

CNC

MELDAS 600M Series

PROGRAMMING MANUAL (M2/M0 FORMAT)



MELDAS is a registered trademark of Mitsubishi Electric Corporation.
Other company and product names that appear in this manual are trademarks or registered trademarks of the respective companies.

Introduction

This instruction manual mainly targets milling machines and machining centers. It is to be used as a guide when using CNC MELDAS 600M Series fixed-format high-performance contour control software.

This instruction manual describes the MELDAS-M2 or M0 format programming for all the models stated above, so read this instruction manual thoroughly before using.

This manual has been written assuming that all functions have been provided in all of the models stated above. However, not all options are provided with every CNC, so always confirm functions, operations, etc., described in this manual with the specifications issued by the machine manufacturer.






Thoroughly read the "Safety Precautions" given on the next page to ensure safe use of this numerical control unit.

Details described in this manual

- (1) This manual gives general explanations from the standpoint of the NC side.
For explanations concerning individual machine tools, refer to the instruction manual issued by the machine manufacturer.
For items described as "Restrictions", "Usable State", etc., the instruction manual issued by the machine manufacturer takes precedence over this manual.
- (2) While every effort has been made to describe special handling in this manual, items not described in this manual should be interrupted as "Not Possible".
- (3) Refer to the following manual for details on the screen operations.
MELDAS M600M Series Instruction Manual BNP-B2237
- (4) To use the manual, set the "base common parameters" (machine parameter) to either of the following:

M2 label O is 0: Program number L, subprogram call number L M2 label O is 1: Program number O, subprogram call number A
--

CAUTION

-  For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine manufacturer takes precedence over this manual.
-  Matters not described in this instruction manual should be interpreted as "not possible".
-  This manual is written on the assumption that all option functions are added. Refer to the specifications issued by the machine manufacturer before starting use.
-  Refer to the instruction manual issued by each machine manufacturer for details on each machine tool.
-  Some screens and functions may differ or may not be usable depending on the NC system version.

Precautions for Safety

Always read the specifications issued by the machine manufacturer, this manual, related manuals and attached documents before installation, operation, programming, maintenance or inspection to ensure correct use.

Understand this numerical control unit, safety items and cautions before using the unit.

This manual ranks the safety precautions into "**DANGER**", "**WARNING**" and "**CAUTION**".



DANGER

When the user may be subject to imminent fatalities or major injuries if handling is mistaken.



WARNING

When the user may be subject to fatalities or major injuries if handling is mistaken.



CAUTION

When the user may be subject to injuries or when physical damage may occur if handling is mistaken.

Note that even items ranked as "**CAUTION**", may lead to major results depending on the situation. In any case, important information that must always be observed is described.



DANGER

Not applicable in this manual.








WARNING

Not applicable in this manual.






CAUTION

1. Items related to product and manual

-  For items described as "Restrictions" or "Usable State" in this manual, the instruction manual issued by the machine manufacturer takes precedence over this manual.
-  Matters not described in this instruction manual should be interpreted as "not possible".
-  This manual is written on the assumption that all option functions are added. Refer to the specifications issued by the machine manufacturer before starting use.
-  Refer to the instruction manual issued by each machine manufacturer for details on each machine tool.
-  Some screens and functions may differ or may not be usable depending on the NC system version.

2. Items related to programming

-  Because of key chattering etc., during editing, the commands with no value after G become a "G00" operation during running.
-  ";", "EOB" and "%" "EOR" are expressions used for the explanation. The actual codes are "line feed" and "%" for ISO, and "EOB" (End of Block) and "EOR" (End of Record) for EIA.
-  Do not change fixed cycle programs without the prior approval of the machine manufacturer.

CONTENTS

1. OUTLINE OF NC PROGRAM.....	1
1.1 Program Format	1
1.2 Word and Address.....	1
1.3 Block.....	1
1.4 End of Block	3
1.5 Modal and Unmodal Commands.....	4
2. BASIC CONCEPT FOR PROGRAMMING	5
2.1 Control Axis and Axis Name.....	5
2.2 Minimum Program Unit and Minimum Movement Amount	5
2.3 Incremental Value Command and Absolute Value Command	8
2.4 Reference Point (machine zero point position) and Zero Point Return	8
3. OUTLINE OF PROGRAMMING.....	9
3.1 Program to Specify the Movement of Tools along the Machining Workpiece Shape	9
3.2 Program to Command the Feed of a Tool at the Designated Rate to Cut a Workpiece.....	10
3.3 Program to Command Various Machine Operations such as Spindle Rotation and Stop and Coolant ON/OFF	10
3.4 Program to Command the Spindle Rotation Speed	11
3.5 Program to Select an Appropriate Tool for Workpiece Machining	11
3.6 Configuration of Program	12
3.7 Other Programming Notes	13
3.8 Coordinate Systems and Coordinate Zero Point Symbols	13
3.9 Optional Block Skip (Block delete)	14
4. POSITION COMMAND	15
4.1 Absolute/Incremental Command; G90/G91	15
4.2 Inch/Metric Changeover; G20/G21	17
4.3 Decimal Point Input I, II	18
4.4 Command Value and Setting Value Range.....	23
5. PREPARATORY FUNCTIONS.....	25
5.1 List of G Commands.....	25
6. POSITIONING AND INTERPOLATION FUNCTIONS.....	27
6.1 Positioning; G00	27
6.2 Unidirectional Positioning; G60	28
6.3 Linear Interpolation; G01	30
6.4 Circular Interpolation (designating center/radius); G02/G03	31
6.5 Hypothetical Axis Interpolation; G07	34
6.6 Cylindrical Interpolation; G07.1	35
6.7 Helical Interpolation; G02/G03	38
6.8 Spiral Interpolation; G02.1/G03.1	40
6.9 NURBS Interpolation; G06.2	42
6.10 Thread cutting; G33.....	45
7. FEED FUNCTIONS.....	49
7.1 Rapid Traverse Rate	49
7.2 Cutting Feedrate.....	50
7.3 Feed per Minute; G94	52
7.4 Feed per Rotation; G95.....	53
7.5 Inverse Time Feed; G93.....	54

7.6	Synchronized Tapping; ,R/,H	56
7.7	Dwell (Time designation); (G94) G04.....	65
7.8	Dwell (Rotation designation); (G95) G04	66
7.9	F1-digit Feed	67
8.	SPINDLE FUNCTIONS.....	70
8.1	Spindle Control Function.....	70
8.2	S Code Output.....	71
8.3	Spindle Position Control (Spindle C axis control).....	72
9.	TOOL FUNCTIONS	73
9.1	T Commands.....	73
10.	MISCELLANEOUS FUNCTIONS	75
10.1	Miscellaneous Functions.....	75
10.2	M Individual Output	76
10.3	2nd Miscellaneous Functions.....	77
11.	TOOL COMPENSATION	78
11.1	Tool Length Offset; G43/G44	78
11.2	Tool Radius Compensation; G40/G41/G42/G38/G39	84
11.2.1	Outline	84
11.2.2	Command format.....	85
11.2.3	Tool radius compensation operations	86
11.2.4	Other commands and operations for tool radius compensation.....	95
11.2.5	G41/G42 commands and I, J, K designation.....	104
11.2.6	Interrupts during tool radius compensation	109
11.2.7	General cautions for tool radius compensation	111
11.2.8	Changing the compensation number in the compensation mode	112
11.2.9	Tool radius compensation start and Z-axis cutting.....	114
11.2.10	Interference check.....	115
11.2.11	Compensation amount diameter designation.....	121
11.3	Three-dimensional Tool Radius Compensation; G40/G41/G42.....	122
11.4	Shape Compensation Amount, Wear Compensation Amount	132
11.4.1	Outline	132
11.4.2	Detailed description of shape compensation amount.....	132
11.4.3	Detailed description of wear compensation amount.....	133
11.5	Number of Compensation Sets	134
11.6	Tool Position Offset; G45/G46/G47/G48.....	135
12.	COORDINATE SYSTEM SETTING FUNCTIONS.....	137
12.1	Automatic Coordinate System Setting.....	137
12.2	Machine Coordinate System	138
12.3	Workpiece Coordinate System; G54 to G59	139
12.4	Local Coordinate System; G52	142
12.5	External Workpiece Coordinate Offset	144
12.6	Coordinate System Setting; G92.....	145
12.7	Rotation Axis Coordinate System.....	146
12.8	Plane Selection; G17/G18/G19	147
12.9	Automatic 1st Reference Point Return; G28/G29	149
12.10	2nd, 3rd and 4th Reference Point Return; G30.....	151
12.11	Reference Point Check; G27.....	153

