C	**I (Dimension X)**	**J (Dimension Y)**	**K (Angle)**
Circle	01		I : 0 to 999999 (Input unit)	
Square, rectangle	02		I : 0 to 999999 J : 0 to 999999 K : 0 to 359999 (Input unit)	
Elongated hole	03		I : 0 to 999999 J : 0 to 999999 K : 0 to 359999 (Input unit)	
Others	00		I : 25.4mm (1 inch) Fixed value	

The upper digit of the tool figure (C) specifies the color of a line drawing and the lower digit specifies the figure.

Upper digit	Lower digit
0 : Green	0 : Asterisk
1 : Red	1 : Circle (line drawing)
2 : Green	2 : Quadrangle (line drawing)
3 : Yellow	3 : ellipse (line drawing)
4 : Blue	4 : Circle (filled–in drawing)
5 : Pink	5 : Quadrangle (filled–in drawing)
6 : Sky blue	6 : ellipse (filled–in drawing)
7 : White	
8 : Gray	

11.4.4 Displaying and Setting Items on the Safety Zone Setting Screen

When the optional safety zone check function is used, the current safety zone can be displayed and changed.

(1) Displaying the screen

1 Press the OFFSET SETTING function key.

2 Press the ▷ menu key several times until the **[SAFETY]** soft key appears.

3 Press the **[SAFETY]** soft key to display the safety zone setting screen.

```
SAFETY ZONE (ABSOLUTE)                   O0017 N01234
 AREA #1                   AREA #3
  X2=   100.000             X2=  1000.000
  X1=   200.000             X1=  1150.000
  Y =   100.000             Y =   110.000
 AREA #2                   AREA #4
  X2=   500.000             X2=  1400.000
  X1=   600.000             X1=  1550.000
  Y =   100.000             Y =   110.000
 TOOL ZONE
  X =     5.000
  Y =    10.000

)_
MEM **** *** ***          11:32:41
[ TOOL ][       ][ SAFETY ][       ][ (OPRT) ]
```

Fig. 11.4.4 Safety zone setting screen

The safety zone is displayed in the workpiece coordinate system. By setting MDP, bit 0 of parameter No. 16502, the zone can be displayed in the machine coordinate system.

(2) Setting items from the MDI

By setting SZI, bit 4 of parameter No. 16502, data for safety zones can be changed.

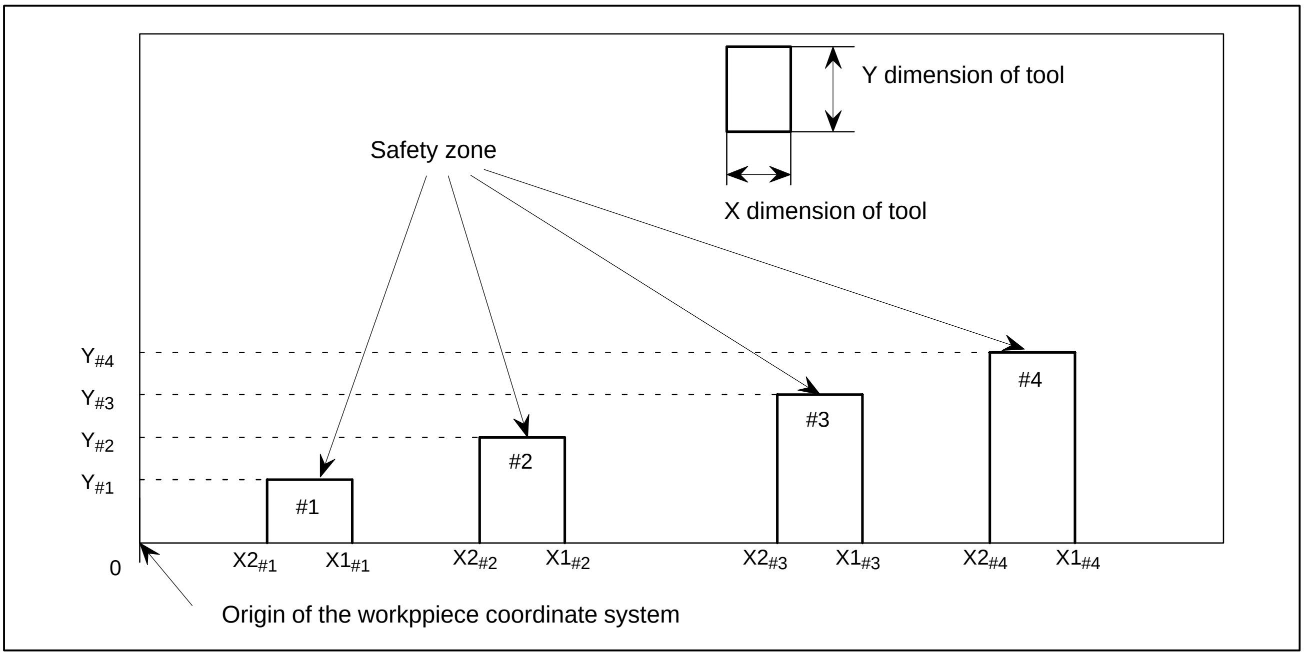
1 Set the mode to MDI.

2 Display the safety zone setting screen by following the steps described in (1).

3 Move the cursor to the safety zone data to be changed with the cursor keys.

4 For absolute programming, enter the data and press the **[INPUT]** soft key.
For incremental programming, enter an increment or decrement and press the **[+INPUT**] soft key.

Safety zone data is as follows:



(a) Safety zone #n (n: 1 to 4) (parameters No. 16505 to No. 16516)
Up to four safety zones can be specified (Optionally, up to eight safety zones can be specified). When a zone is specified in the workpiece coordinate system, the data is converted to that in the machine coordinate system and stored in the parameters.
Safety zone #n must be located closer to the origin than safety zone #n+1.
Settings range from –99999999 to +99999999.

NOTE
For unnecessary safety zones, specify the coordinates of a tool located when it returns to the reference position.
Workpiece coordinate system : Coordinates of a position for setting the automatic coordinate system (specified in parameter No. 1250)
Machine coordinate system : 0

(b) Tool position (parameters No. 16517 to No. 16532)
Specify the dimensions of a tool along the X- and Y-axes. Settings range from 0 to 99999999.

NOTE
Data items (a) and (b) can also be specified on the parameter screen.

11.4.5 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.

- **Stop in the canned cycle**

If the predetermined sequence number is found in a block which has a canned–cycle command, the execution of the program stops after the return operation is completed.

- **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.

11.4.6 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 =  2001/07/05
                 TIME =    11:32:52

> _                                     S    0  T0000
MDI **** *** ***        16:05:59
[ OFFSET ] [ SETING ] [ WORK ] [      ] [ (OPRT) ]
```

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.

6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press soft key **[INPUT]**.

Display items

• PARTS TOTAL

This value is incremented by one when M02, M30, or an M code specified by parameter 6710 is executed. This value cannot be set on this screen. Set the value in parameter 6712.

• PARTS REQUIRED

It is used for setting the number of machined parts required.
When the “0” is set to it, there is no limitation to the number of parts. Also, its setting can be made by the parameter (NO. 6713).

- **PARTS COUNT**

This value is incremented by one when M02, M30, or an M code specified by parameter 6710 is executed. The value can also be set by parameter 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- **POWER ON**

Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter 6750.

- **OPERATING TIME**

Indicates the total run time during automatic operation, excluding the stop and feed hold time.
This value can be preset in parameter 6751 or 6752.

- **CUTTING TIME**

Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter 6753 or 6754.

- **FREE PURPOSE**

This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- **CYCLE TIME**

Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- **DATA and TIME**

Displays the current date and time. The date and time can be set on this screen.

Limitations

- **Usage**

When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter (NO. 6710) is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (parameter PCM (No. 6700#0) is set to 1). For details, see the manual issued by machine tool builders.

Restrictions

- **Run time and part count settings**

Negative value cannot be set. Also, the setting of “M” and “S” of run time is valid from 0 to 59.
Negative value may not be set to the total number of machined parts.

- **Time settings**

Neither negative value nor the value exceeding the value in the following table can be set.

Item	Maximum value	Item	Maximum value
Year	2085	Hour	23
Month	12	Minute	59
Day	31	Second	59

11.4.7 Displaying and Setting the Workpiece Origin Offset Value

Displays the workpiece origin offset for each workpiece coordinate system (G54 to G59) and external workpiece origin offset. The workpiece origin offset and external workpiece origin offset can be set on this screen.

Procedure for Displaying and Setting the Workpiece Origin Offset Value

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
          C   0.000                     C   112.000

   01     X  20.000               03    X   300.000
  (G54)   Y  50.000              (G56)  Y   200.000
          C  30.000                     C   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:

3–1 Press the page up PAGE ↑ or page down PAGE ↓ key.

3–2 Enter the workpiece coordinate system number (0:external workpiece origin offset, 1 to 6: workpiece coordinate systems G54 to G59) and press operation selection soft key **[NO.SRH]**.

4 Turn off the data protection key to enable writing.

5 Move the cursor to the workpiece origin offset to be changed.

6 Enter a desired value by pressing numeric keys, then press soft key **[INPUT]**. The entered value is specified in the the workpiece origin offset value. Or, by entering a desired value with numeric keys and pressing soft key **[+INPUT]**, the entered value can be added to the previous offset value.

7 Repeat **5** and **6** to change other offset values.

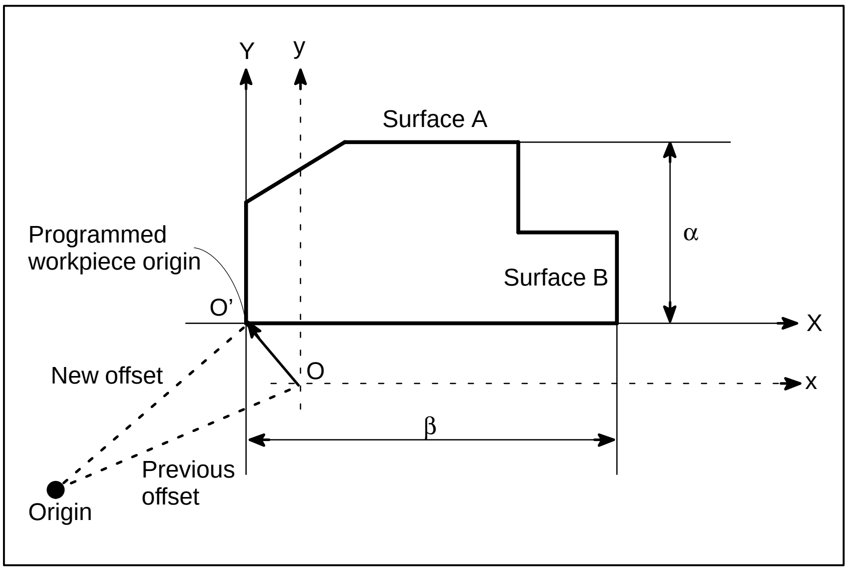
8 Turn on the data protection key to disable writing.

11.4.8 Input of Measured Workpiece Origin Offsets

This function is used to compensate for the difference between the programmed workpiece coordinate system and the actual workpiece coordinate system. The measured offset for the origin of the workpiece coordinate system can be input on the screen such that the command values match the actual dimensions.
Selecting the new coordinate system matches the programmed coordinate system with the actual coordinate system.

Procedure for Inputting of Measured Workpiece Origin Offsets

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 tool without changing the Y coordinate.

3 Measure distance α between surface A and the programmed origin of the workpiece coordinate system as shown above.

4 Press function key OFFSET SETTING .

5 To display the workpiece origin offset setting screen, press the chapter selection soft key **[WORK]**.

```
WORK COORDINATES                      O1234 N56789
(G54)

    NO.      DATA          NO.      DATA
    00    X    0.000       02    X    0.000
   (EXT)  Y    0.000      (G55)  Y    0.000
          C    0.000             C    0.000


    01    X    0.000       03    X    0.000
   (G54)  Y    0.000      (G56)  Y    0.000
          C   0.000              C    0.000

> Z100.                          S   0  T0000
MDI **** *** ***      16:05:59
[ NO.SRH ][ MEASUR ][      ][ +INPUT ][ INPUT ]
```

6 Position the cursor to the workpiece origin offset value to be set.

7 Press the address key for the axis along which the offset is to be set (Y–axis in this example).

8 Enter the measured value (α) then press the **[MEASUR]** soft key.

9 Move the reference tool manually until it touches surface B of the workpiece.

10 Retract the tool without changing the X coordinate.

11 Measure distance β then enter the distance at X on the screen in the same way as in steps 7 and 8.

Limitations

- **Consecutive input**

Offsets for two or more axes cannot be input at the same time.

- **During program execution**

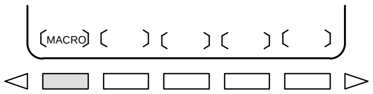
This function cannot be used while a program is being executed.

11.4.9 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #199 and #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



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
     C      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:

3–1 Enter the variable number and press soft key **[NO.SRH]**.

3–2 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 C , then press soft key **[INP.C.]**.

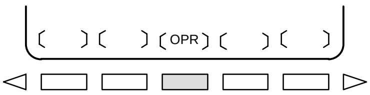
6 To set a blank in a variable, just press soft key **[INPUT]**. The value field for the variable becomes blank.

11.4.10 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.
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.