13. PROGRAMMING SUPPORT FUNCTIONS 154

- 13.1 High-speed Machining Mode; G05 154
- 13.2 Subprogram Control; G22/G23 156
- 13.3 Fixed Cycle; G80 to G89/G73/G74/G76..... 158
 - 13.3.1 Outline 158
 - 13.3.2 Command format..... 158
 - 13.3.3 Description of operation 160
 - 13.3.4 Detailed description..... 162
 - 13.3.5 Description of each command..... 163
- 13.4 Corner Chamfering, Corner R; ,C/,R 172
 - 13.4.1 Outline 172
 - 13.4.2 Corner chamfering..... 172
 - 13.4.3 Corner R..... 174
 - 13.4.4 Corner chamfering/corner R speed E designation 176
- 13.5 Mirror Image; G50.1/G51.1/G62..... 177
 - 13.5.1 Outline 177
 - 13.5.2 External switch mirror image and parameter set mirror image..... 177
 - 13.5.3 G command mirror image..... 178
- 13.6 Variable Commands..... 180
 - 13.6.1 Outline 180
 - 13.6.2 Detailed description..... 180
- 13.7 User Macro Specifications..... 182
 - 13.7.1 User macro commands 182
 - 13.7.2 Macro call instruction..... 183
 - 13.7.3 Variables 192
 - 13.7.4 Types of variables 194
 - 13.7.5 Arithmetic commands..... 215
 - 13.7.6 Control commands 219
 - 13.7.7 External output commands..... 222
 - 13.7.8 Checkpoints..... 224
 - 13.7.9 Actual examples of using user macros..... 226
- 13.8 Linear Angle Command..... 230
- 13.9 Geometric Command 231
- 13.10 Pattern Rotation; G22 I_J 232
- 13.11 Program Coordinate Rotation; G68.1/G69.1 234
- 13.12 Parameter Coordinate Rotation..... 236
- 13.13 Special Fixed Cycles; G34/G35/G36/G37 238
- 13.14 Circular Cutting; G12/G13 243
- 13.15 Scaling; G50/G51 244
- 13.16 Normal Line Control; G40.1/G41.1/G42.1 247
- 13.17 Macro Interrupt; ION / IOF..... 252
- 13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1 256
- 13.19 Cutting Mode; G64 272
- 13.20 Automatic Corner Override; G62..... 273
- 13.21 Tapping Mode; G63..... 274
- 13.22 High-accuracy Control; G61.1 275
- 13.23 High-accuracy Control for Die Machining; G61.2/G61.3 280
- 13.24 High-accuracy Control for Part Machining; G61.5 283
- 13.25 Program Parameter Input; G10L70/G11.1 285
- 13.26 Program Compensation Input; G10..... 286
- 13.27 N Number Single Command Skip 288
- 13.28 End Point Error Check Clear; G69 290
- 13.29 Coordinate Read Function;G14..... 293

14. AUTOMATED SUPPORT FUNCTIONS	296
14.1 High-speed Skip; G31	296
14.2 Multi-step Skip; G31.1 to G31.4	299
14.3 Automatic Tool Length Measurement; G37.1	302
15. PROTECTION FUNCTIONS	304
15.1 Pre-movement Stroke Check; G22.1/G23.1	304
APPENDIX 1. LIST OF VARIABLE NUMBERS	306
APPENDIX 2. FIXED CYCLE PROGRAM	309
2.1 Parameters for Fixed Cycle Program Operation	309
2.2 Input/Output of Fixed Cycle Programs	310
2.3 Standard Fixed Cycle Programs	311
APPENDIX 3. LIST OF ALARMS	317
3.1 Program Errors	317
3.2 Program Warnings	330
APPENDIX 4. M2/M0 MACRO CONVERTOR	331
4.1 Restrictions	331
4.2 Operation	333
4.3 Detailed description	333

1. OUTLINE OF NC PROGRAM

1.1 Program Format

1. OUTLINE OF NC PROGRAM

This chapter describes some of the terminologies of NC programming.

1.1 Program Format

There are some predefined formats to supply control information to numerical control unit (hereinafter referred to as CNC) to control the movement of NC machine tool. This CNC employs the format of the word address system.

In this system, the control information supplied to the machine is configured of a combination of each alphabetic initial character and multiple digits of numeric characters. As the role (address) is assigned to each alphabet, it can be specified in any order.

The following programs cause the machine to perform the same operation.

Example 1) N001 G00 X-100.0 Y150.0 M03 S1000 ;

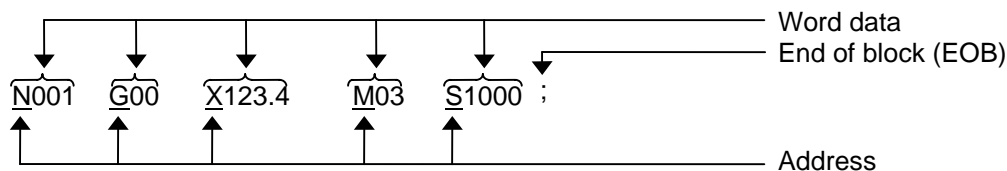
Example 2) M03 G00 S1000 Y150.0 N001 X-100.0 ;

It is recommended to use the order of N, G, X, Y, Z, F, M, S, T... shown in Example 1) because it is easier to read the program.

1.2 Word and Address

Each alphabetic initial character described in section 1.1 is called address, and the control information that follows the address, consisting of multiple digits of numeric value, is called word data (or simply word). The address has the predefined function.

The number of digits that can be designated for each word data varies depending on the address.



Therefore, each word data is the basic element of the NC program and the series of these word data forms the program that performs actual workpiece cutting.

The word data that are represented by the addresses G, M, S and T are called G code, M code, S code and T code, respectively. Table 1 lists the specified number of digits of each word data.

1.3 Block

To cause the machine to make a necessary movement, generally, it is required to specify a line of statement (control information) that is configured of multiple word data continuously. This line of statement is called "block" (or 1 block).

A block is a self-sufficient program and the series of these blocks constitutes the machining program (or NC program) used in the actual cutting.

1. OUTLINE OF NC PROGRAM

1.3 Block

Table 1 Details of the number of digits in word data

Item	Input unit	Metric commands (mm)	Inch commands (inch)
Program number		L8 (O8)	
Sequence number		N5	
Ready function		G3/G21	
Move-ment axis	A 0.01 (°), mm/0.001 inch	X+52 Y+52 Z+52 α+52	X+53 Y+53 Z+53
	B 0.001 (°), mm/0.0001 inch	X+53 Y+53 Z+53 α+53	X+44 Y+44 Z+44
	B' 0.001 (°), mm/0.0001 inch	X+43 Y+43 Z+43 α+43	X+34 Y+34 Z+34
	C 0.0001 (°), mm/0.00001 inch	X+44 Y+44 Z+44 α+44	X+35 Y+35 Z+35
Auxiliary axis	A 0.01 (°), mm/0.001 inch	I+52 J+52 K+52	I+53 J+53 K+53
	B 0.001 (°), mm/0.0001 inch	I+53 J+53 K+53	I+44 J+44 K+44
	B' 0.001 (°), mm/0.0001 inch	I+43 J+43 K+43	I+34 J+34 K+34
	C 0.0001 (°), mm/0.00001 inch	I+44 J+44 K+44	I+35 J+35 K+35
Arc center	A 0.01 (°), mm/0.001 inch	I+62 J+62 K+62 R+62	I+53 J+53 K+53 R+53
	B 0.001 (°), mm/0.0001 inch	I+63 J+63 K+63 R+63	I+54 J+54 K+54 R+54
	B' 0.001 (°), mm/0.0001 inch	I+53 J+53 K+53 R+53	I+44 J+44 K+44 R+44
	C 0.0001 (°), mm/0.00001 inch	I+54 J+54 K+54 R+54	I+45 J+45 K+45 R+45
Dwell	A 0.01 (rev/s)	X53 P8	X53 P8
	B 0.001 (rev/s)	X53 P8	X53 P8
	B' 0.001 (rev/s)	X53 P8	X53 P8
	C 0.001 (rev/s)	X53 P8	X53 P8
Feed function (feed per minute)	A 0.01 (rev), mm/0.001 inch	F62	F53
	B 0.001 (rev), mm/0.0001 inch	F63	F54
	B' 0.001 (rev), mm/0.0001 inch	F53	F44
	C 0.0001 (rev), mm/0.00001 inch	F54	F45
Feed function (feed per rotation)	A 0.01 (rev), mm/0.001 inch	F53	F43
	B 0.001 (rev), mm/0.0001 inch	F44	F34
	B' 0.001 (rev), mm/0.0001 inch	F34	F24
	C 0.0001 (rev), mm/0.00001 inch	F35	F25
Fixed cycle	A 0.01 (°), mm/0.001 inch	R+52 Q52 P8 L4	R+53 Q53 P8 L4
	B 0.001 (°), mm/0.0001 inch	R+53 Q53 P8 L4	R+44 Q44 P8 L4
	B' 0.001 (°), mm/0.0001 inch	R+43 Q43 P8 L4	R+34 Q34 P8 L4
	C 0.0001 (°), mm/0.00001 inch	R+44 Q44 P8 L4	R+35 Q35 P8 L4
Tool compensation number		H3/D3	
Miscellaneous function		M8	
Spindle function		S8	
Tool function		T8	
2nd miscellaneous function		B8/A8/C8	
Subprogram		L8H5L4 (A8H5L4)	
Variable number		#5	

B' is internal sub-micrometric system (unit of display/setting is one micron, and an internal unit of processing is 0.1 microns). Please inquire to the machine manufacturer whether the machine of your use is an internal sub-micrometric system.