```
OPERATOR'S PANEL                    O0000 N00000

MODE :  MDI  AUTO  EDIT  HNDL  JOG  ZRN

 STEP MULTI. :  *1   *10   100
 RAPID OVRD. : 100%  50%   25%   F0
 JOG FEED  :   2.0%
              **************
 FEED OVRD.  :  100%
                      ***
ACTUAL POSITION (ABSOLUTE)
    X     0.000        Y     0.000
    C     0.000
>_
MDI **** *** ***      16:05:59
[ MACRO ] [     ] [  OPR  ] [TOOLLF] [ (OPRT) ]
```

```
OPERATOR'S PANEL                    O0000 N00000

BLOCK SKIP     :   OFF    ON
SINGLE BLOCK   :   OFF    ON
MACHINE LOCK   :   OFF    ON
PROTECT KEY    :   PROTECT    RELEASE
FEED HOLD      :   OFF

ACTUAL POSITION (ABSOLUTE)
    X     0.000        Y     0.000
    C     0.000

                              S    0   T0000
MDI **** *** ***      16:05:59
[ MACRO ] [     ] [  OPR  ] [TOOLLF] [ (OPRT) ]
```

4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓ .

5 Push the cursor move key ← or → to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform jog feed. Press the 5 key together with an arrow key to perform jog rapid traverse.

8↑ 9↗

4← 5 6→

1↙ 2↓

Explanations

● Valid operations

The valid operations on the software operator's panel are shown below. Whether to use the 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 LCD/MDI indicates other than the software operator's panel screen and diagnostic screen, jog feed is not conducted even if the arrow key is pushed.

● Jog feed and arrow keys

The feed axis and direction corresponding to the arrow keys can be set with parameters (Nos. 7210 to 7217).

● General purpose switches

Eight optionally definable switches are added as an extended function of the software operator's panel. The name of these switches can be set by parameters (Nos. 7220 to 7283) as character strings of max. 8 characters. For the meanings of these switches, refer to the manual issued by machine tool builder.

11.5 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM

When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.

This chapter describes how to set parameters on the MDI panel. Parameters can also be set with external input/output devices such as the Handy File (see Chapter 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 Chapter 7 for the diagnostic screens displayed by pressing function key SYSTEM .

11.5.1 Displaying and Setting Parameters

When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.
Normally, the user need not change parameter setting.

Procedure for displaying and setting parameters

1 Set 1 for **PARAMETER WRITE** to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key [SYSTEM] .

3 Press chapter selection soft key **[PARAM]** to display the parameter screen.

```
PARAMETER (SETTING)                 O0010 N00002

   0000       SEQ                 INI  ISO   TVC
                0    0     0    0     0    0

   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

Procedure

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 alarm No. 000 has occurred, however, turn off the power supply and then turn it on, otherwise the alarm is not released.

Explanations

- **Setting parameters with external input/output devices**

See Chapter 8 for setting parameters with external input/output devices such as the Handy File.

- **Parameters that require turning off the power**

Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm 000. In this case, turn off the power, then turn it on again.

- **Parameter list**

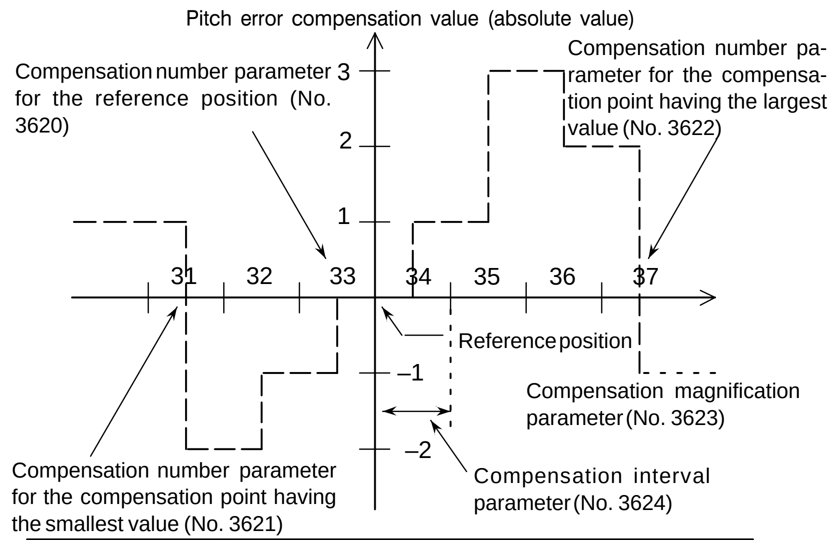
Refer to the FANUC Series 16*i*/18*i*–B Parameter Manual (B–63530EN) for the parameter list.

- **Setting data**

Some parameters can be set on the setting screen if the parameter list indicates ”Setting entry is acceptable”. Setting 1 for **PARAMETER WRITE** is not necessary when three parameters are set on the setting screen.

11.5.2 Displaying and Setting Pitch Error Compensation Data

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.
Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.
The pitch error compensation data is set according to the characteristics of the machine connected to the NC. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced. In principle, the end user must not alter this data.
Pitch error compensation data can be set with external devices such as the Handy File (see Chapter 8). Compensation data can also be written directly with the MDI panel.
The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation point number set by these parameters.
In the following example, 33 is set for the pitch error compensation point at the reference position.



Compensation position number	31	32	33	34	35	36	37
Compensation value to be set	+3	–1	–1	+1	+2	–1	–3

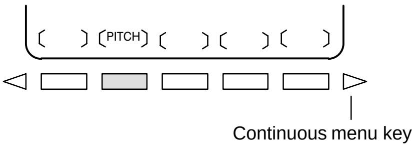
- Number of the pitch error compensation point at the reference position (for each axis) : Parameter 3620
- Number of the pitch error compensation point having the smallest value (for each axis) : Parameter 3621
- Number of the pitch error compensation point having the largest value (for each axis) : Parameter 3622
- Pitch error compensation magnification (for each axis) : Parameter 3623
- Interval of the pitch error compensation points (for each axis) : Parameter 3624
- Travel distance per revolution of pitch error compensation of the rotary axis type (for each axis): Parameter 3625

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 3620
- Number of the pitch error compensation point having the smallest value (for each axis): Parameter 3621
- Number of the pitch error compensation point having the largest value (for each axis): Parameter 3622
- Pitch error compensation magnification (for each axis): Parameter 3623
- Interval of the pitch error compensation points (for each axis): Parameter 3624
- Travel distance per revolution of pitch error compensation of the rotary axis type (for each axis): Parameter 3625



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.

```
PROGRAM                              O1000 N00000
O1000 ;
N100 G92 G00 X1270 Y1270.  ;
N110 G90 G00 X15. Y15. T02 ;
N120 X1085. Y15. ;
N130 X1085. Y635. ;
N140 X15.  Y635. ;
N200 G72 X150. Y80. ;
N210 G76 I40. J0 K4 T03 ;
N300 G72 X400. Y80. ;
N310 G76 I40. J0 K7 ;
N400 G72 X770. Y80. ;

> _
EDIT **** *** ***          15:05:41
[ PRGRM ][  LIB  ][       ][ C.A.P. ][ (OPRT) ]
```

Sequence No.
Program No.

The program number and sequence number displayed depend on the screen and are given below:

On the program screen in the EDIT mode on Background edit screen : The program No. being edited and the sequence number just prior to the cursor are indicated.

Other than above screens :

The program No. and the sequence No. executed last are indicated.

Immediately after program number search or sequence number search :

Immediately after the program No. search and sequence No. search, the program No. and the sequence No. searched are indicated.

11.6.2 Displaying the Status and Warning for Data Setting or Input/Output Operation

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the 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

```
(9) Data is out of range.
(Note)  Actually, this is displayed in the area starting from (2).

              (5)
              —EMG—      (Note)  Actually, 5 is displayed in
                         the area for (3) and (4).

(1)    (2)     (3)    (4)   (6)   (7)       (8)
EDIT   STOP    MTN    FIN   ALM   hh:mm:ss  INPUT

               (Display soft keys)
```

(1) Current mode

MDI : Manual data input
MEM : Automatic operation
RMT : Automatic operation (Tape 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 hasbeen 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 restarte.
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

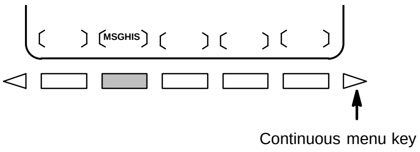
External operator messages can be preserved as history data.
Preserved history data can be displayed on the external operator message history screen.

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)

```
MESSAGE HISTORY                          O0000 N00000
  01/01/01 17:25:00                            PAGE:1
    NO. ****



 MEM  STRT  MIN  FIN  ALM  09:36:48
[          ] [MSGHIS] [          ] [          ] [(OPRT)]
```

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.

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 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

When programming is completed, the graphic function can be used to check whether machining will be performed as desired without operating the machine by drawing the programmed tool path and machining profile on the graphic display screen. After checking the profile, perform machining.
The function has the following features to simplify locating an error in the machining program:

(i) Can draw a blank figure (workpiece).

(ii) Can draw the figure of a tool used at the position where punching is performed.
A circle, a rectangle, and a capsule can be used for drawing the tool figure. The figures of other tools are drawn using asterisks.

(iii) Can draw a workpiece holder that holds a blank.
The operator can check whether punching is performed at the position of the workpiece holder. This is used to ensure that the workpiece holder is not punched during machining.

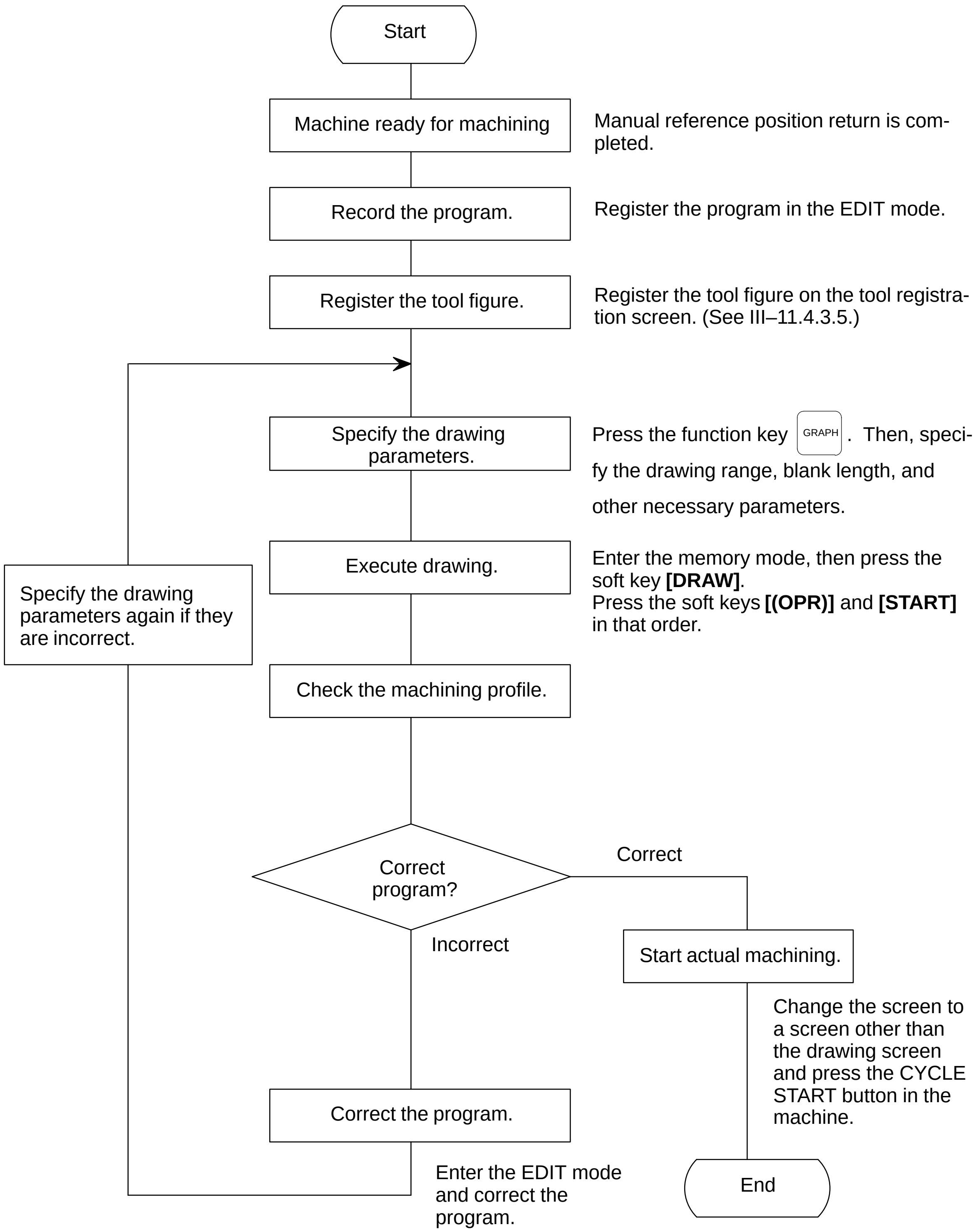
(iv) Can draw a tool path.
The tool path is drawn using a dashed line for rapid traverse and a solid line for cutting feed.

(v) Can specify a drawing plane.
A drawing plane can be selected from four types. Drawing can be performed as if a blank is set in the machine.

(vi) Checks the format of the program.
The format of the program, stored stroke limit 1, and the safety zone are checked during drawing.

12.1 OPERATION

The following flowchart shows an example of drawing a programmed figure on the screen. Refer to the flowchart if you forget the procedure.



12.2 REGISTERING THE TOOL FIGURE

To draw a machining profile, register the dimensions of the tool on the tool figure registration screen. (See III–11.4.3.5.)

12.3 SPECIFYING DRAWING PARAMETERS

Specify the parameters for graphic drawing.

(1) Procedure

1 Press the function key GRAPH . The graphic parameter setting screen appears. If it does not appear, press the soft key **[PARA]**.

```
GRAPHIC PARAMETER                         O1234 N00200
 AXES   ( 0,1,2,3,4 )                     P=         0

 RANGE                 (MAX.)
  X=   1150000   Y=      700000  Z=             0
 RANGE                 (MIN.)
  X=   -100000   Y=     -100000  Z=             0
 SCALE           K=          10
 GRAPHIC CENTER
  X=    525000   Y=      300000  Z=             0
 PROGRAM STOP    N=           0
 AUTO ERASE      A=           1

>_
MEM **** *** ***            16:19:18
[ PARAM ] [ GRAPH ] [        ] [        ] [        ]
```

Fig. 12.3 (a) Graphic parameter setting screen

```
GRAPHIC PARAMETER                         O1234 N00200
 START POINT
  X=         0   Y=           0  Z=             0
 WORK LENGTH
  X=   1100000   Y=      650000
 RAPID PATH
 (1:ON 0:OFF)    P=           0
 HOLDER POSITION      HOLDER LENGTH
 X1=    300000   X=       40000  Y=         20000
 X2=    700000   X=       40000  Y=         20000




>_
MEM **** *** ***            16:23:21
[ PARAM ] [ GRAPH ] [        ] [        ] [        ]
```

Fig. 12.3 (b) Graphic parameter setting screen

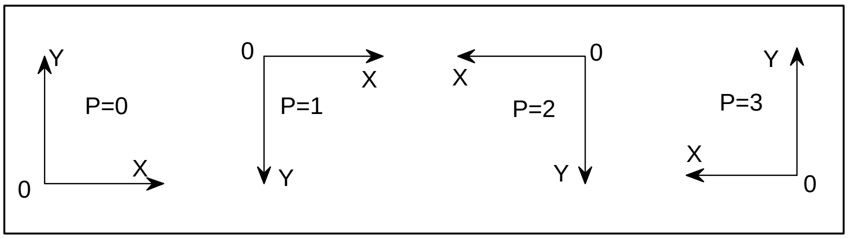
2 Move the cursor to the parameter to be changed by pressing the cursor keys.

3 Enter new data and press the INPUT key.

The parameters are described below:

(a) Drawing plane

This parameter specifies a plane for drawing.



(b) Drawing range (maximum and minimum values)

This parameter specifies the desired drawing range for each axis using the maximum and minimum values. Specifying these values automatically determines the magnification for a screen so that the drawing can be made using the whole drawing range. The center of the drawing range matches that of the screen during drawing. (Setting: 0 to ± 99999999, unit is specified in the parameter)

> **NOTE**
> The drawing range cannot be specified for the Z-axis, however.

(c) Magnification

This parameter specifies the screen magnification. The magnification refers to the magnification to the center of the screen.
Specifying the drawing range in (b) automatically determines the effective magnification. To magnify part of a figure, specify another magnification based on the automatically specified value.
(Setting: 0 to 10000 multiplied by 0.01)

(d) Coordinates based on the center of a screen

This parameter specifies the coordinates in the workpiece coordinate system whose origin matches the center of the screen.
Specifying the drawing range in (b) automatically determines the coordinates. To magnify part of the figure, specify other coordinates based on the automatically specified coordinates.
(Setting: 0 to ± 99999999, unit is specified in the parameter)

> **NOTE**
> The coordinates cannot be specified for the Z-axis, however.

(e) Drawing end block

To display part of the programmed figure, specify the sequence number of the end block. Once the figure is drawn, the sequence number is automatically canceled.

(f) Automatic deletion

1 : Previously drawn figures are automatically deleted when automatic operation is started in the reset state.

0 : Previously drawn figures are not automatically deleted.

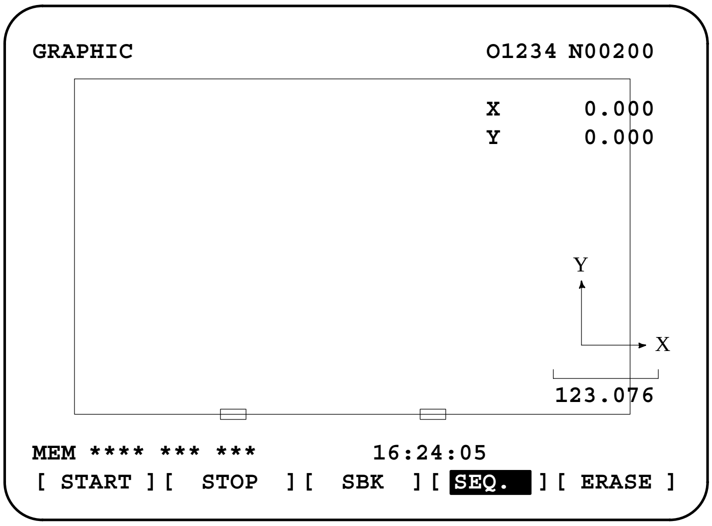
(g) Drawing start position

When a coordinate system command, G92, is not specified in a drawing program, this parameter specifies the drawing start position using the coordinates in the workpiece coordinate system.
(Setting: 0 to ± 99999999, unit is specified in the parameter)

> **NOTE**
> The drawing start position cannot be specified for the Z-axis, however.

(h) Blank length

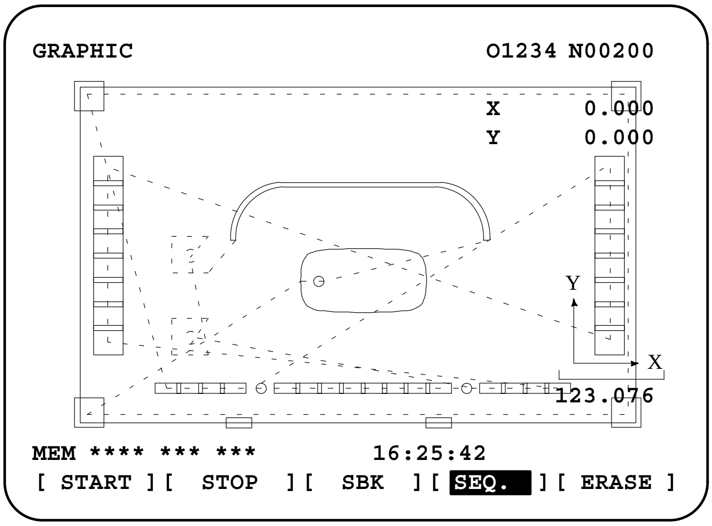
This parameter specifies the blank length along the X-axis and Y-axis. Setting this parameter draws a blank figure when the drawing screen is selected or when the drawing soft key **[ERASE]** is pressed.
The end of the blank figure in the drawing plane described in (a) is used as the origin of the workpiece coordinate system.
(Setting: 0 to ± 99999999, unit is specified in the parameter)



(i) Rapid traverse

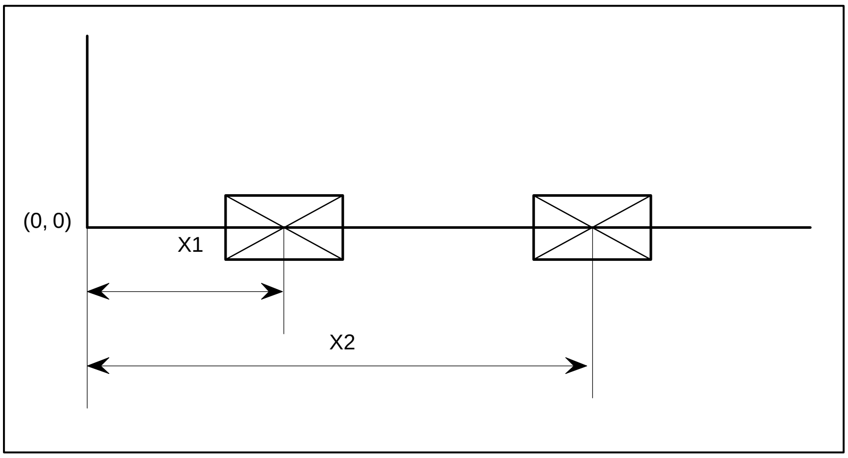
1 : A tool path for rapid traverse is drawn as a dotted line.

0 : No tool path for a rapid traverse is drawn.



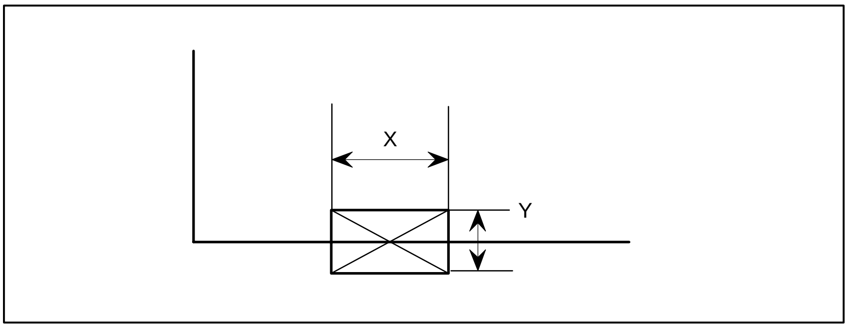
(j) Position of a workpiece holder

This parameter specifies the position of a workpiece holder in the workpiece coordinate system.
(Setting: 0 to ± 99999999, increment system)



(k) Length of a workpiece holder

This parameter specifies the horizontal length and vertical length of a workpiece holder.
(Setting: 0 to ± 99999999, unit is specified in the parameter)

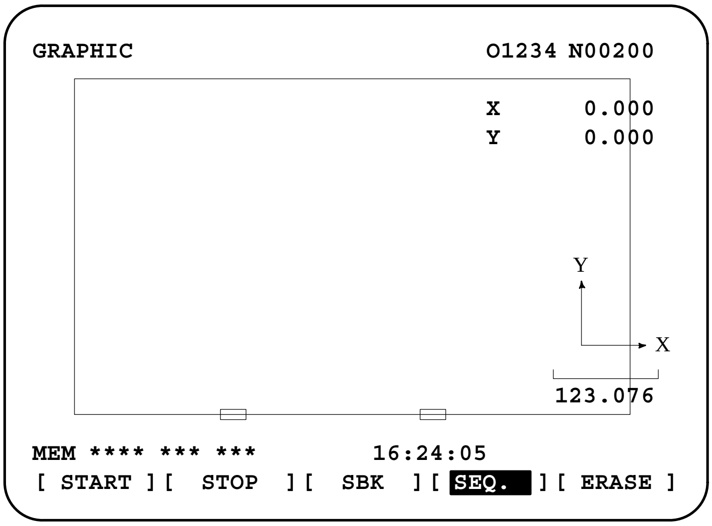


NOTE
Data items (j) and (k) for the workpiece holder cannot be specified. Specifying the safety zone displays the figure of the workpiece holder (That is set for #1 to #4.).

12.4 GRAPHIC DISPLAY SCREEN AND DRAWING

1) Drawing screen

Press **[GRAPH]** key after pressing GRAPH key, the following graphic display screen appears.



Be selecting this screen, tool paths and holes figures are drawable for checking NC programs.

2) Software keys, status display for drawing, and ruler

1 MDI address keys are used for drawing.

[START] key : Drawing start key

[STOP] key : Drawing stop key

[ERASE] key : Graphic display erasing key

[SEQ] key : Continuous drawing mode key

[SBK] key : Single block drawing mode key

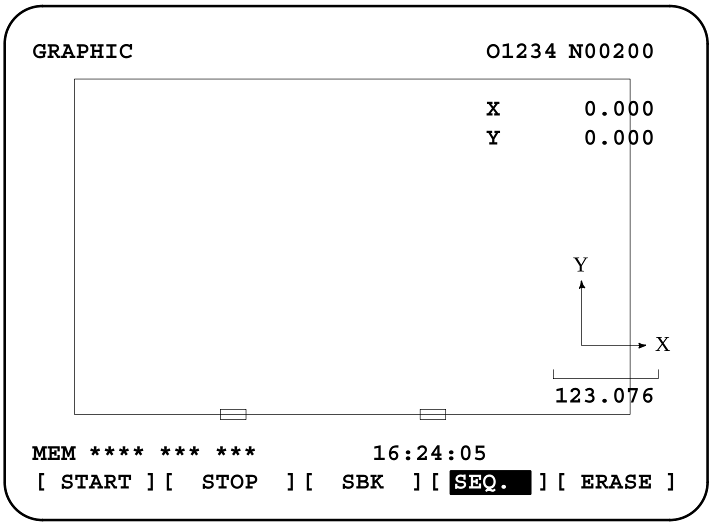
2 Ruler

The ruler is displayed at the lower right part of the screen. The value below shows the length to the line (Input unit).

3) Drawing operation

1 Select either MEMORY mode.
(This is the preparation for automatic operation, such as heading of a program, etc.)

2 Depress **[ERASE]** key. ....
Depress this key when the previous graphic display is not required. After erasing graphic display, the work sheet figure and work holder mark are drawn as shown in the left figure.



3 Depress **[SEQ]** and **[START]** keys (continuous drawing). ....
Drawing is started and continued up to the end of the NC program.

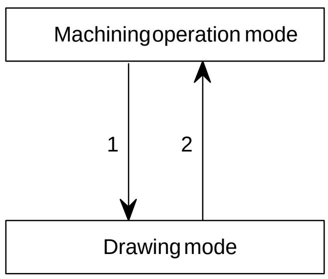
4 Depress **[SBK]** and **[*SEQ]** keys (Single block drawing). ....
Operation stops after drawing one block.
Drawing is done every block each time **[START]** key is depressed hereafter.

5 Depress **[STOP]** key
The system enters the feed hold state.
Pressing the **[STOP]** key during continuous drawing changes the operation to single-block drawing, causing the system to enter the feed hold state.
To perform continuous drawing again, press the **[SEQ]** key, then press the **[START]** key.

(4) Relation between drawing mode and machining operation mode
The drawing mode means a condition which is not the drawing end condition (that is, drawing in progress or single block drawing stop condition), while the machining operation mode means the drawing end condition.
The relation between the drawing mode and the machining operation mode is illustrated below.

<u>In case of (1)</u>;

When **[START]** key is depressed in the MEMORY mode on the drawing screen, the mode is swtiched to the drawing mode, and then, drawing is started.



<u>In case of (2)</u>;

Drawing terminates and the drawing mode is switched to the machining operation mode in the following cases.

- Reset button or external reset button was depressed.
- M02 or M30 was commanded on an NC program.

Since drawing is done under such a condition as MACHINE LOCK, the modal information, absolute coordinate value, etc. are updated.

When the mode is switched from the machining operation mode to the drawing mode, the following information is stored.

(1) Relative coordinate value (RELATIVE)

(2) Work coordinate system value (ABSOLUTE)

(3) Machine coordiante system value (MACHINE)

(4) T code

(5) Tool position offset

After drawing, the stored information is restored as before, and the mode is switched to the machining operation mode after heading of the drawn program.

NOTE

1 None of the execution of dwell (G04), sending of M, S, T codes, and sending of the press start signal is done.

2 Output signals OP, STL, SPL, AL are sent to the machine tool even during drawing in the same way as in normal machining operation.

3 Input signals ERS, *SP, *ESP, ST, etc. of the machine tool are efective even during drawing. Don't operate these switches, accordingly.
(The single block switch SBK is ineffective.)

4 If manual reference point rerturn function is provided, the stored stroke limit 1 is checked even during drawing. Thus, the machine tool position must coincide with actual machining start position before drawing start.

5 The tool profile is drawn at the end point in a block where positioning is done even in the punch off mode using M function or under the turned-on condition of the punch off switch on the machine tool side.