(Note 1) "α" indicates the U, V, W, A or B additional axis address.

(Note 2) This format is the same for every numerical value input made from the tape, memory, MDI or the setting and display unit.

(Note 3) Excluding tool compensation numbers, all leading zeros in numbers can be omitted.

(Note 4) Program numbers are commanded in independent blocks. Command them in the head block of each program.

(Note 5) The significance of the abbreviations used are as follows.

Example 1) O8 : 8-digit program number

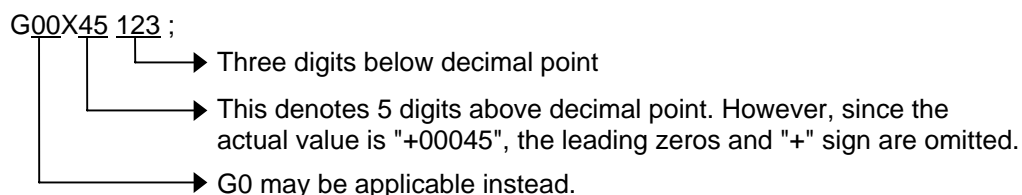
Example 2) G21 : Address G has two digits to the left of the decimal point and one digit to the right.

Example 3) X+53 : Address X has a "+" or "-" sign and five digits to the left of the decimal point and three digits to the right.

1. OUTLINE OF NC PROGRAM

1.4 End of Block

The example given below is a case where the X axis is positioned (G00) at the 45.123mm position in the absolute value (G90) mode.



Example) Configuration of one block in the machining program

```

•
•
N100 G01 X123.4 Y-100.0 F1000.0 ;
N006 M03 S1000 T01 ;
N007 G0 Z-100.0 ;
•
•

```

To configure one block of data, care should be taken because the following restrictions exist.

- 1) The number of digits of word data that can be specified is determined by the type of address. Refer to Table 1 for details.
- 2) The word data except those which are represented by the addresses G and M cannot be designated in duplicate. If they are designated in duplicate, the word data which is designated later becomes valid.
- 3) The total number of characters of each word that can be designated in a single line is limited to within 128 characters including the end of block code (EOB) (described in later section).
- 4) The initial character of the address can be either lower or upper case.

(Note 6) The addresses of the program number and subprogram call number depend on the parameter setting.

		Address of program number	Address of subprogram call number
Base common parameter (Machine parameter)	M2 label 0 = 0	L	L
	M2 label 0 = 1	O	A

1.4 End of Block

The end of block is abbreviated as EOB and is the information code to inform the CNC unit of the end of each block of command. Therefore, the EOB should be designated for each block. In this manual, the EOB is represented by the symbol " ; ". This corresponds to key on the keyboard of CNC unit or the key of a personal computer.

The CNC unit ignores all information until the first EOB code is read after the power is turned ON or reset. To ensure the execution from the start block of the machining program, the EOB should be designated in the start block.

Example 1) N001 G00 X100.0 Y-50.0 ;
 N002 G00 Z-30.0 ;
 N003 S3000 M03 ;

If this program is started after the NC power is turned ON or reset, the block N001 is ignored and the block N002 and onward are executed. Therefore, the following programming should be made as shown in Examples 2) and 3). (Except machining programs stored in the NC memory.)

Example 2) ;
 N001 G00 X100.0 Y-50.0 ;
 N002 G00 Z-30.0 ;
 N003 S3000 M03 ;

Example 3) TEST PROGRAM #1 ;
 N001 G00 X100.0 Y-50.0 ;
 N002 G00 Z-30.0 ;
 N003 S3000 M03 ;

1. OUTLINE OF NC PROGRAM

1.5 Modal and Unmodal Commands

1.5 Modal and Unmodal Commands

Some of the word data are stored in the CNC once they are designated and to when they are updated or canceled, while other data need to be designated for each block each time they are used. The former is called modal command and the latter is called unmodal command. For example, the word data F that designates the feedrate described in later section of this manual is a typical modal command and the word data that designates the movement amount such as data X, Y or Z is a typical unmodal command. Avoiding the duplicate designation of the modal commands leads to efficient programming.

2. BASIC CONCEPT FOR PROGRAMMING

This section describes some of the basic concepts for programming.

2.1 Control Axis and Axis Name

The normal vertical machining center with a CNC (hereinafter referred to as MC) performs cutting (machining) operation by singly or simultaneously moving the table that moves horizontally (left/right and forward/backward) and the spindle head that moves up and down.

The mobile parts that move in three directions are called the "control axis" and have the following axis names.

Control axis that carries out left/right feed of the table X axis
Control axis that carries out forward/backward feed of the table Y axis
Control axis of the spindle head that moves up and down Z axis

The axes of the horizontal machining center are called as follows because the spindle head is placed horizontally (forward and back word movement)

Control axis that carries out left/right feed of the table X axis
Control axis of the spindle head that moves up and down Y axis
Control axis that carries out forward/backward feed of the table Z axis

The X, Y and Z axes are usually used for linear axis while the A, B and C axes are used for the rotating axis such as rotary table. The U, V and W axes are used for the address of the additional axis name.

Up to 14 axes can be used according to the CNC type. This instruction manual assumes the maximum number of the control axes is 4.

Refer to instruction manual of machine manufacturer for the actual configuration of the control axes.

Fig. 1 shows the configuration example of the three control axes (X, Y, Z).

2.2 Minimum Program Unit and Minimum Movement Amount

In contrast to the movement amount of the control axis, the minimum unit of movement amount that can be designated by the program and the minimum movement amount that the machine actually makes consist of three types as shown in the Table 2. The type is selectable by the parameters specified by the CNC unit.

The type CS-B of 1 μ m system shown in the Table 2 is most frequently used.

2. BASIC CONCEPT FOR PROGRAMMING
2.2 Minimum Program Unit and Minimum Movement Amount

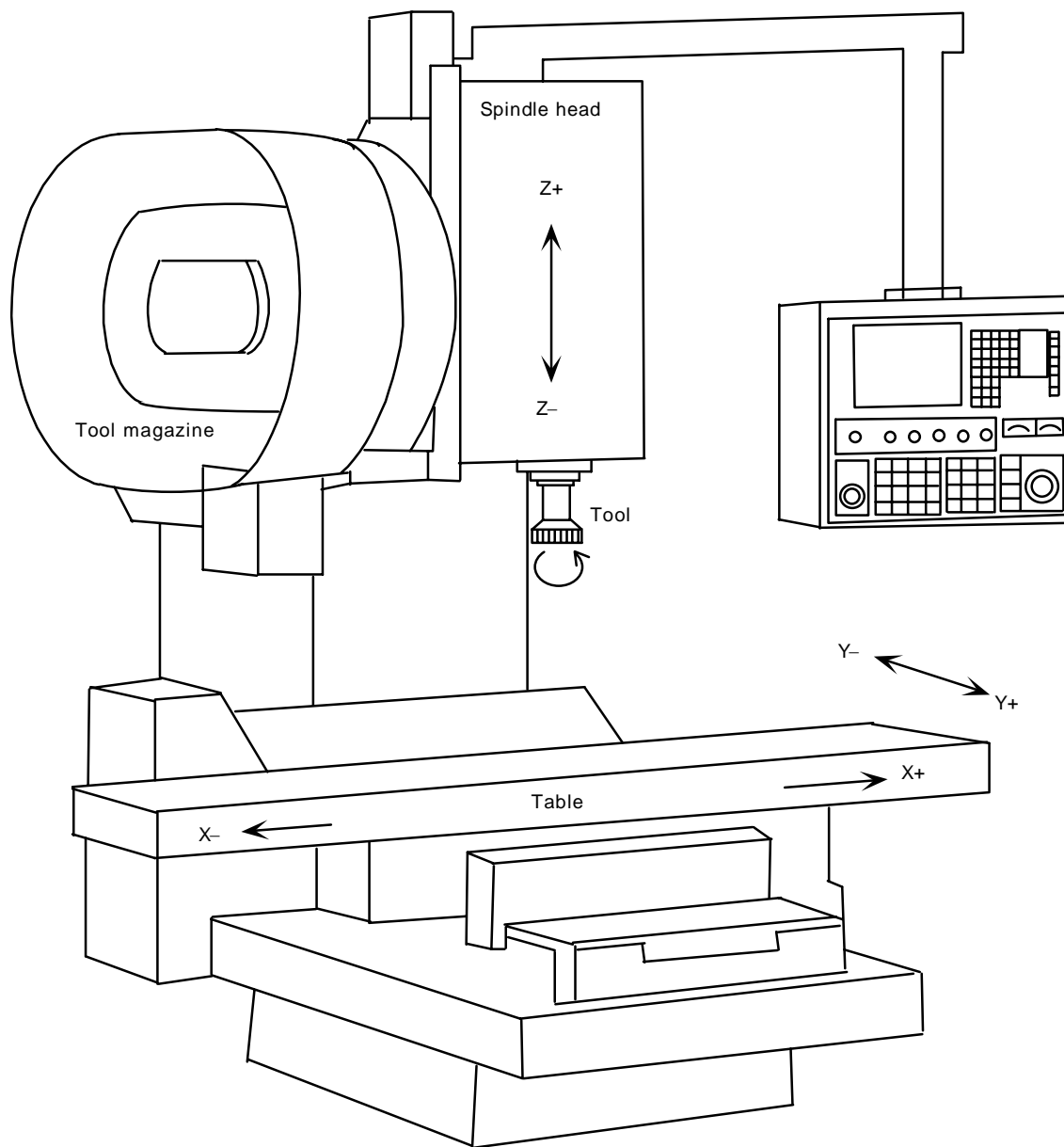


Fig. 1 Outline of three-control-axis type vertical machining center

2. BASIC CONCEPT FOR PROGRAMMING
2.2 Minimum Program Unit and Minimum Movement Amount

Table 2 Input command unit type and the type of minimum movement amount

	Type	Linear axis		Rotating axis (°)
		Metric command (mm)	Inch command (inch)	
Input command unit	CS-A	0.01	0.001	0.01
	CS-B	0.001	0.0001	0.001
	CS-C	0.0001	0.00001	0.0001

(Note 1) The metric/inch command can be changed over by the parameters of CNC unit and by the G command (G20/G21).

(Note 2) The metric/inch command cannot be used simultaneously.

[Example of designating the 1 μ m system]

Designated by the program	Movement amount of the machine
X1	X axis 0.001mm
Y10	Y axis 0.01mm
Z1000	Z axis 1.0mm
X0.1	X axis 0.1mm
Y10.0	Y axis 10.0mm
A-90000	A axis 99.0°
B10.009	B axis 10.009°
Z-1.0001	Cannot designate because the 1 μ m system is limited to less than three digits under the decimal point.

[Example of designating by the 0.1 μ m system] (This is especially called submicrometer command.)

Designated by the program	Movement amount of the machine
X100	X axis 0.01mm
Y-10	Y axis -0.001mm
Z-1	Z axis -0.0001mm
X0.0001	X axis 0.0001mm
Y1.0	Y axis 1.0mm
Z10.0009	Z axis 10.0009mm
A90.0001	A axis 90.0001°
B9000	B axis 9.0°

Refer to the instruction manual of machine manufacturer for details of the actual setting unit.

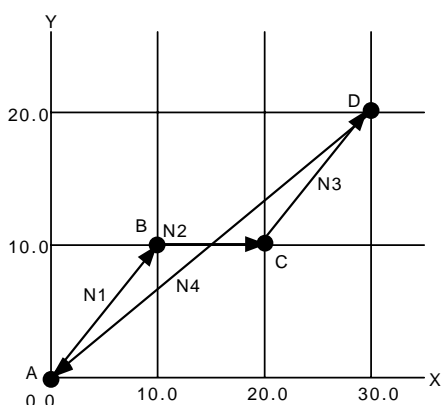
2. BASIC CONCEPT FOR PROGRAMMING

2.3 Incremental Value Command and Absolute Value Command

2.3 Incremental Value Command and Absolute Value Command

To designate the movement amount of the control axis, either the relative value command method (incremental) or the absolute value command method (absolute) can be chosen. The type can be selected freely, but depending on the program details some machines need to be designated by the incremental value type only.

Examples are given in the following for better illustration. Refer to Section "4. POSITION COMMAND" for details.



Difference is shown between the increment value command program and the absolute value command program when the table returns immediately to the point A after the table has moved from the point A to the point D (positioning).

	Incremental value command method	Absolute value command method
A→B	N1 G0 X10.0 Y10.0;	N1 G0 X10.0 Y10.0;
B→C	N2 X10.0 Y0; ^(Note 1)	N2 X20.0 Y10.0;
C→D	N3 X10.0 Y10.0;	N3 X30.0 Y20.0;
D→A	N4 X-30.0 Y-20.0;	N4 X0 Y0;

(Note 1) The increment value command can be omitted if the movement amount is zero.

The absolute value command method is more frequently used.

2.4 Reference Point (machine zero point position) and Zero Point Return

For each control axis of the machine, a specific machine position is specified as the reference point of the machine coordinate system. This machine position is called reference point (or machine zero point position). The reference point has the important role of being the reference point when changing the coordinate system of the machining program or the tool as well as when setting the tool offset amount. Most CNC machine tools are equipped with the function to automatically return to the zero point position by the program or by manual operation. This is called zero point return. Refer to the instruction manual of machine manufacturer for the reference point position of each axis.

If the operation is to be restarted after the NC power has been turned OFF or after emergency stop, the return to the zero point return is performed. (Except in the case where the absolute position detection function is provided.)

3. OUTLINE OF PROGRAMMING

3.1 Program to Specify the Movement of Tools along the Machining Workpiece Shape

3. OUTLINE OF PROGRAMMING

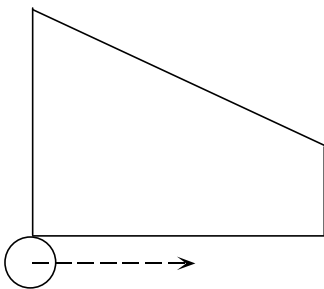
3.1 Program to Specify the Movement of Tools along the Machining Workpiece Shape

The shape of all machining workpieces is configured of combinations of lines and arcs. Therefore, the program commands ^(Note 1) are divided into two groups; commands that move the tool along the line and commands that move the tool along the arc. But, in the actual machining, commands to traverse the table to the specified position at a higher speed are needed in addition to the commands that move the tool along the shape.

(1) Program to command the movement of a tool along the line

Program format

G01 X Y F;



G01 (or G1).... G code that designates the movement along the line ^(Note 2)

X, Y End point coordinate value of linear movement

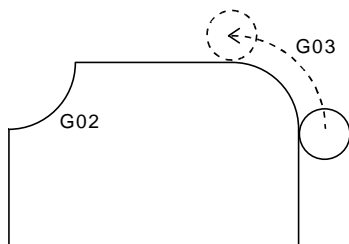
F..... Tool feedrate

(2) Program to command the movement of a tool along the arc

Program format

G02 X Y R F;

G03 X Y R F;



G02 G code to designate the clockwise arc movement

G03 G code to designate the counterclockwise arc movement

X, Y Coordinate value of arc end point

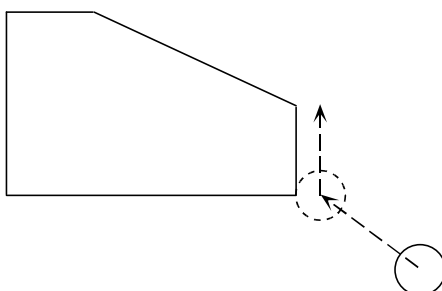
R Arc radius (The arc center may be designated by I, J, and K instead of arc radius. Refer to details in later section.)

F Tool feedrate

(3) Program to designate the tool rapid traverse

Program format

G00 X Y;



G00 G code to designate the tool rapid traverse movement

X, Y Coordinate value of positioning end point

3. OUTLINE OF PROGRAMMING

3.2 Program to Command the Feed of a Tool at the Designated Rate to Cut a Workpiece

(Note 1) In the actual machine, the table may move while the tool may not. This manual assumes the tool moves to the workpiece. There is no difference in programming.

(Note 2) The leading zeros of the numeric value data that follow the address such as G00, G01 and G02 can be omitted. The data can be represented as G0, G1 and G2.

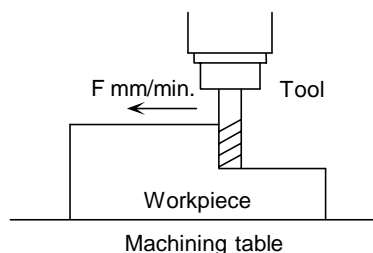
The function to move the tool along the linear or the arc is called interpolating function and the function to traverse the tool at a high rate is called positioning or rapid traverse function.

The G codes such as G00, G01, G02 and G03 are called preparation function (or G function) and there are many kinds of G functions. Refer to Section "6. POSITIONING AND INTERPOLATION FUNCTIONS" for details.

3.2 Program to Command the Feed of a Tool at the Designated Rate to Cut a Workpiece

To cut a workpiece, the feedrate of a tool should be considered according to the material or tool used. The function to designate the feedrate is called feed function (or F function). The feedrate is designated by numeric values.

For example, command F200.0 (or F200.) to feed a tool at 200mm per minute.



The F command is usually designated for the same block as the interpolation function such as G01 but it can be designated singly.

As this is one of the modal commands, it needs not be designated for each block until the feedrate needs to be changed. This applies to all interpolation functions (G01, G02, G03).

Refer to Section "7. FEED FUNCTIONS" for details.

3.3 Program to Command Various Machine Operations such as Spindle Rotation and Stop and Coolant ON/OFF

For machining a workpiece, it is necessary to rotate the spindle, to apply coolant (cutting oil) and to turn ON/OFF devices on the other machine side. The function that commands this series of operations is called miscellaneous function (or M function). This is generally designated by a two digit of numeric value (M code) that follows the address M.