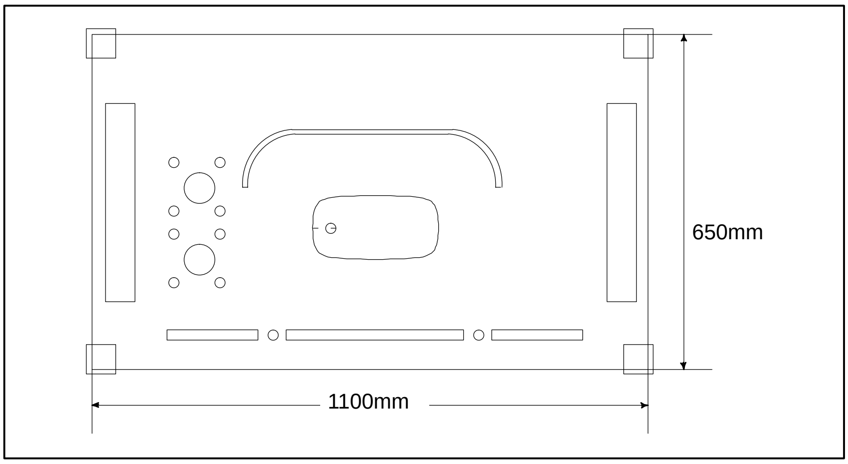
6 If the tool profile code of the designated T code is 00, mark (*) is drawn.

7 Machining operation is not executed in the drawing mode. Drawing is not attempted halfway during machining operation.

8 It is impossible to draw a helically interpolated figure.

12.5 EXAMPLE

Set drawing parameters for drawing the following NC program as follows.



: 00002 ; (NC program)

```
G92X1270. Y1270.  ;
G90G00X15. Y15.  T02  ;
X1085.  Y15.  ;
X1085.  Y635.  ;
X15.  Y635.  ;
G72X150.  Y80.  ;
G76I40.  JOK4T03  ;
G72X400.  Y80.  ;
G76I40.  JOK7  ;
G72X770.  Y80.  ;
G76I40.  JOK4  ;
G72X50.  Y200.  ;     ↗
G76I40. J90. K8T02  ;
G72X1050.  Y200.  ;
G76I40.  J90.  K8  ;
G90X375.  Y80.  T04  ;
X745.  Y80.  ;
X230.  Y250.  T05  ;
G26I50.  J45.  K4T06  ;
X230.  Y400.  T05  ;
G26I50.  J45.  K4T06  ;
G72X500.  Y400  ;
G77I150. J180. P-5. K18T04  ;
G76I15.  JOK20  ;     ↗
G72X800. Y400.  ;
G77I150.  J90.  P-5K18  ;
G90X550.  Y325.  T04  ;
M24  ;
G01X500.  Y325.  F3000  ;
X500.  Y300.  ;
G03X550.  Y250.  I50.  ;
G01X750.  Y250.  ;
G03X800.  Y300.  J50.  ;
G01X800.  Y350.  ;
G03X750.  Y400.  I-50.  ;
G01X550.  Y400.  ;
G03X500.  Y350.  J-50.  ;
G01X500.  Y325.  ;
M25  ;
G00G90X1270.  Y1270.  M02  ;
```

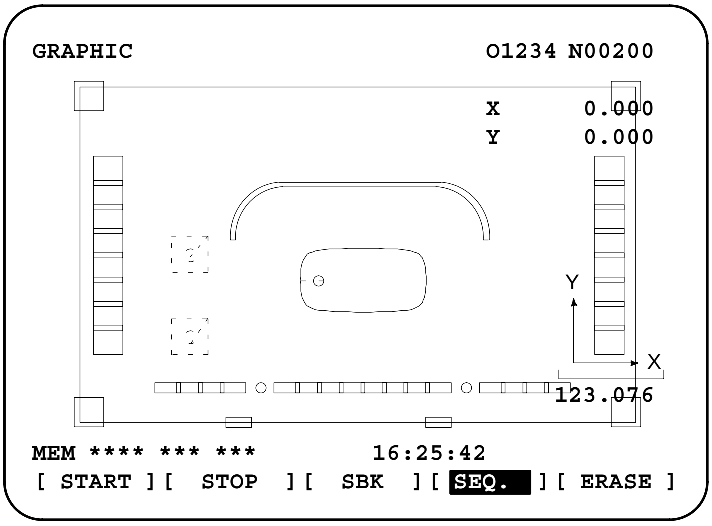
```
GRAPHIC PARAMETER                        O1234 N00200
 AXES   ( 0,1,2,3,4 )                    P=          0

 RANGE              (MAX.)
  X=   1150000  Y=       700000  Z=              0
 RANGE              (MIN.)
  X=   -100000  Y=      -100000  Z=              0
 SCALE          K=           10
 GRAPHIC CENTER
  X=    525000  Y=       300000  Z=              0
 PROGRAM STOP   N=            0
 AUTO ERASE     A=            1

>_
MEM **** *** ***           16:19:18
[ PARAM ] [ GRAPH ] [        ] [        ] [        ]
```

```
GRAPHIC PARAMETER                         O1234 N00200
 START POINT
  X=         0  Y=            0  Z=         0
 WORK LENGTH
  X=   1100000  Y=       650000
 RAPID PATH
 (1:ON 0:OFF)   P=            0
 HOLDER POSITION       HOLDER LENGTH
 X1=    300000  X=        40000  Y=     20000
 X2=    700000  X=        40000  Y=     20000



>_
MEM **** *** ***         16:23:21
[ PARAM ][ GRAPH ][        ][        ][        ]
```

Tool figure data

T02 : Rectangle, X dimension = 50 mm, Y dimension = 50 mm
T03 : Rectangle, X dimension = 50 mm, Y dimension = 20 mm
T04 : Circle, diameter = 15.5 mm
T05 : Circle, diameter = 30 mm
T06 : Circle, diameter = 5 mm

The following figures show the programmed figure.



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

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
[ ALM ]    [ OPR ]    [ PARA ]    [          ]  [          ]
```

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 **[ALM]** 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  : TOOL LENGTH COMPENSATION C
ALARM  :
    IN TOOL LENGTH COMPENSATION TYPE C,
    NO AXIS IS DESIGNATED IN G43 & G44
    BLOCKS. IN TOOL LENGTH COMPENSATION
    TYPE C, IT TRIES TO LATCH ON TO
    ANOTHER AXIS WITHOUT OFFSET CANCE–
    LING.

>100                                       S    0 T0000
MEM **** *** ***             10:12:25
[ ALM ][  OPR  ]  [ PARA ]  [          ]  [          ]
```

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
[ A LM  ] [  OPR  ]  [ PARA ]  [        ]  [        ]
```

Fig. 13 (c) ALARM DETAIL Screen when No Alarm is issued

3 To get details on another alarm number, first enter the alarm number, then press soft key **[SELECT]**. This operation is useful for investigating alarms not currently being raised.

```
>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 **[OPR]** 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
( ALM     )( OPR    )( PARA  )(        )( (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.

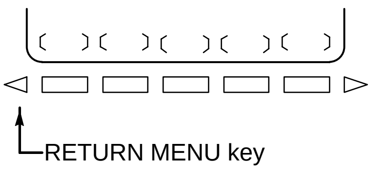
```
>1                                          S    0 T0000
MEM  **** *** ***                   10:12:25
[        ] [        ]  [         ] [        ]  [ SELECT ]
```

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
[   ALM  ] [  OPR  ] [ PARA ] [        ] [        ]
```

Fig. 13 (h) Selected OPERATION METHOD screen



5 To return to the OPERATION METHOD menu screen, press the RETURN MENU key to display “**[OPR]**” again, and then press the **[OPR]** 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 **[PARA]** key on the HELP (INITIAL MENU) screen. A list of parameter Nos. for each function is displayed. (See Fig 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.

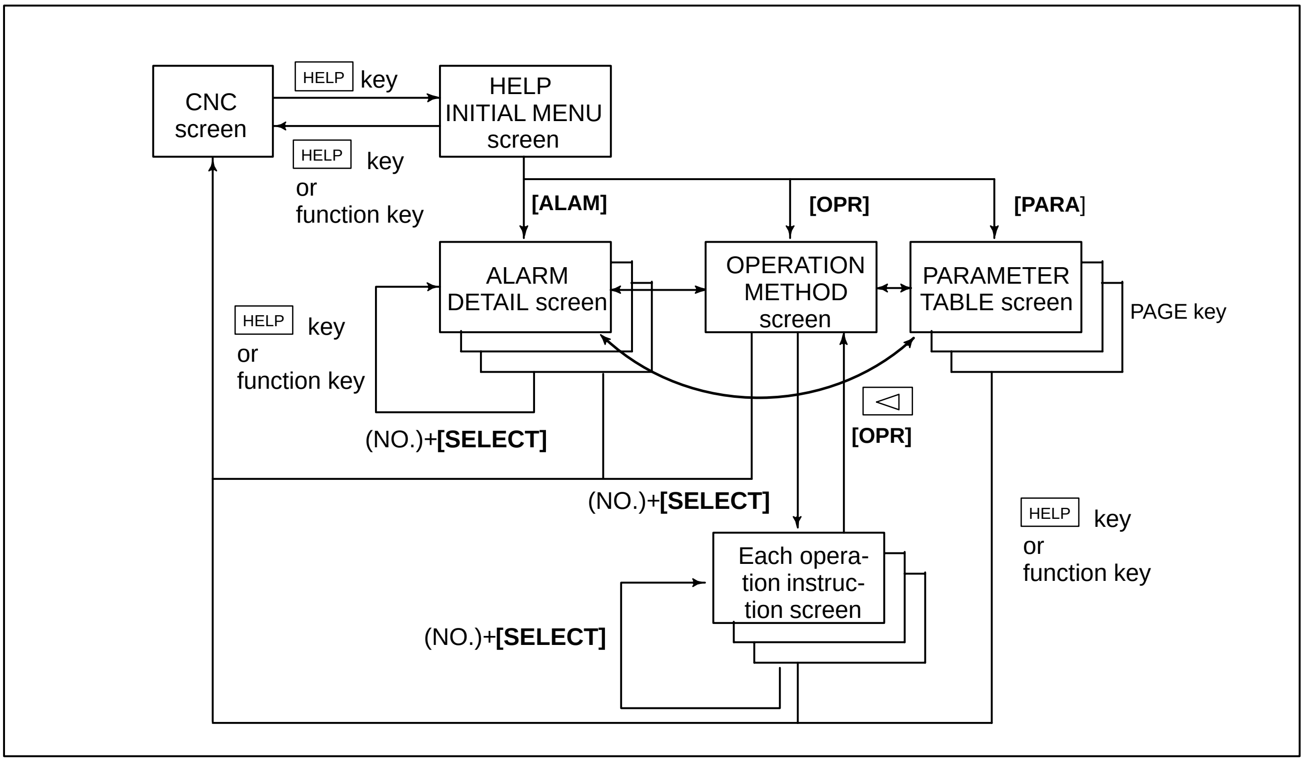
```
HELP (PARAMETER TABLE)                     01234  N00001
                                                     1/4