For example, M03 commands the forward rotation of the spindle, M04 does the reverse rotation of the spindle and M05 stops the spindle. Refer to instruction manual of machine manufacturer for details of the functions of the M code.

The M codes are not fully defined by machine manufacturer. The functions such as M00, M01, M02, M30, M98 and M99 are fixed on the CNC side.

Refer to Section "10. MISCELLANEOUS FUNCTIONS" for details.

3. OUTLINE OF PROGRAMMING

3.4 Program to Command the Spindle Rotation Speed

3.4 Program to Command the Spindle Rotation Speed

Like the F command, an appropriate tool rotation speed (spindle rotation speed) must be designated according to the material of the workpiece and the tool used. The rotation speed of the tool can be calculated based on the relative speed of the tool and the workpiece (cutting speed).

For example, if a workpiece needs to be machined at a cutting speed of 200m per minute with a tool of 100 \varnothing , the rotation spindle speed is 640r/min. So, designate the rotation speed (r/min) after the address S. That is the code of S640 used in this case.

It is also possible to command the required value directly without calculation.

The command related to the spindle rotation speed is called spindle function (S function). Since the S function is a modal command, the designated spindle rotation speed does not need to be set until the next change. If any new S function is commanded while the rotation spindle is rotating, the rotation speed is changed immediately. If the power has been turned ON or the spindle has been stopped by M05, the spindle rotation speed should be commanded simultaneously with spindle rotation commands (M03 or M04) or a spindle rotation command (M03, M04) should be issued first.

Refer to Section "8. SPINDLE FUNCTIONS" for details.

3.5 Program to Select an Appropriate Tool for Workpiece Machining

In the actual workpiece machining process, milling, drilling, tapping and boring are performed and appropriate tools should be used. Each tool has a number (tool number) and is commanded by the 2-digit numeric value (T code) following the address T. This is called tool function (or T function).

For example, if a 10 \varnothing drill has the tool number 5, command program T05 or T5 to select it.

The tool selection is closely related to the tool change method (ATC) of each machine manufacturer. Refer to instruction manual of machine manufacturer for details.

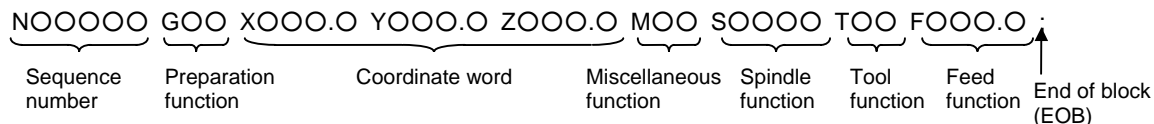
Refer to Section "9. TOOL FUNCTIONS" for other details.

3. OUTLINE OF PROGRAMMING

3.6 Configuration of Program

3.6 Configuration of Program

One block of program is configured as follows.



The sequence number designated at the head of each block is commanded by the maximum 5-digit numeric value following the address N. This is displayed on the screen while executing the program. It can be used to monitor the machining program execution states or to call a specific process of the machining program (program search).

The sequence number does not show the order in which the commands are executed. The program is executed according to the order of the block written in the program. (The command in the block is executed simultaneously.) The sequence number does not need to be commanded for each block. There is no influence on the machining even if it is not commanded at all.

The words which are not needed for block operation (G, X, Y, Z, ... F) can be omitted.

The machining program is generally configured as follows.

	At the head of program EOB (;)
;	Program number
L(O)1234 ;	Block
N001 G90 G00 X-10.0 Y-20 Z-30.0 M03 S1000 ;	Block
N002 M08 ;	Block
N003 G01 Z-40.0 F500.0 ;	Block
N004 G00 Z-30.0 ;	Block
;	;
;	;
N999 M02	Program end

The EOB (;) at the head of the program is followed by the program number and each block for machining, and the program end code of M02 or M30 is written at the end of the program.

The program number (maximum 8-digit numeric value following the address O) is displayed on the screen in the same way as the sequence number. By controlling this number, it is possible to know what machining program is being executed. This also used to select the program necessary for the machining when some kinds of machining program are stored in the memory of the CNC unit.

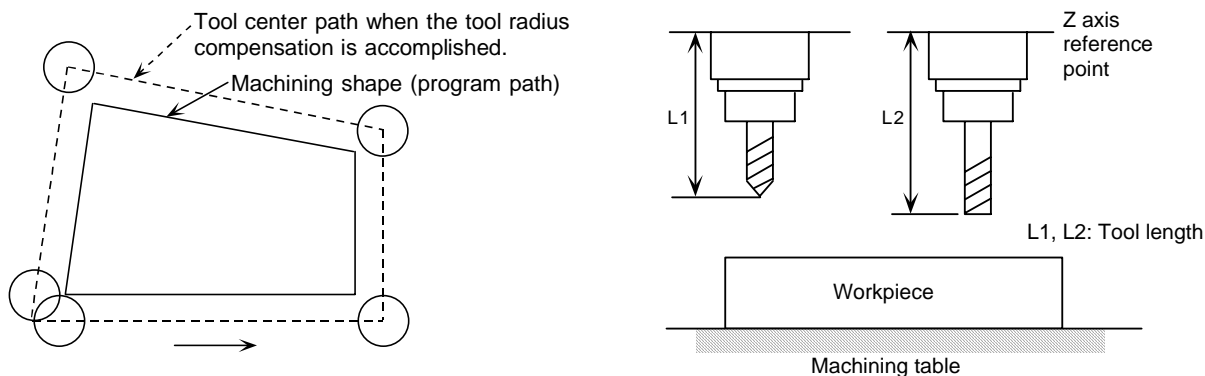
The program number should be designated for each machining program. The same number cannot be used twice. The block number is automatically assigned by the CNC, and need not be designated when programming. This is cleared to zero each time a new program number or sequence number is read, or counted up each time one block is read. If the count reaches 99999, it is cleared to zero and the counting starts again. The block number is displayed on the screen as in the same way as the sequence number.

3. OUTLINE OF PROGRAMMING

3.7 Other Programming Notes

3.7 Other Programming Notes

The machining program is usually written in such a way that the tool center moves along the contour lines on the machining drawing. But, in the actual machining, as tools of various thickness (tool radius) and length (tool length) are used, the actual tool path deviates by the amount of the radius and the length, and the exact machining cannot be expected if no measure is taken.

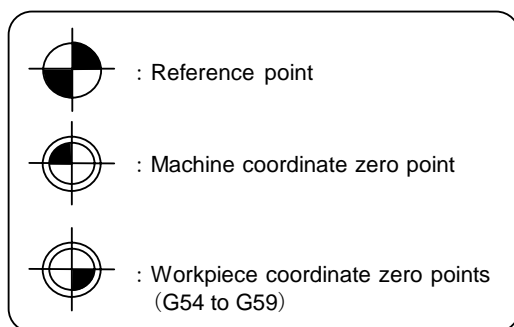


Therefore, this software is provided with the following tool offset functions so that programmers may write programs without considering the deviation.

- 1) Tool radius compensation function
- 2) Tool length offset function

These functions allow to register the radius and the length of a tool in the memory (offset memory) of the CNC unit and automatically compensate for the tool displacement due to the tool diameter and the tool length when the dedicated G code and the registration number are designated. Refer to Section "11. TOOL COMPENSATION" for details.

3.8 Coordinate Systems and Coordinate Zero Point Symbols



3.9 Optional Block Skip (Block delete)

This function selectively ignores specific blocks in a machining program which start with the " / " (slash) code.

(1) Optional block skip

Provided that the optional block skip switch is ON, blocks starting with the " / " code are ignored. They are executed if the switch is OFF.

Parity check is valid regardless of whether the optional block skip switch is ON or OFF.

When, for instance, all blocks are to be executed for one workpiece but specific blocks are not to be executed for another workpiece, the same machining program can be used to machine different parts by inserting the " / " code at the head of those specific blocks.

(2) Notes for using optional block skip

1) Put the " / " code at the head of the block. (In this example, an error of dividing by zero occurs.)

Example: N20 G01 X-25.0 /Y-25.0; Correct
/N20 G01 X-25.0 Y-25.0; Incorrect

2) Parity checks (H and V) are conducted regardless of the optional block skip switch status.

3) The optional block skip is processed immediately before the pre-read buffer.

Consequently, it is not possible to skip up to the block which has been read into the pre-read buffer.

4) This function is valid even during a sequence number search.

5) All blocks with the " / " code are also input and output during tape storage and tape output, regardless of the optional block skip switch status.

4. POSITION COMMAND

4.1 Absolute/Incremental Command; G90/G91

(1) Outline

The two methods of absolute/incremental commands are as follows.

- (a) G code command method (G90/G91)
- (b) Incremental axis address method

Which method is used is selected by the parameters.

(2) Detailed description

(a) G code command method (G90/G91)

When issuing the axis coordinate data with the machining program commands, the incremental command method (G91) to command the relative distance from the current position, or the absolute command method (G90) to move to the designated position of the predetermined coordinate system can be selected. Note that with some formats, G90 and G91 are used for other function commands, so the G code command method cannot be selected.

1) Command format

G90 Absolute command (Absolute value command)
 G91 Incremental command (Incremental value command)

These G codes can be commanded multiple times within one block.