* 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
 [  ALM  ] [  OPR  ] [ PARA ] [        ] [          ]
```

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



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:

1.1 REPLACING THE BATTERY FOR CONTROL UNIT

1.2 BATTERY FOR THE ABSOLUTE PULSE CODER

1.3 BATTERY FOR SEPARATE ABSOLUTE PULSE CODERS (6 VDC)

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 THE BATTERY FOR CONTROL UNIT

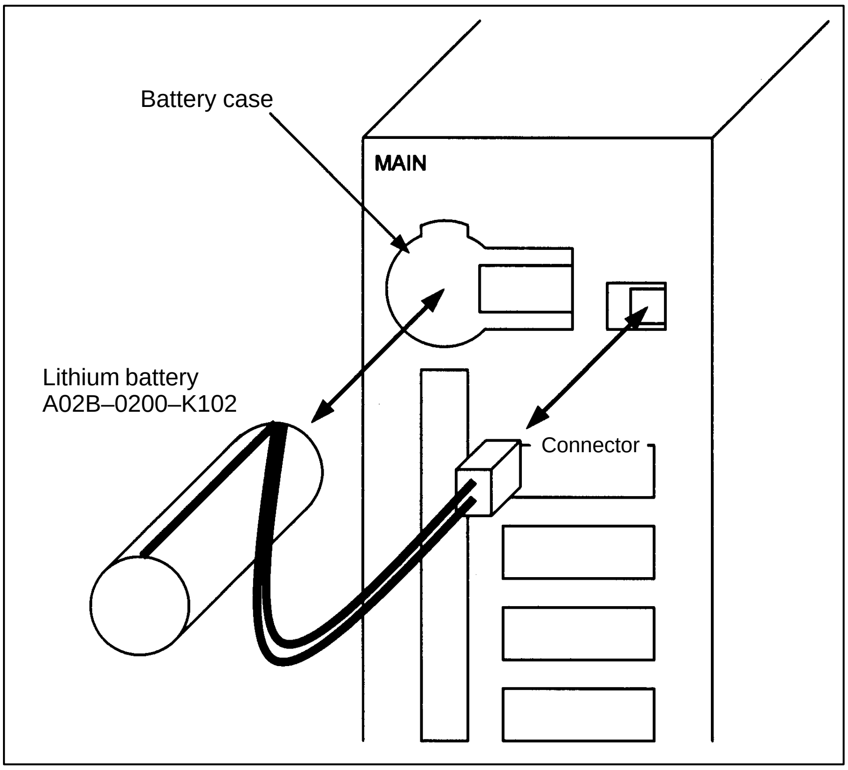
- **Replacing the battery**

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.

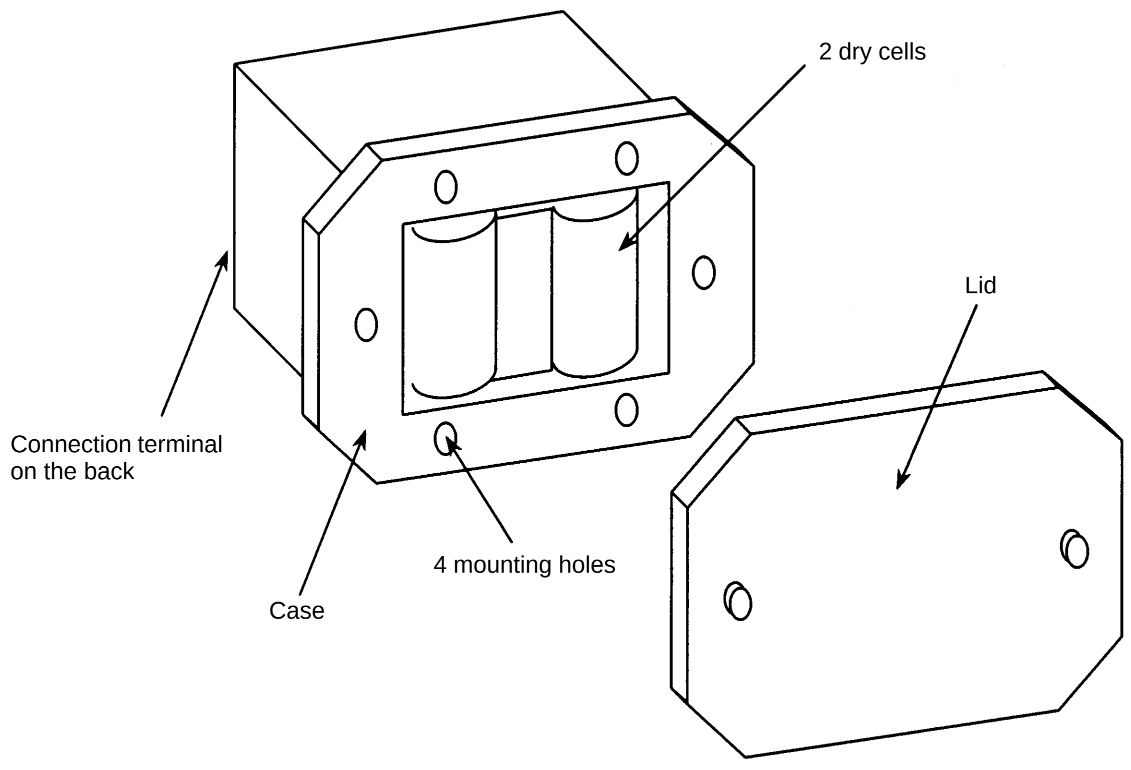
(2) Turn the CNC on.

(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.

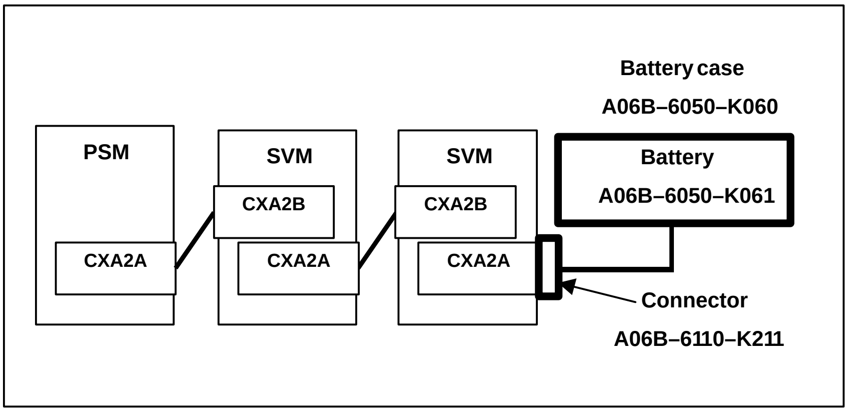
> **NOTE**
> In the power–off state, the battery should be replaced as in the case of the lithium battery, which is descried above.



1.2 BATTERY FOR THE ABSOLUTE PULSE CODER

The battery unit for the absolute pulse coder can be connected using [Connection scheme 1] and [Connection scheme 2] explained below.

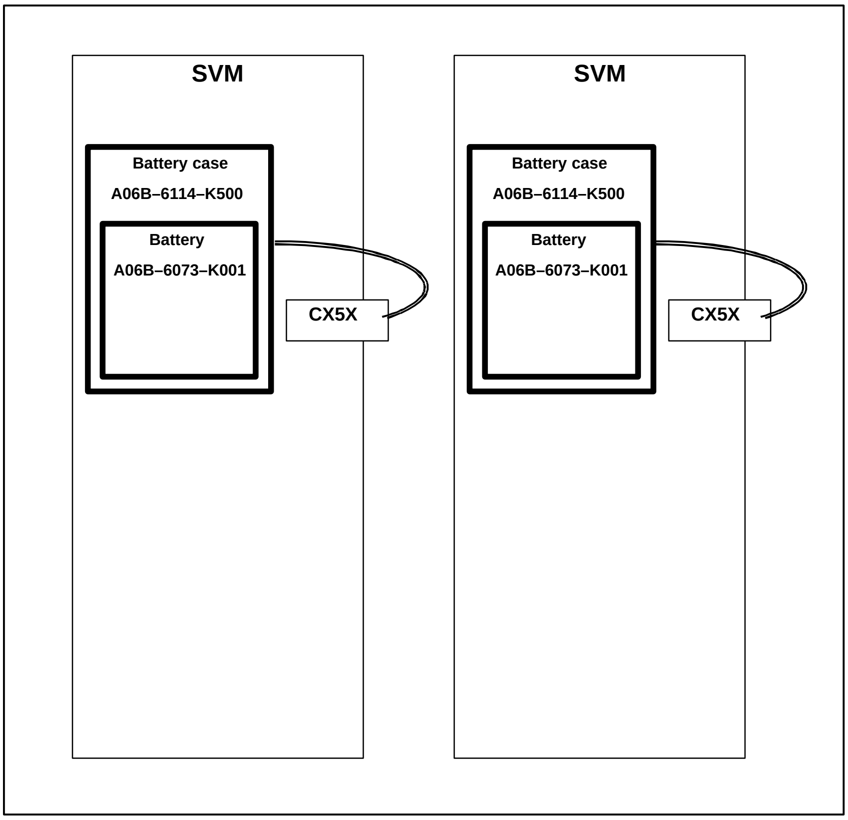
[Connection scheme 1] Supplying power from one battery unit to more than one SVM



– 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 α*i* 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 a 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 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 α*i* 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 α*i* 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.

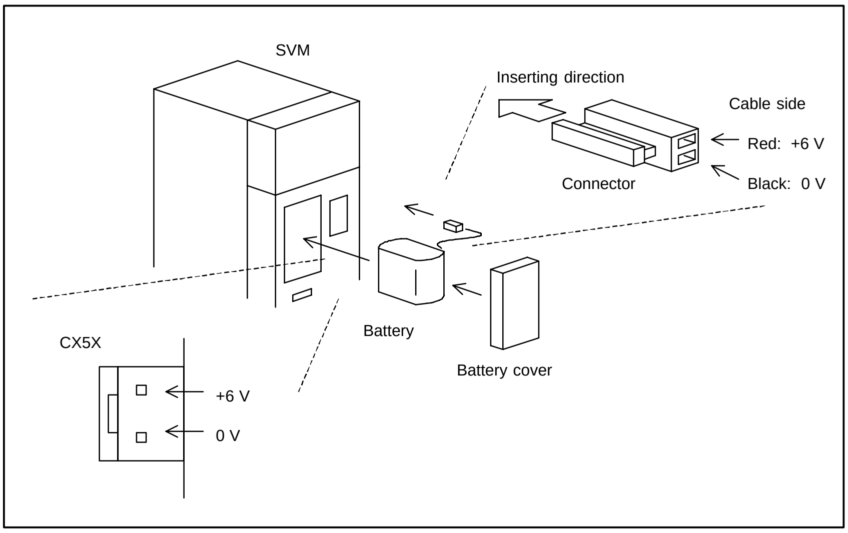
[Installation procedure for the battery]

(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.

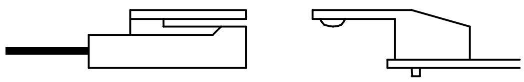
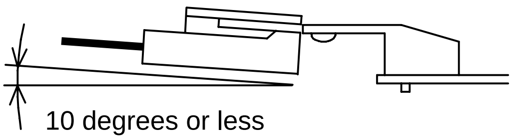
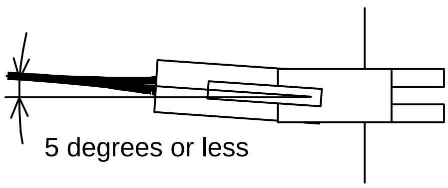
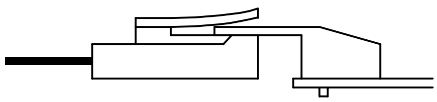
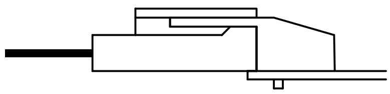
3 Install the battery with correct polarity. If the battery is installed with incorrect polarity, it may overheat, blow out, or catch fire.

4 Avoid a short–circuit between the +6 V and 0 V lines of a battery or cable. A short–circuit may lead to a hot battery, an explosion, or fire.

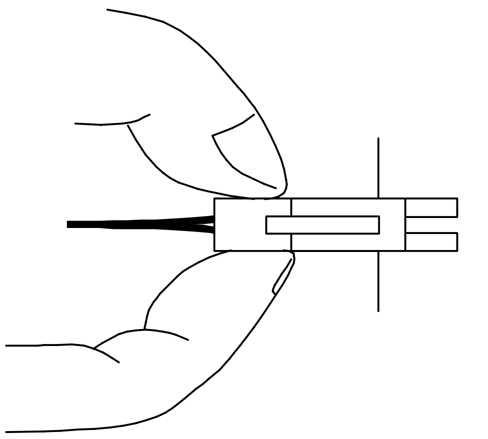
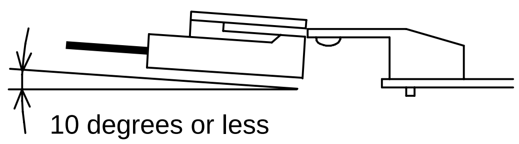
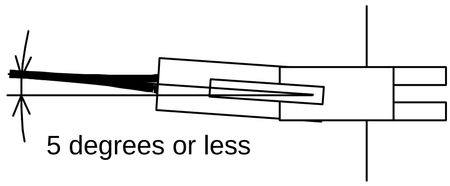
[Caution No. 2 for battery replacement]

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.

1.3 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 3n7 is displayed, replace the battery as soon as possible. In general, the battery should be replaced within two or three weeks, however, this depends on the number of pulse coders used.
If the voltage of the battery becomes any lower, the 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.
Therefore, FANUC recommends that the battery be replaced every year regardless of the occurrence of an APC alarm.

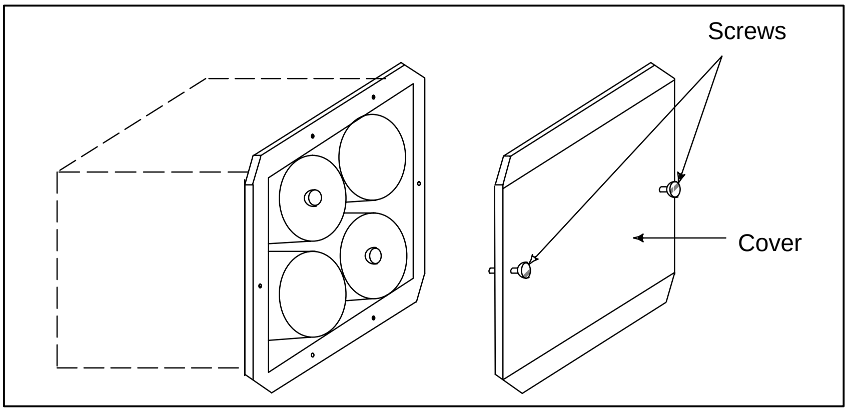
Replacing batteries

Obtain four commercially available alkaline batteries (size D).

(1) Turn on the power to the machine (Series 0*i*).

(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 (Series 0*i*).

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 Series 0*i* is on. Note that, if batteries are replaced while no power is supplied to the CNC, the recorded absolute position is lost.

SERVO AMPLIFIER β series

The battery is connected in either of 2 ways as follows.

Method 1: Attach the lithium battery to the SVM.
Use the battery: A06B–6093–K001.
Method 2: Use the battery case (A06B–6050–K060).
Use the battery: A06B–6050–K061 or D–size alkaline battery.

Method	Item	Ordering specification
Method 1	Battery (Lithium battery)	A06B–6093–K001
Method 2	Battery (4 pieces of D–size alkaline battery)	A06B–6050–K061

- Attach the lithium battery to the amplifier. (Method 1)
  Attach the lithium (A06B–6093–K001) battery to the amplifier.

  [Attachment procedure]

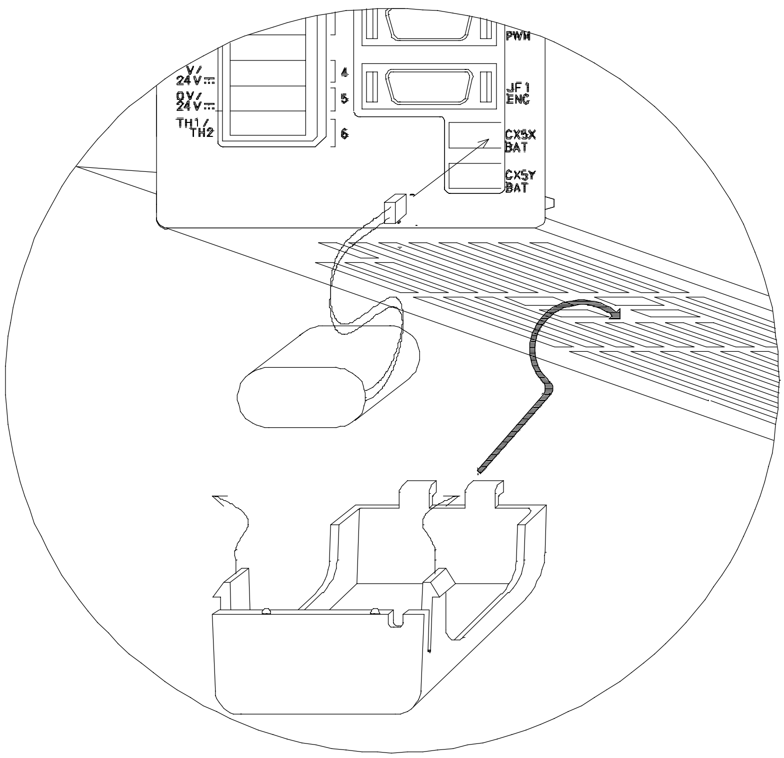
  (1) Check the item 1 to 3 of "Replacement procedure".

  (2) 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.

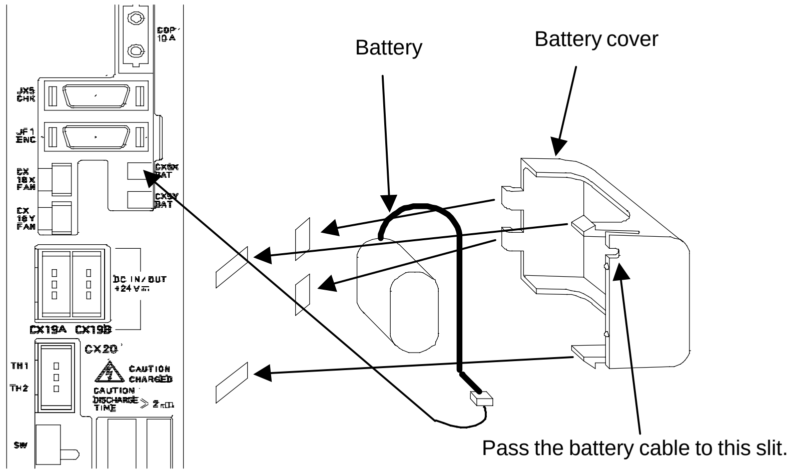
  (3) Remove the battery from the servo unit.

  (4) Replace the battery and connect the battery cable with the connector CX5X or CX5Y of the servo unit.

  (5) Mount the battery cover.



SVU–12, SVU–20



SVU–40, SVU–80

CAUTIONS
- The connector of the battery can be connected with either of CX5X and CX5Y.

- Replacement of batteries in the battery case. (Method 2)
  Replace four D–size alkaline batteries in the battery case installed in the machine.

  [Attachment procedure]

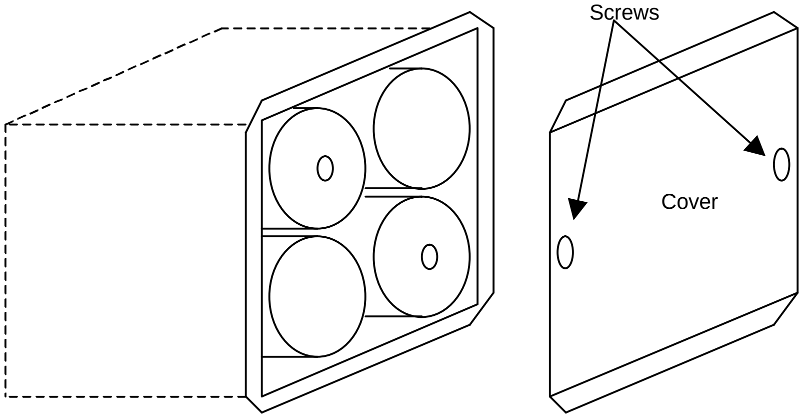
  (1) Check the item 1 to 3 of ”Replacement procedure”.

  (2) Have four D–size alkaline batteries on hand.

  (3) Loosen the screws on the battery case. Remove the cover.

  (4) Replace the alkaline batteries in the case. Pay careful attention to the polarity of the alkaline batteries.

  (5) Attach the cover.



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		

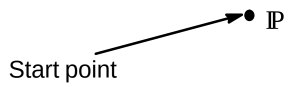
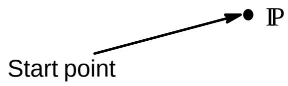
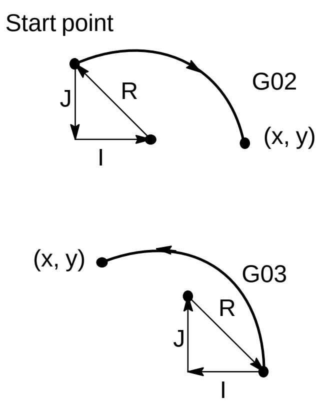
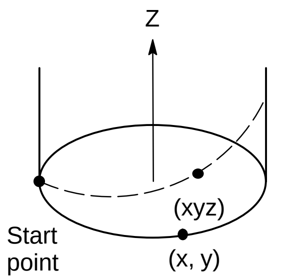
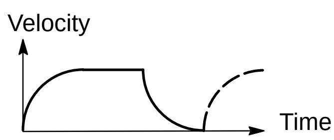
ISO code										EIA code										Meaning		
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Without CUSTOM MACRO B	With CUSTOM MACRO B
DEL	○	○	○	○	○	○	○	○	○	Del		○	○	○	○	○	○	○	○	Delete (deleting a mispunch)	×	×
NUL						○				Blank						○				No punch. With EIA code, this code cannot be used in 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	

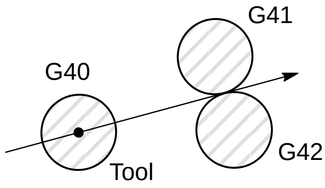
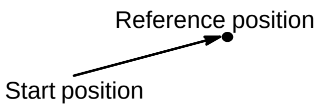
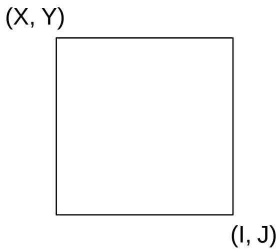
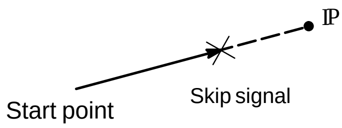
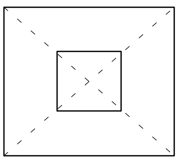
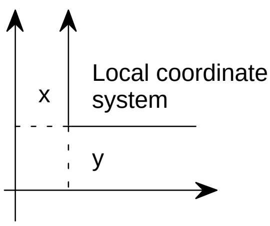
NOTE

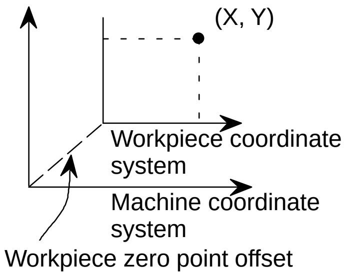
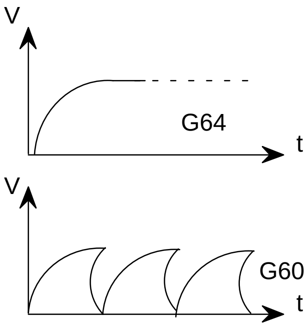
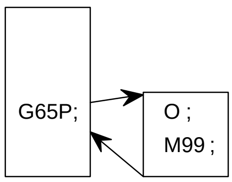
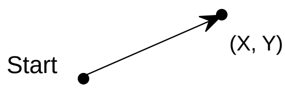
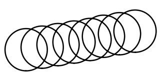
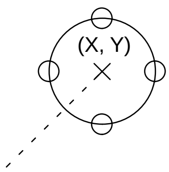
1 The symbols used in the remark column have the following meanings.
   (Space) : The character will be registered in memory and has a specific meaning. It it is used incorrectly in a statement other than a comment, an alarm occurs.
   × : The character will not be registered in memory and will be ignored.
   Δ : The character will be registered in memory, but will be ignored during program execution.
   ○ : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.
   □ : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.
2 Codes not in this table are ignored if their parity is correct.
3 Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.
4 A character with all eight holes punched is ignored and does not generate TH alarm in EIA code.

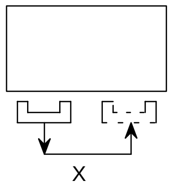
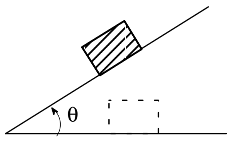
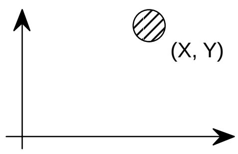
B LIST OF FUNCTIONS AND TAPE FORMAT

Some functions cannot be added as options depending on the model.

Functions	Illustration	Tape format
Positioning (G00)		G00X_Y_C_ ;
Linear interpolation (G01)		G01X_Y_F_ ;
Circular interpolation (G02, G03)		$\begin{Bmatrix}G02\\G03\end{Bmatrix}X\_\ Y\_\begin{Bmatrix}R\_\\I\_\ J\_\end{Bmatrix}F\_\ ;$
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) (In case of X–Y plane)		$G04\begin{Bmatrix}X\_\\P\_\end{Bmatrix}\ ;$
Exact stop (G09)		$G09\begin{Bmatrix}G01X\_Y\_\\G02\_\\G03\end{Bmatrix}\ ;$
Change of offset value by program (G10)		G10 P_R_;

Functions	Illustration	Tape format
Cutter compensation C (G40 – G42)		G40 G41 X_Y_D_ ; G42 D : Tool offset
Normal–line direction control (G40.1, G41.1, G42.1) (G150, G151, G152)		G41.1 (G151) Left–side normal–line direction control G42.1 (G152) Right–side normal–line direction control G40.1 (G150) Cancel normal–line direction control
Inch/millimeter conversion (G20, G21)		G20 ; Inch input G21 ; Millimeter input
Reference position return check (G28)		G28 ;
Stored stroke check (G22, G23)		G22X_Y_I_J_ ; G23 ; Cancel
Skip function (G33)		G33IP_F_ ;
Scaling (G50, G51)		G51X_Y_P_ ; G50 ; Cancel
Local coordinate system setting (G52)		G52X_Y_ ;
Command in machine coordinate system (G53)		G53IP_ ;

Functions	Illustration	Tape format
Setting in workpiece coordinate sytem		$\left\{\begin{matrix}G54\\ :\\ G59\end{matrix}\right\}$ X_Y_ ;
Pattern function (G26, G76, G77, G78, G79, G86, G87, G88, G89)	Refer to “Pattern Function”	G26 ___ ; G76 ___ ; G77 ___ ; G78 ___ ; G79 ___ ; G86 ___ ; G87 ___ ; G88 ___ ; G89 ___ ;
Cutting and exact stop mode (G64–G61)		G64 ___ ; Cutting mode G61 ___ ; Exact stop mode
Custom macro (G65, G66, G67)		Macro simple cll G65P_(Argument specification) P: Program No. Modal call G66P_(Argument specification) G67:...Cancel
Postioning & Press off (G70)		G70X_Y_C_ ;
Nibbling function (G68, G69, M12–M13)		G68I_J_K_P_Q_ ; G69I_J_P_Q_ ; M12 ; to M13 ;
Pattern base point command (G72)		G72X_Y_ ;

Functions	Illustration	Tape format
Automatic repositioning (G75)		G75X_ ;
Multi–piece machining function (G73, G74, G98)	Refer to “Multi–piece machining”.	G73 } W_Q_ ; G74 } W:Macro number G98X_Y_I_P_J_K_ ;
Coordinate rotation (G84, G85)		G84X_Y_R_ ; G85;Cancel
Absolute/Incremental Command (G90/G91)		G90 ____ ; Absolute command G91 ____ ; Incremental command
Change of work coordinate system (G92)		G92X_Y_;

C RANGE OF COMMAND VALUE

Linear axis

• In case of millimeter input, feed screw is millimeter

	Increment system	
	IS–A	IS–B
Least input increment	0.01 mm	0.001 mm
Least command increment	0.01 mm	0.001 mm
Max. programmable dimension	±999999.99 mm	±99999.999 mm
Max. rapid traverse **Note**	240000 mm/min	240000 mm/min
Feedrate range **Note**	1 to 240000 mm/min	1 to 240000 mm/min
Incremental feed	0.01, 0.1, 1, 10 mm/step	0.001, 0.01, 0.1, 1 mm/step
Tool compensation	0 to ±999.99 mm	0 to ±999.999 mm
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec

• In case of inch input, feed screw is millimeter

	Increment system	
	IS–A	IS–B
Least input increment	0.001 inch	0.0001 inch
Least command increment	0.01 mm	0.001 mm
Max. programmable dimension	±99999.999 inch	±9999.9999 inch
Max. rapid traverse **Note**	240000 mm/min	240000 mm/min
Feedrate range **Note**	0.01 to 9600 inch/min	0.01 to 9600 inch/min
Incremental feed	0.001, 0.01, 0.1, 1 inch/step	0.0001, 0.001, 0.01, 0.1 inch/step
Tool compensation	0 to ±99.999 inch	0 to ±99.9999 inch
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec

- **In case of inch input, feed screw is inch**

	Increment system	
	IS–A	**IS–B**
Least input increment	0.001 inch	0.0001 inch
Least command increment	0.001 inch	0.0001 inch
Max. programmable dimension	±99999.999 inch	±9999.9999 inch
Max. rapid traverse **Note**	9600 inch/min	9600 inch/min
Feedrate range **Note**	0.01 to 9600 inch/min	0.01 to 9600 inch/min
Incremental feed	0.001, 0.01, 0.1, 1 inch/step	0.0001, 0.001, 0.01, 0.1 inch/step
Tool compensation	0 to ±99.999 inch	0 to ±99.9999 inch
Dwell time	0 to 99999.999 sec	0 to 99999.999 sec

- **In case of millimeter input, feed screw is inch**

	Increment system	
	IS–A	**IS–B**
Least input increment	0.01 mm	0.001 mm
Least command increment	0.001 inch	0.0001 inch
Max. programmable dimension	±999999.99 mm	±99999.999 mm
Max. rapid traverse **Note**	9600 inch/min	9600 inch/min
Feedrate range **Note**	1 to 240000 mm/min	1 to 240000 mm/min
Incremental feed	0.01, 0.1, 1, 10 mm/step	0.001, 0.01, 0.1, 1 mm/step
Tool compensation	0 to ±999.99 mm	0 to ±999.999 mm
Dwell time	0 to 99999.999 sec	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.

D NOMOGRAPHS

D.1 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. D.1 (a).
Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

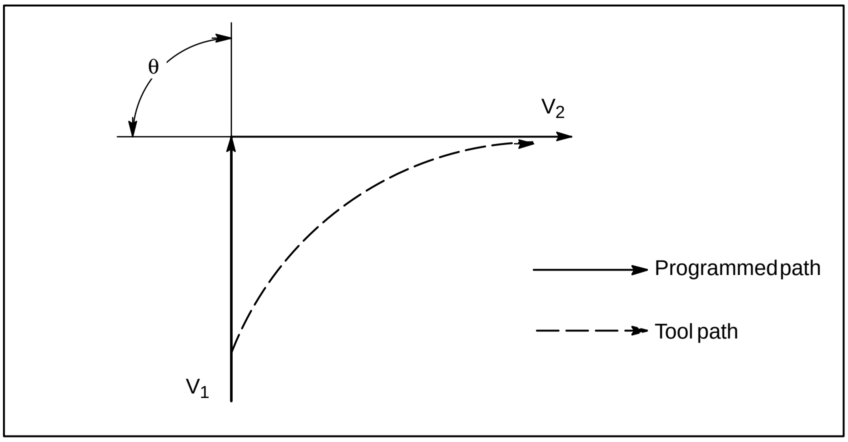


Fig. D.1 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:

- **Feedrate (V_1, V_2)**
- **Corner angle (θ)**
- **Exponential acceleration / deceleration time constant (T_1) at cutting (T_1 = 0)**
- **Presence or absence of buffer register.**

The above parameters are used to theoretically analyze the tool path and above tool path is drawn with the parameter which is set as an example.
When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.
In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

Analysis

The tool path shown in Fig. D.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.)

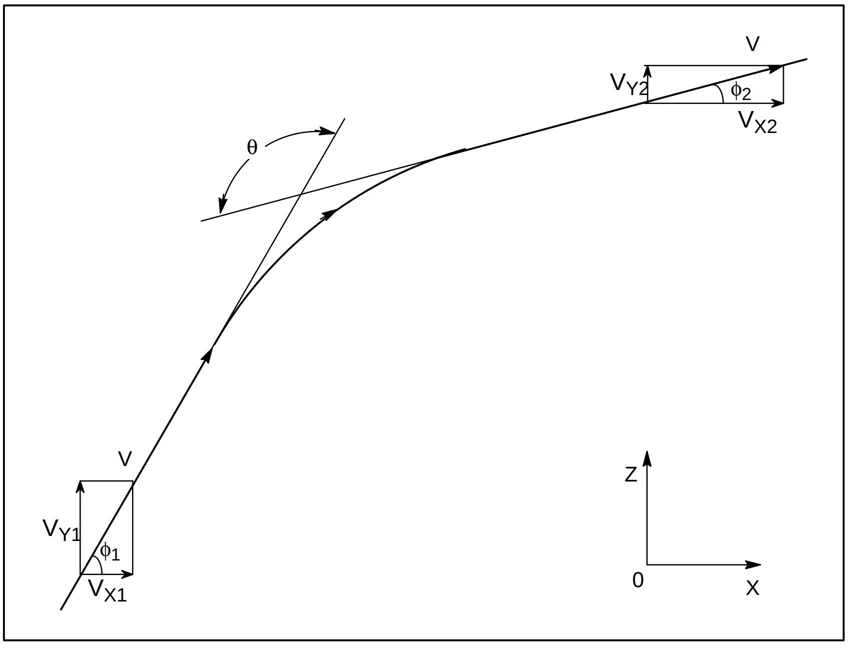


Fig. D.1 (b) Example of tool path

- **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

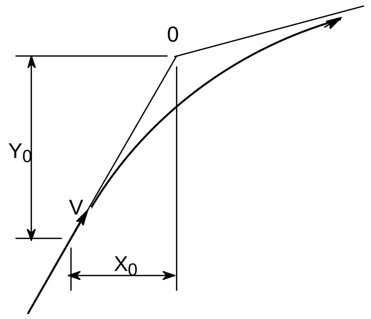


Fig. D.1 (c) Initial value

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:

$$X(t) = \int_0^t V_X(t)dt-X_0$$

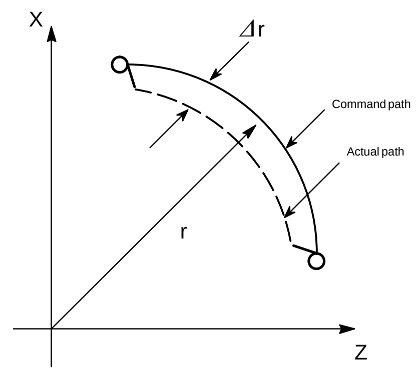
$$= \frac{V_{X2}-V_{X1}}{T_1-T_2}\{T_1{}^2\exp(-\frac{t}{T_1})-T_2{}^2\exp(-\frac{t}{T_2})\}-V_{X2}(T_1 + T_2-t)$$

$$Y(t) = \int_0^t V_Y(t)dt-Y_0$$

$$= \frac{V_{Y2}-V_{Y1}}{T_1-T_2}\{T_1{}^2\exp(-\frac{t}{T_1})-T_2{}^2\exp(-\frac{t}{T_2})\}-V_{Y2}(T_1 + T_2-t)$$

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 tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, sepecially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}\left(T_1{}^2 + T_2{}^2(1-\alpha^2)\right)\frac{V^2}{r} \quad \cdots\cdots \quad (1)$$

- 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)
- α : Feed forward coefficient (%)

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:

Linear acceleration/deceleration after cutting feed interpolation

$$\Delta r = \left(\frac{1}{24}T_1{}^2 + \frac{1}{2}T_2{}^2(1-\alpha^2)\right)\frac{V^2}{r}$$

Bell–shaped acceleration/deceleration after cutting feed interpolation

$$\Delta r = \left(\frac{1}{48}T_1{}^2 + \frac{1}{2}T_2{}^2(1-\alpha^2)\right)\frac{V^2}{r}$$

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 v (mm /sec) is determined by equation (1).
Since the acceleration/deceleration time constant at cutting which is set by this equipment varies with the machine tool, refer to the manual issued by the machine tool builder.

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

Parameter 3402 (CLR) is used to select whether resetting the CNC places it in the cleared state or in the reset state (0: reset state/1: cleared state).
The symbols in the tables below mean the following :
○:The status is not changed or the movement is continued.
×:The status is cancelled or the movement is interrupted.

Item		When turning power on	Cleared	Reset
Setting data	Offset value	○	○	○
	Data set by the MDI setting operation	○	○	○
	Parameter	○	○	○
Various data	Programs in memory	○	○	○
	Contents in the buffer storage	×	×	○ : MDI mode × : Other mode
	Display of sequence number	○	○ (Note 1)	○ (Note 1)
	One shot G code	×	×	×
	Modal G code	Initial G codes. (The G20 and G21 codes return to the same state they were in when the power was last turned off.)	Initial G codes. (G20/G21 are not changed.)	○
	F	Zero	Zero	○
	S, T, M	×	○	○
	K (Number of repeats)	×	×	×
Work coordinate value		Zero	○	○
Action in opera-tion	Movement	×	×	×
	Dwell	×	×	×
	Issuance of M, S and T codes	×	×	×
	Cutter compensation	×	×	○ : MDI mode × : Other modes
	Storing called subpro-gram number	×	× (Note 2)	○ : MDI mode × : Other modes (Note 2)

Item		When turning power on	Cleared	Reset
Output signals	CNC alarm signal AL	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm
	Reference position return completion LED	×	○ (× : Emergency stop)	○ (× : Emergency stop)
	S, T and B codes	×	○	○
	M code	×	×	×
	M, S and T strobe signals	×	×	×
	Spindle revolution signal (S analog signal)	×	○	○
	CNC ready signal MA	ON	○	○
	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 by heading function. Execution cannot be started from the middle of the subprogram.

F CHARACTER–TO–CODES CORRESPONDENCE TABLE

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		”	034	Quotation mark
H	072		#	035	Hash sign
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		’	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		–	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	HAtI mark
1	049		[	091	Left square bracket
2	050		^	092	
3	051		]	094	Right square bracket
4	052		_	095	Underscore
5	053				

G ALARM LIST

1) Program errors (P/S alarm)

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign " – " was input without an address at the beginning of a block. Modify the program .
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign " ." input error (Sign " – " was input after an address with which it cannot be used. Or two or more " – " signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point " – " input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G–CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
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 CIRCLE RADIUS	The command for circular interpolation lacks arc radius R or coordinate I, J, or K of the distance between the start point to the center of the arc.
028	ILLEGAL PLANE SELECT	In the plane selection command, two or more axes in the same direction are commanded. Modify the program.
030	ILLEGAL OFFSET NUMBER	The offset number specified by D/H code for tool length offset or cutter compensation is too large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.

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	G40 is commanded on the plane other than offset plane in cutter compensation B. 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.
059	PROGRAM NUMBER NOT FOUND	In an external program number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background eiting.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63 (basic), 125, 200, 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.
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.

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 off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.
090	REFERENCE RETURN INCOMPLETE	The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position, or specify a sufficiently fast speed for reference position return.
091	REFERENCE RETURN INCOMPLETE	Manual reference position return cannot be performed when automatic operation is halted.
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.
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.

Number	Message	Contents
122	DUPLICATE MACRO MODAL–CALL	The macro modal call is specified fourfold. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, 1≦ n ≦3 is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation > is used. Modify the program.
130	ILLEGAL AXIS OPERATION	An axis control command was given by PMC to an axis controlled by CNC. Or an axis control command was given by CNC to an axis controlled by PMC. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
138	SUPERIMPOSED DATA OVERFLOW	The total distribution amount of the CNC and PMC is too large during superimposed control of the extended functions for PMC axis control.
139	CAN NOT CHANGE PMC CONTROL AXIS	An axis is selected in commanding by PMC axis control. Modify the program.
141	CAN NOT COMMAND G51 IN CRC	G51 (Scaling ON) is commanded in the tool offset mode. Modify the program.
142	ILLEGAL SCALE RATE	Scaling magnification is commanded in other than 1 – 999999. Correct the scaling magnification setting (G51 Pp .. or parameter 5411 or 5421).
143	SCALED MOTION DATA OVERFLOW	The scaling results, move distance, coordinate value and circular radius exceed the maximum command value. Correct the program or scaling mangification.
144	ILLEGAL PLANE SELECTED	The coordinate rotation plane and arc or cutter compensation C plane must be the same. Modify the program.
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)
179	PARAM. (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 COMMAND M198/M199	M198 and M199 are executed in the schedule operation. Or M198 is executed in the DNC operation.

Number	Message	Contents
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 tool compensation of the shift type is executed in the synchronous control. Correct the program.
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 bit 0 of parameter 1005 ZRN_X is 0.
231	ILLEGAL FORMAT IN G10 OR L50	Any of the following errors occurred in the specified format at the programmable–parameter input. 1) Address N or R was not entered. 2) A number not specified for a parameter was entered. 3) The axis number was too large. 4) An axis number was not specified in the axis–type parameter. 5) An axis number was specified in the parameter which is not an axis type. 6) An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PSSWD) when they are protected by a password. Correct the program.
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.
4500	REPOSITIONING INHIBITED	A repositioning command was specified in the circular interpolation (G02, G03) mode.
4502	ILLEGAL COMMAND IN BOLT HOLE	In a bolt hole circle (G26) command, the radius (I) was set to zero or a negative value, or the number of holes (K) was set to zero. Alternatively, I, J, or K was not specified.