<u>G90</u> X100.	<u>G91</u> Y200.	<u>G90</u> Z300.	;
Absolute value	Incremental value	Absolute value	

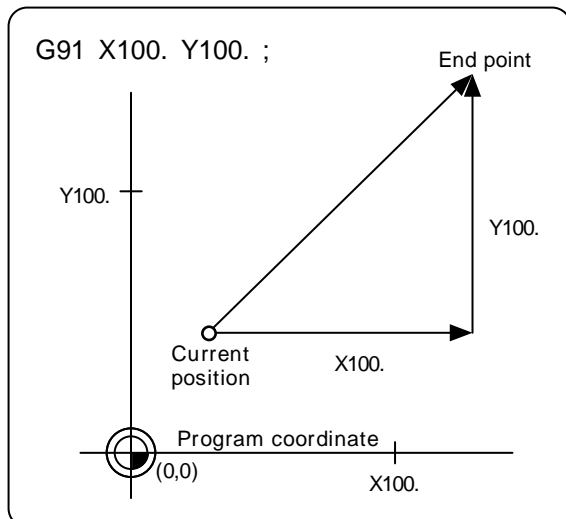
(Note) Even in the MDI command, the modal executed before is followed when there is no G90/G91 designation as with the memory command.

2) Combined use of absolute and incremental commands in the same block is possible. The changeover is carried out with G90 or G91.

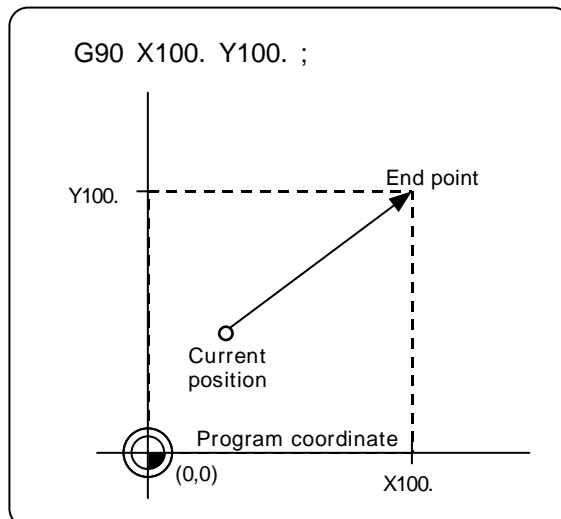
Note that the arc radius designation (R) is an incremental command. The arc center designation (I, J, K) can be changed by the parameters so it is fixed to incremental designation or so that it follows the G90 and G91 modal.

3) Figure description

(Incremental value command)



(Absolute value command)



4. POSITION COMMAND

4.1 Absolute/Incremental Command; G90/G91

(b) Incremental axis address method

When issuing the axis coordinate data with the machining program commands, the incremental command method to command the relative distance from the current position, or the absolute command method to move to the designated position of the predetermined coordinate system can be selected. Note that the arc radius designation (R) or arc center designation (I, J, K) are incremental designations.

1) Command method

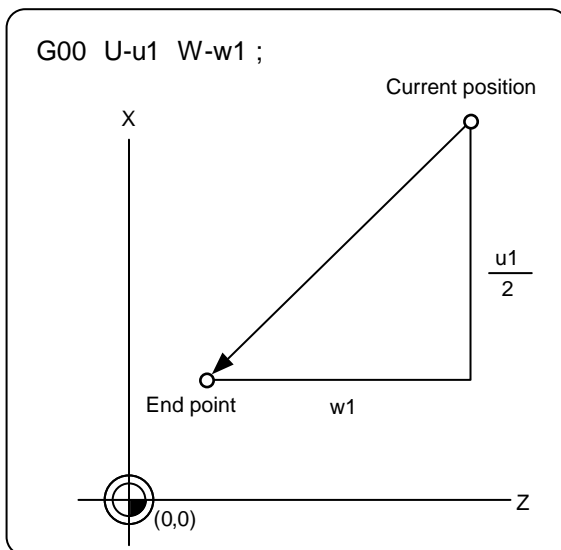
The incremental axis address method is validated when the ABS/INC address (user parameter) is ON, and the incremental axis name is set in the incremental axis name (machine parameter). Even when the ABS/INC address is ON, the G code (G90/G91) is followed if the incremental axis name is not set.

For standard: Absolute command (absolute value command) X, Z
 Incremental command (incremental value command) U, W

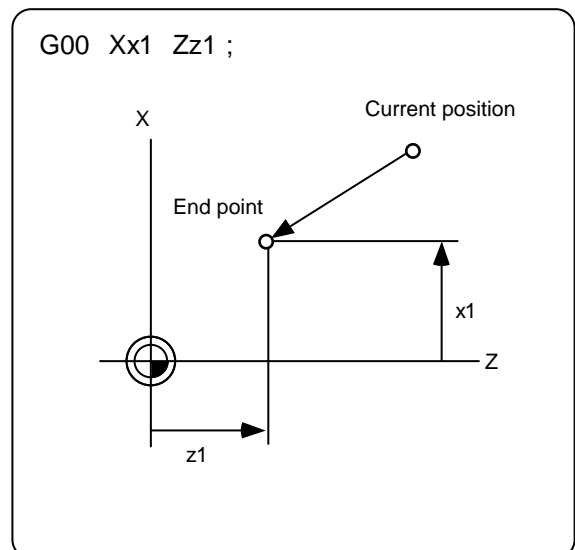
$G00 \quad \frac{X100.}{\text{Absolute value}} \quad \frac{W200.}{\text{Incremental value}} \quad ;$
--

2) Figure description (For a diameter command)

(Incremental value command)



(Absolute value command)



4. POSITION COMMAND
4.2 Inch/Metric Changeover; G20/G21

4.2 Inch/Metric Changeover; G20/G21

(1) Outline

The two unit systems handled by the NC are the metric unit system and the inch unit system. The unit system can be designated by the parameters and the machining program. Commands from the machining program are only valid in machining program commands; the unit system of the position display, etc., does not change.

(2) Detailed description

Inch/metric changeover	Target	Remarks
Input unit changeover (parameters)	Parameters (excluding some) Tool compensation amount Workpiece offset Machining program command Position display	Automatic conversion at parameter changeover (mm fixed internally). Restart necessary.
Output unit changeover (parameters)	Parameters (those depending on ball screws)	No automatic changeover. Restart necessary.
G20/G21	Machining program command only	

(3) Command format

- Inch command

G20 ;

- Metric command

G21 ;

4. POSITION COMMAND
4.3 Decimal Point Input I, II

4.3 Decimal Point Input I, II

(1) Outline

This function enables the decimal point command to be input. It assigns the zero point in millimeter or inch units for the machining program input information that defines the tool paths, distances and speeds.

The two types of decimal point commands are as follows. They can be selected by the parameters.

- 1) Decimal point input: Type 1 (When control parameter "Decimal point type 2" is OFF.)
- 2) Decimal point input: Type 2 (When control parameter "Decimal point type 2" is ON.)

(2) Detailed description

(a) The decimal point command is valid for distance, angle, time, speed and scaling magnification commands in the machining program.

(b) The two types of handling of command values for data with no decimal points are Decimal point input: Type 1 and Type 2. The type is selected by the parameters.

1) Decimal point input: Type 1 (When control parameter "Decimal point type 2" is OFF.)

When the axis coordinates, etc., data is issued in the machining program command, the program data designation can be simplified by using decimal point input. The least digit of commands not using a decimal point becomes the minimum setting unit.

Besides the axis coordinate value, other addresses in which this input type can be used are speed commands and dwell commands.

The decimal point position is an mm unit during metric mode, an inch unit during inch mode and a second unit during dwell time setting. For the rotation axis, the decimal point position is a degree, regardless of metric or inch mode.

2) Decimal point input: Type 2 (When control parameter "Decimal point type 2" is ON.)

Type 2 is different from Type 1. When there is no decimal point, the last digit is an mm unit during metric mode, an inch unit during inch mode, and a second unit in time setting.

Always designate a "." (point) when commands below the decimal point are necessary.

(c) The valid addresses of the decimal point command are XYZUVWABCDIJKEFGPQR. Refer to the list for details.

(d) The decimal point command is valid even for commands defining the variable data used in subprograms.

(e) Decimal point commands for decimal point invalid addresses are processed as integer data only and everything below the decimal point is ignored. Decimal point invalid addresses are HLMNOS T.

All variable commands, however, are handled as data with decimal points.

(f) Example of program

Machining program	Unit interpretation (for metric system)	
	Type I	Type II
G00 X100. Y-200.5	X100mm, Y-200.5mm	X100mm, Y-200.5mm
G1 X100 F20.	X100 μ m, F20mm/min	X100mm, F20mm/min
Y200 F100 (Note 1)	Y200 μ m, F100mm/min	Y200mm, F100mm/min
G4 X1.5 (Note 2)	Dwell 1.5s	Dwell 1.5s
X2	2ms	2s

(Note 1) F unit is mm/min. (inch/min. for inch system) for either types I or II.