4503	ILLEGAL COMMAND IN LINE AT ANGLE	In a line-at-angle (G76) command, the number of holes (K) was set to zero or a negative value. Alternatively, I, J, or K was not specified.
4504	ILLEGAL COMMAND IN ARC	In an arc (G77) command, the radius (I) or the number of holes (K) was set to zero or a negative value. Alternatively, I, J, K, or P was not specified.
4505	ILLEGAL COMMAND IN GRID	In a grid (G78, G79) command, the number of holes (P, K) was set to zero or a negative value. Alternatively, I, J, K, or P was not specified.
4506	ILLEGAL COMMAND IN SHARE PROOFS	In a shear proof (G86) command, the tool size (P) was set to zero, or the blanking length (I) was 1.5 times larger than the tool size (P) or less. Alternatively, I, J, or P was not specified.
4507	ILLEGAL COMMAND IN SQUARE	In a square (G87) command, the tool size (P,Q) was set to zero or a negative value, or the blanking length (I, J) was three times larger than the tool size (P, Q) or less. Alternatively, I, J, P, or Q was not specified.

Number	Message	Contents
4508	ILLEGAL COMMAND IN RADIUS	In a radius (G88) command, the traveling pitch (Q) or radius (I) was set to zero or a negative value, or the traveling pitch (Q) was greater than or equal to the arc length. Alternatively, I, J, K, P, or Q was not specified.
4509	ILLEGAL COMMAND IN CUT AT ANGLE	In a cut-at-angle (G89) command, the traveling pitch (Q) was set to zero, negative value, or another value larger than or equal to the length (I). Alternatively, I, J, P, or Q was not specified.
4520	T, M INHIBITED IN NIBBLING-MODE	T code, M code, G04, G70 or G75 was specified in the nibbling mode.
4521	EXCESS NIBBLING MOVEMENT (X, Y)	In the nibbling mode, the X-axis or Y-axis traveling distance was larger than or equal to the limit (No. 16188 to 16193).
4522	EXCESS NIBBLING MOVEMENT (C)	In the circular nibbling (G68) or usual nibbling mode, the C-axis traveling distance was larger than or equal to the limit (No. 16194).
4523	ILLEGAL COMMAND IN CIRCLE-NIBBL	In a circular nibbling (G68) command, the traveling pitch (Q) was set to zero, a negative value, or a value larger than or equal to the limit (No. 16186, 16187), or the radius (I) was set to zero or a negative value. Alternatively, I, J, K, P, or Q was not specified.
4524	ILLEGAL COMMAND IN LINE-NIBBL	In a linear nibbling (G69) command, the traveling pitch (Q) was set to zero, negative value, or a value larger than or equal to the limit (No. 16186, 16187). Alternatively, I, J, P, or Q was not specified.
4530	A/B MACRO NUMBER ERROR	The number for storing and calling by an A or B macro was set to a value beyond the range from 1 to 5.
4531	U/V MACRO FORMAT ERROR	An attempt was made to store a macro while storing another macro using a U or V macro. A V macro was specified although the processing to store a macro was not in progress. A U macro number and V macro number do not correspond with each other.
4532	IMPROPER U/V MACRO NUMBER	The number of an inhibited macro (number beyond the range from 01 to 99) was specified in a U or V macro command.
4533	U/V MACRO MEMORY OVERFLOW	An attempt was made to store too many macros with a U or V macro command.
4534	W MACRO NUMBER NOT FOUND	Macro number W specified in a U or V macro command is not stored.
4535	U/V MACRO NESTING ERROR	An attempt was made to call a macro which is defined three times or more using a U or V macro command. An attempt was made to store 15 or more macros in the storage area for macros of number 90 to 99.
4536	NO W, Q COMMAND IN MULTI-PIECE	W or Q was not specified in the command for taking multiple workpieces (G73, G74).
4537	ILLEGAL Q VALUE IN MULTI-PIECE	In the command for taking multiple workpieces (G73, G74), Q is set to a value beyond the range from 1 to 4.
4538	W NO. NOT FOUND IN MULTI-PIECE	Macro number W specified in the command for taking multiple workpieces (G73, G74) is not stored.
4539	MULTI-PIECE SETTING IS ZERO	The command for taking multiple workpieces (G73, G74) was specified although zero is specified for the function to take multiple workpieces (No. 16206 or signals MLP1 and MLP2 (PMC address G231, #0 and #1)).
4540	MULTI-PIECE COMMAND WITHIN MACRO	The command for taking multiple workpieces (G73, G74) was specified when a U or V macro was being stored.
4542	MULTI-PIECE COMMAND ERROR	Although G98P0 was specified, the G73 command was issued. Although G98K0 was specified, the G74 command was issued.

Number	Message	Contents
4543	MULTI-PIECE Q COMMAND ERROR	Although G98P0 was specified, the Q value for the G74 command was not 1 or 3. Although G98K0 was specified, the Q value for the G73 command was not 1 or 2.
4544	MULTI-PIECE RESTART ERROR	In the command for resuming taking multiple workpieces, the resume position (P) is set to a value beyond the range from 1 to total number of workpieces to be machined.
4600	T, C COMMAND IN INTERPOLATION	In the linear interpolation (G01) mode or circular interpolation (G02, G03) mode, a T command or C-axis command was specified.
4601	INHIBITED T, M COMMAND	In the block of G52, G72, G73, or G74, a T or M command was specified.
4602	ILLEGAL T-CODE	The specified T command is not cataloged on the tool register screen.
4606	A T COMMAND WAS ISSUED	A T command was issued during normal–line control.
4650	IMPROPER G-CODE IN OFFSET MODE	In the cutter compensation mode, an inhibited G code (pattern command, G73, G74, G75, etc.) was specified.
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)
5010	END OF RECORD	The end of record (%) was specified.
5011	PARAMETER ZERO(CUT MAX)	The maximum cutting feedrate (parameter No. 1422)is 0 in the HPCC mode.
5064	DIFFERRENT AXIS UNIT (IS–B, IS–C)	Circular interpolation has been specified on a plane consisting of axes having different increment systems.
5065	DIFFERENT AXIS UNIT (PMC AXIS)	Axes having different increment systems have been specified in the same DI/DO group for PMC axis control. Modify the setting of parameter No. 8010.
5073	NO DECIMAL POINT	No decimal point has been specified for an address requiring a decimal point.
5074	ADDRESS DUPLICATION ERROR	The same address has been specified two or more times in a single block. Alternatively, two or more G codes in the same group have been specified in a single block.
5082	DATA SERVER ERROR	This alarm is detailed on the data server message screen.
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.
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.
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.

Number	Message	Contents
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.
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.
5257	G41/G42 NOT ALLOWED IN MDI MODE	G41/G42 (cutter compensation C: M series, tool–nose radius compensation: T series) was specified in MDI mode. (Depending on the setting of bit 4 of parameter No. 5008)
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.

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.
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	Return to the origin has been attempted without first rotating the motor one or more times. Before returning to the origin, rotate the motor one or more times then turn off the power.

4) Serial pulse coder (SPC) alarms

No.	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.

No.	Message	Description
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**

	#7	#6	#5	#4	#3	#2	#1	#0
202		CSA	BLA	PHA	PCA	BZA	CKA	SPH

#6 (CSA) : Check sum alarm has occurred.
#5 (BLA) : Battery low alarm has occurred.
#4 (PHA) : Phase data trouble alarm has occurred.
#3 (PCA) : Speed count trouble alarm has occurred.
#2 (BZA) : Battery zero alarm has occurred.
#1 (CKA) : Clock alarm has occurred.
#0 (SPH) : Soft phase data trouble alarm has occurred.

	#7	#6	#5	#4	#3	#2	#1	#0
203	DTE	CRC	STB	PRM				

#7 (DTE) : Data error has occurred.
#6 (CRC) : CRC error has occurred.
#5 (STB) : Stop bit error has occurred.
#4 (PRM) : Parameter error alarm has occurred. In this case, a servo parameter error alarm (No. 417) is also output.

5) Servo alarms

Number	Message	Contents
400	SERVO ALARM: n–TH AXIS OVERLOAD	The n–th axis (axis 1 to 8) overload signal is on. Refer to diagnosis display No. 201 for details.
401	SERVO ALARM: n–TH AXIS VRDY OFF	The n–th axis (axis 1 to 8) servo amplifier READY signal (DRDY) went off.
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.

Number	Message	Contents
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 has occurred during easy synchronization control operation. 1) The position deviation of the synchronization axis has exceeded the value specified by the parameter (No. 8314).
409	SERVO ALARM: n AXIS TORQUE ALM	Abnormal servo motor load has been detected. Alternatively, abnormal spindle motor load has been detected in Cs mode.
410	SERVO ALARM: n–TH AXIS – EXCESS ERROR	One of the following failures has occurred. 1) The position deviation when n–th axis stops has exceeded the value specified by the parameter (No. 1829). 2) Under easy synchronization control, the maximum compensation during synchronization has exceeded the value specified by the parameter (No. 8325). This alarm is issued 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. Note) Limit value must be set to parameter No.1828 for each axis.
413	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis (axis 1 to 8) are beyond the range of -2^{31} to 2^{31}. 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.
416	SERVO ALARM: n–TH AXIS – DISCONNECTION	Position detection system fault in the n–th axis (axis 1 to 8) pulse coder (disconnection alarm). Refer to diagnosis display No. 201 for details.
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) con tains 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 parame ter No. 1023 (servo axisnumber).
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.

Number	Message	Contents
431	n AXIS : CNV. OVERLOAD	1) PSM: Overheat occurred. 2) β series SVU: Overheat occurred.
432	n AXIS : CNV. LOWVOLT CON./ POWFAULT	1) PSM: Phase missing occurred in the input voltage. 2) PSMR: 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) PSM: The DC link voltage has dropped. 2) PSMR: 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	SVM: The control power supply voltage has dropped.
435	n AXIS : INV. LOW VOLT DC LINK	SVM: 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	PSM: Overcurrent flowed into the input circuit.
438	n AXIS : INV. ABNORMAL CURRENT	1) SVM: 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) PSM: The DC link voltage is too high. 2) PSMR: 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) PSMR: 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.
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) PSM: The spare discharge circuit of the DC link is abnormal. 2) PSMR: 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) PSM: The internal stirring fan failed. 2) PSMR: The internal stirring fan failed. 3) β series SVU: The internal stirring fan failed.
444	n AXIS : INV. COOLING FAN FAILURE	SVM: 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) SVM: IPM (intelligent power module) detected an alarm. 2) α series SVU: IPM (intelligent power module) detected an alarm.

Number	Message	Contents
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.
465	n AXIS : READ ID DATA FAILED	At power–up, amplifier initial ID information could not be read.
466	n AXIS : MOTOR/AMP COMBINATION	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)

- **Details of servo alarm**

The details of servo alarm are displayed in the diagnosis display (No. 200, 201, and No.204) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
200	OVL	LV	OVC	HCA	HVA	DCA	FBA	OFA

#7 (OVL) : An overload alarm is being generated.
#6 (LV) : A low voltage alarm is being generated in servo amp.
#5 (OVC) : A overcurrent alarm is being generated inside of digital servo.
#4 (HCA) : An abnormal current alarm is being generated in servo amp.
#3 (HVA) : An overvoltage alarm is being generated in servo amp.
#2 (DCA) : A regenerative discharge circuit alarm is being generated in servo amp.
#1 (FBA) : A disconnection alarm is being generated.
#0 (OFA) : An overflow alarm is being generated inside of digital servo.

	#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):

#7 (ALD) 0 : Motor overheating
1 : Amplifier overheating

When FBAL 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	#6	#5	#4	#3	#2	#1	#0
204		OFS	MCC	LDA	PMS			

#6 (OFS) : A current conversion error has occured in the digital servo.
#5 (MCC) : A magnetic contactor contact in the servo amplifier has welded.
#4 (LDA) : The LED indicates that serial pulse coder C is defective
#3 (PMS) : A feedback pulse error has occured because the feedback cable is defective.

6) 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 **Notes**)
501	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1 to 8) – side stored stroke limit I. (Parameter No.1321 or 1327 **Notes**)
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)
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.

7) Overheat alarms

Number	Message	Contents
700	OVERHEAT: CONTROL UNIT	Control unit overheat. Check that the fan motor operates normally, and clean the air filter.
701	OVERHEAT: FAN MOTOR	The fan motor on the top of the cabinet for the contorl unit is overheated. Check the operation of the fan motor and replace the motor if necessary.

8) Safety zone alarms

Number	Message	Contents
4800	ZONE : PUNCHING INHIBITED 1	When a safety zone check was executed, a punch command was specified in area 1 where punching is inhibited.
4801	ZONE : PUNCHING INHIBITED 2	When a safety zone check was executed, a punch command was specified in area 2 where punching is inhibited.
4802	ZONE : PUNCHING INHIBITED 3	When a safety zone check was executed, a punch command was specified in area 3 where punching is inhibited.
4803	ZONE : PUNCHING INHIBITED 4	When a safety zone check was executed, a punch command was specified in area 4 where punching is inhibited.
4810	ZONE : ENTERING INHIBITED 1 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 1 into which entry is inhibited.
4811	ZONE : ENTERING INHIBITED 1 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 1 into which entry is inhibited.

Number	Message	Contents
4812	ZONE : ENTERING INHIBITED 2 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 2 into which entry is inhibited.
4813	ZONE : ENTERING INHIBITED 2 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 2 into which entry is inhibited.
4814	ZONE : ENTERING INHIBITED 3 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 3 into which entry is inhibited.
4815	ZONE : ENTERING INHIBITED 3 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 3 into which entry is inhibited.
4816	ZONE : ENTERING INHIBITED 4 +X	When a safety zone check was executed, the machine moving in the positive X direction entered area 4 into which entry is inhibited.
4817	ZONE : ENTERING INHIBITED 4 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 4 into which entry is inhibited.
4825	ZONE : ENTERING INHIBITED 8 –X	When a safety zone check was executed, the machine moving in the negative X direction entered area 8 into which entry is inhibited.
4830	ZONE : ENTERING INHIBITED 1 +Y	When a safety zone check was executed, the machine moving in the positive X direction entered area 1 into which entry is inhibited.
4831	ZONE : ENTERING INHIBITED 1 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 1 into which entry is inhibited.
4832	ZONE : ENTERING INHIBITED 2 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 2 into which entry is inhibited.
4833	ZONE : ENTERING INHIBITED 2 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 2 into which entry is inhibited.
4834	ZONE : ENTERING INHIBITED 3 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 3 into which entry is inhibited.
4835	ZONE : ENTERING INHIBITED 3 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 3 into which entry is inhibited.
4836	ZONE : ENTERING INHIBITED 4 +Y	When a safety zone check was executed, the machine moving in the positive Y direction entered area 4 into which entry is inhibited.
4837	ZONE : ENTERING INHIBITED 4 –Y	When a safety zone check was executed, the machine moving in the negative Y direction entered area 4 into which entry is inhibited.
4870	AUTO SETTING FEED ERROR	The feed rate of safety zone auto setting is other than the parameter value (No. 16538, No. 16539).
4871	AUTO SETTING PIECES ERROR	In safety zone auto setting, the safety zone pieces are not correct. Or the position detector has gone wrong, please tell your machine tool builder.
4872	AUTO SETTING COMMAND ERROR	M code, S code or T code is specified with safety zone auto setting command (G32). G32 is specified in the nibbling mode, in the cutter compensation, in the rotation mode or the scaling mode.

9) System alarms
(These alarms cannot be reset with reset key.)

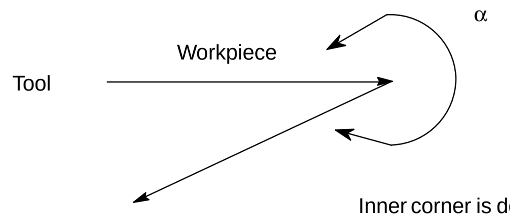
Number	Message	Contents
900	ROM PARITY	A parity error occurred in the CNC, macro, or servo ROM. Correct the contents of the flash ROM having the displayed number.
910	SRAM PARITY : (BYTE 0)	A RAM parity error occurred in the part program storage RAM. Clear the RAM, or replace the SRAM module or motherboard. 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.
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]

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).
Advanced preview 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.
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.
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.  Inner corner is defined by 180° ≤ α
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.
Bending compensation	Compensating the position of a hole for displacement due to the workpiece being bent.

Term	Description
Block	One of the command units constituting a program.
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.

Term	Description
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 mid-way.
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,
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.

Term	Description

[D]

Term	Description
D code	Coded number, following the D address, that specifies a tool offset number (machining center).
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.
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]

Term	Description
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. (I would remove this item, too confusing and no help)
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

Term	Description
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.
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.

Term	Description
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.
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.

Term	Description
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.
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.

Term	Description
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.

[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.
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.

Term	Description
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 work-piece 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).
Mode selection	Selecting an operation mode.

Term	Description
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.

Term	Description
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.
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.

Term	Description
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.
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.
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.

Term	Description
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.
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.
Punch forbidden area	Disabling punch commands (if any) in a safety zone.

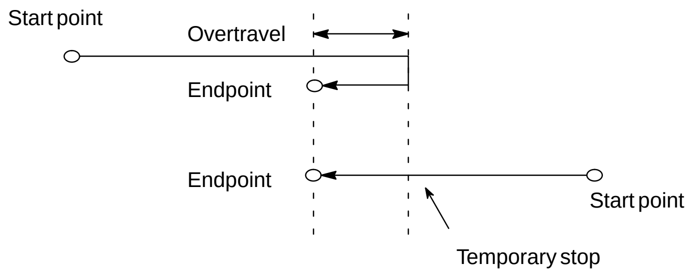
[R]

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.

Term	Description
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.

[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.

Term	Description
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.
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.
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.

Term	Description
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.
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.

Term	Description
Tape start	Symbol indicating the beginning of a program file.
Tapping mode	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.
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)

Term	Description
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.
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.
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]

[R]

[S]

[T]

[U]

[V]

[W]

Revision Record

FANUC Series 0*i*–PC OPERATOR'S MANUAL (B–64154EN)

Edition	Date	Contents	Edition	Date	Contents
01	Jun., 2004				