(Note 2) Valid/invalid changeover of the decimal point command is possible with the control parameter "G04P decim pt valid".

4. POSITION COMMAND
4.3 Decimal Point Input I, II

(g) Addresses used and valid/invalid decimal point commands

Ad- dress	Decimal point command	Application	M/L	Remarks
A	Valid	Coordinate position data		
	Invalid	Miscellaneous function code		
	Valid	Geometric linear angle data		
	Invalid	Rotation table	M	
	Invalid	MRC program number	L	
	Invalid	Data setting, axis number	L	
	Valid	Deep hole drilling cycle (2) Safety distance	L	
B	Valid	Coordinate position data		
	Invalid	Miscellaneous function code		
	Invalid	Rotation table	M	
C	Valid	Coordinate position data		
	Invalid	Miscellaneous function code		
	Valid	Corner chamfering amount		, C
	Invalid	Rotation table	M	
	Valid	Program tool compensation input Tool nose radius compensation amount (incremental)	L	
D	Invalid	Offset number (tool position, tool radius compensation)	M	
	Valid	Automatic tool length measurement, deceleration range d	L	
	Valid	Data setting	L	
	Invalid	Subprogram storage device No.	M	, D
E	Valid	Inch threads, Precision thread lead		
F	Valid	Feedrate		
	Valid	Thread lead		
G	Valid	Preparatory function code		
H	Invalid	Sequence number in subprograms		
	Invalid	Offset number (tool position, tool length offset)	M	
I	Valid	Circular center coordinate		
	Valid	Tool radius compensation/tool nose radius compensation vector component		
	Valid	Special fixed cycle, hole pitch	M	
	Valid	Deep hole drilling cycle (2) First cut amount	L	
J	Valid	Circular center coordinate		
	Valid	Tool radius compensation/tool nose radius compensation vector component		
	Valid	Special fixed cycle, hole pitch and angle	M	
	Invalid	Deep hole drilling cycle (2) Dwell at return point	L	

4. POSITION COMMAND
4.3 Decimal Point Input I, II

Ad- dress	Decimal point command	Application	M/L	Remarks
K	Valid	Circular center coordinate		
	Valid	Tool radius compensation/tool nose radius compensation vector component		
	Invalid	Special fixed cycle, number of holes	M	
	Invalid	Hole drilling cycle, number of repetitions	L	
	Valid	Deep hole drilling cycle (2) Second cut amount	L	
	Invalid	Fixed cycle, number of repetitions	M	
L	Invalid	Program number/subprogram call number		L2, L10, L11
	Invalid	Program tool compensation input type selection	L	
	Invalid	Data setting selection	L	
M	Invalid	Miscellaneous function code		
N	Invalid	Sequence number		
	Invalid	Data setting, data number	L	
O	Invalid	Program number		
P	Valid/Invalid	Dwell time		Parameter
	Invalid	Special fixed cycle, number of holes	M	
	Invalid	Helical number of pitches	M	
	Invalid	Surface speed radius	M	
	Valid	Scaling magnification	M	
	Invalid	High-speed mode type	M	
	Invalid	2nd reference point number	L	
	Invalid	Constant surface speed control, axis number	L	
	Invalid	MRC finishing shape start sequence number	L	
	Valid	Cut-off cycle Shift amount/cut amount	L	
	Invalid	Compound thread cutting cycle, number of cutting passes, chamfering, tool nose angle	L	
	Valid	Compound thread cutting cycle Thread height	L	
	Invalid	Data setting, broad classification number	L	
	Invalid	Return sequence number from subprogram	L	
	Valid	Coordinate position data	L	
Invalid	Offset number (G10)			
Invalid	Fixed cycle/subprogram, number of repetitions			
Q	Valid	Deep hole drilling cycle, cut amount	M	
	Valid	Back boring shift amount	M	
	Valid	Fine boring shift amount	M	
	Invalid	Minimum spindle clamp rotation speed	L	

4. POSITION COMMAND
4.3 Decimal Point Input I, II

Ad- dress	Decimal point command	Application	M/L	Remarks	
Q	Invalid	MRC finishing shape end sequence number	L		
	Valid	Cut-off cycle Cut amount/shift amount	L		
	Valid	Compound thread cutting cycle Minimum cut amount	L		
	Valid	Compound thread cutting cycle First cut amount	L		
	Valid	Deep hole drilling cycle 1 Cut amount of each pass	L		
	Invalid	Program tool compensation input Hypothetical tool nose point number	L		
	Invalid	Deep hole drilling cycle 2 Dwell at cut point	L		
R	Valid	R-designated arc radius		, R	
	Valid	Corner R arc radius			
	Valid	Offset amount (G10)			
	Valid	Fixed cycle R point			M
	Invalid	Synchronous tap/asynchronous tap changeover			M
	Valid	Automatic tool length measurement, deceleration range r			L
	Valid	MRC longitudinal/face, escape amount			L
	Invalid	MRC forming, division number			L
	Valid	Cut-off cycle, retract amount			L
	Valid	Cut-off cycle, escape amount			L
	Valid	Compound thread cutting cycle, finishing allowance			L
	Valid	Compound thread cutting cycle/cutting cycle, taper difference			L
	Valid	Hole drilling cycle/deep hole drilling cycle (2), distance to R point			L
Valid	Coordinate position data	L			
S	Invalid	Spindle function code			
	Invalid	Maximum spindle clamp rotation speed			L
	Invalid	Constant surface speed control, surface speed			L
T	Invalid	Tool function code			
U	Valid	Coordinate position data			
	Invalid	2nd miscellaneous function code			L
	Valid	Program tool compensation input			L
V	Valid	Coordinate position data			
	Invalid	2nd miscellaneous function code			L
	Valid	Program tool compensation input			L

4. POSITION COMMAND
4.3 Decimal Point Input I, II

Ad- dress	Decimal point command	Application	M/L	Remarks
W	Valid	Coordinate position data		
	Invalid	2nd miscellaneous function code	L	
	Valid	Program tool compensation input	L	
X	Valid	Coordinate position data		
	Valid	Dwell time		
	Invalid	2nd miscellaneous function code	L	
	Valid	Program tool compensation input	L	
Y	Valid	Coordinate position data		
	Invalid	2nd miscellaneous function code	L	
	Valid	Program tool compensation input	L	
Z	Valid	Coordinate position data		
	Invalid	2nd miscellaneous function code	L	
	Valid	Program tool compensation input	L	

4. POSITION COMMAND
4.4 Command Value and Setting Value Range

4.4 Command Value and Setting Value Range

(1) Outline

A command range (maximum/minimum) exists for the setting values from the screen, command values in the machining programs, etc. Commands outside of that range cause an error.

(2) Detailed description

(a) Details of the number of digits in words data

	Minimum setting unit		Command value and setting value range	
	Metric command	Inch command	Metric command	Inch command
Program number O	O8			
Sequence number N	N5			
Preparation function G	G3/G21			
Movement axis X, Y, Z, α	A: 0.01 B: 0.001 B': 0.001 C: 0.0001	A: 0.001 B: 0.0001 B': 0.0001 C: 0.00001	X+52 X+53 X+43 X+44	X+43 X+44 X+34 X+35
Auxiliary axis I, J, K	A: 0.01 B: 0.001 B': 0.001 C: 0.0001	A: 0.001 B: 0.0001 B': 0.0001 C: 0.00001	I+52 I+53 I+43 I+44	I+43 I+44 I+34 I+35
Arc center (radius) (mm), (inch) I, J, K, R	A: 0.01 B: 0.001 B': 0.001 C: 0.0001	A: 0.001 B: 0.0001 B': 0.0001 C: 0.00001	I+62 I+63 I+53 I+54	I+53 I+54 I+44 I+45
Feed function (feed per minute) F	A: 0.01 B: 0.001 B': 0.001 C: 0.0001	A: 0.001 B: 0.0001 B': 0.0001 C: 0.00001	0.01 to 480000.00 0.001 to 480000.000 0.001 to 100000.000 0.0001 to 100000.0000	0.001 to 18897.6378 0.0001 to 18897.6378 0.0001 to 3937.0078 0.00001 to 3937.00787
Feed function (feed per rotation) F	A: 0.01 B: 0.001 B': 0.001 C: 0.0001	A: 0.001 B: 0.0001 B': 0.0001 C: 0.00001	0.001 to 99999.999 0.0001 to 9999.9999 0.0001 to 999.9999 0.00001 to 999.99999	0.001 to 3937.007 0.0001 to 393.7007 0.0001 to 39.3700 0.00001 to 39.37007
Dwell	A: 0.01 (rev/s) B: 0.001 (rev/s) B': 0.001 (rev/s) C: 0.001 (rev/s)	A: 0.01 (rev/s) B: 0.001 (rev/s) B': 0.001 (rev/s) C: 0.001 (rev/s)	X53 P8 X53 P8 X53 P8 X53 P8	
Fixed cycle	A: 0.01 B: 0.001 B': 0.001 C: 0.0001	A: 0.001 B: 0.0001 B': 0.0001 C: 0.00001	R+52 Q52 P8 L4 R+53 Q53 P8 L4 R+43 Q43 P8 L4 R+44 Q44 P8 L4	R+43 Q43 P8 L4 R+44 Q44 P8 L4 R+34 Q34 P8 L4 R+35 Q35 P8 L4
Tool compensation number	H3/D3, T1/T2			
Miscellaneous function	M8			
Spindle function	S8			
Tool function	T8			
2nd miscellaneous function	B8/A8/C8			
Subprogram	P8H5L4			

B' is internal sub-micrometric system (unit of display/setting is one micron, and an internal unit of processing is 0.1 microns). Please inquire to the machine manufacturer whether the machine of your use is an internal sub-micrometric system.

4. POSITION COMMAND

4.4 Command Value and Setting Value Range

- (Note 1)** α indicates the additional axis address.
- (Note 2)** Except for the tool compensation numbers, all leading zeros can be omitted.
- (Note 3)** Program numbers are commanded in independent blocks. Command them in the head block of each program.
- (Note 4)** The significance of the symbols and abbreviations used are as follows.

Example 1) O8 : 8-digit program number

Example 2) G21 : Address G has two digits to the left of the decimal point, one digit to the right.

Example 3) X+53 : Address X has a "+" or "-" sign. It has five digits to the left of the decimal point, and three to the right.

The example given below is a case where the X axis is positioned (G00) at the 45.123mm position in the absolute value (G90) mode.

G00 X45.123 ;

-
- Three digits below decimal point
 - This denotes five digits above the decimal point. However, since the actual value is "+00045", the leading zeros and "+" sign are omitted.
 - G0 may be applicable instead.

5. PREPARATORY FUNCTIONS
5.1 List of G Commands

5. PREPARATORY FUNCTIONS

5.1 List of G Commands

G code	Group	Function	Refer- ence	G code	Group	Function	Refer- ence
00	01	Positioning	6.1	31.3	00	Multi-step skip 3	14.2
01	01	Linear interpolation	6.3	31.4	00	Multi-step skip 4	14.2
02	01	Circular interpolation CW	6.4	32		(Spare)	
03	01	Circular interpolation CCW	6.4	33	01	Thread cutting	6.10
02.1	01	Spiral interpolation CW	6.8	34	00	Special fixed cycle (bolt hole circle)	13.13
03.1	01	Spiral interpolation CCW	6.8	35	00	Special fixed cycle (line at angle)	13.13
04	00	Dwell	7.7	36	00	Special fixed cycle (arc)	13.13
05	00	High-speed mode	13.1	37.1	00	Automatic tool length measurement	14.3
06		(Spare)		37	00	Special fixed cycle (grid)	13.13
06.2	01	NURBS interpolation	6.9	38	00	Tool radius compensation vector designation	11.2
07	00	Hypothetical axis interpolation	6.5	39	00	Tool radius compensation corner arc	11.2
07.1	00	Cylindrical interpolation	6.6	40	07	Tool radius compensation cancel	11.2
08		(Spare)		41	07	Tool radius compensation Left compensation	11.2
09	00	Exact stop check		42	07	Tool radius compensation Right compensation	11.2
10	00	Program parameter input/compensation input	13.26	40.1	15	Normal line control cancel	13.16
11.1	00	Program parameter input cancel	13.25	41.1	15	Normal line control Left	13.16
12	00	Circle cutting CW	13.14	42.1	15	Normal line control Right	13.16
13	00	Circle cutting CCW	13.14	43	08	Tool length offset (+)	11.1
14		(Spare)		44	08	Tool length offset cancel	11.1
15		(Spare)		45	00	Tool position offset increase	11.6
16		(Spare)		46	00	Tool position offset decrease	11.6
17	02	Plane selection X-Y	12.8	47	00	Tool position offset 2-fold increase	11.6
18	02	Plane selection Z-X	12.8	48	00	Tool position offset 2-fold decrease	11.6
19	02	Plane selection Y-Z	12.8	49		(Spare)	
20	06	Inch command	4.2	50	11	Scaling cancel	13.15
21	06	Metric command	4.2	51	11	Scaling ON	13.15
22		Subprogram call/pattern rotation	13.2	50.1	19	Mirror image cancel	13.5
23		Subprogram return	13.2	51.1	19	Mirror image ON	13.5
22.1	04	Stroke check before movement ON	15.1	52	00	Local coordinate system setting	12.4
23.1	04	Stroke check before movement OFF	15.1	53	00	Machine coordinate system setting	12.2
26		(Spare)		54	12	Workpiece coordinate system 1 selection	12.3
27	00	Reference point verification	12.11	55	12	Workpiece coordinate system 2 selection	12.3
28	00	Reference point return	12.9	56	12	Workpiece coordinate system 3 selection	12.3
29	00	Start point return	12.9	57	12	Workpiece coordinate system 4 selection	12.3
30	00	2nd to 4th reference point return	12.10	58	12	Workpiece coordinate system 5 selection	12.3
31	00	Skip	14.1	59	12	Workpiece coordinate system 6 selection	12.3
31.1	00	Multi-step skip 1	14.2	54.1	12	Workpiece coordinate system (expansion) selection	12.3
31.2	00	Multi-step skip 2	14.2	60	00	Unidirectional positioning	6.2

5. PREPARATORY FUNCTIONS

5.1 List of G Commands

G code	Group	Function	Reference	G code	Group	Function	Reference
61	13	Exact stop check mode		88	09	Fixed cycle (boring)	13.3
61.1	13	High-accuracy control mode	13.22	89	09	Fixed cycle (boring)	13.3
61.2	13	High-accuracy control for die machining (CAM data)	13.23	90	03	Absolute value command	4.1
61.3	13	High-accuracy control for die machining (height data)	13.23	91	03	Incremental value command	4.1
61.5	13	High-accuracy control for part machining	13.24	92	00	Machine coordinate system setting	12.6
62	13	G command mirror image	13.5	93	05	Inverse time feed	7.5
63	13	Tapping mode	13.21	94	05	Asynchronous feed (feed per minute)	
64	13	Cutting mode	13.19	95	05	Synchronous feed (feed per rotation)	
65	00	User macro Simple call	13.7	96	17	Constant surface speed control ON	
66	14	User macro Modal call A	13.7	96.1	17	Rotation axis constant tangential velocity control	
66.1	14	User macro Modal call B	13.7	97	17	Constant surface speed control cancel	
67	14	User macro Modal call cancel	13.7	98	10	Fixed cycle Initial level return	13.3
68		User macro Subprogram end	13.7	99	10	Fixed cycle R point level return	13.3
69	00	End point error check clear	13.29				
68.1	16	Coordinate rotation ON	13.11				
69.1	16	Coordinate rotation cancel	13.11				
70		User fixed cycle					
71		User fixed cycle					
72		User fixed cycle					
73	09	Fixed cycle (step)	13.3				
74	09	Fixed cycle (reverse step)	13.3				
75		User fixed cycle					
76	09	Fixed cycle (fine boring)	13.3				
77		User fixed cycle					
78		User fixed cycle					
79		User fixed cycle					
80	09	Fixed cycle cancel	13.3				
81	09	Fixed cycle (drill/spot drill)	13.3				
82	09	Fixed cycle (drill/counter-boring)	13.3				
83	09	Fixed cycle (deep hole drilling)	13.3				
84	09	Fixed cycle (tap)	13.3				
85	09	Fixed cycle (boring)	13.3				
86	09	Fixed cycle (boring)	13.3				
87	09	Fixed cycle (back boring)	13.3				

6. POSITIONING AND INTERPOLATION FUNCTIONS

6.1 Positioning; G00

(1) Outline

This command is a function that sets the current point as the start point, and carries out high-speed positioning commanded with the movement command value in the program at the rapid traverse rate.

(2) Command format

G00 Xx1 Yy1 Zz1 ; (simultaneously with additional axis A, B, C, U, V and W also possible)
(x1, y1, z1 are numerical values indicating position data)

Positioning operation at the rapid traverse rate is carried out by the commands above. The path takes the shortest distance in a straight line toward the end point.

Refer to the section on "Rapid traverse rate" for details on the NC rapid traverse rate.

Because the actual rapid traverse rate differs according to the machine, refer to the machine specifications.

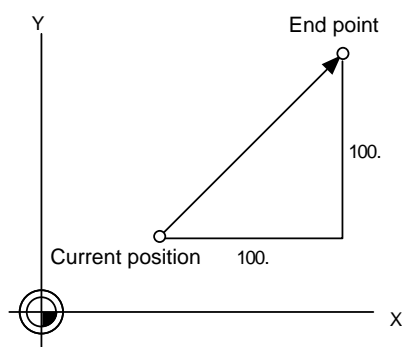
(3) Detailed description

- (a) The rapid traverse rate can be set independently for each axis with the parameters.
- (b) The number of axes that can be simultaneously driven depends on the specifications (number of simultaneous control axes). Axis combination within that range is random.
- (c) Once this command has been issued, the G00 mode is retained until it is changed by another G function (the G01, G02 or G03, etc.) in the 01 group is issued. If the next command is G00, all that is required is simply to specify the coordinate words.
- (d) Acceleration and deceleration are constantly carried out at the start and end of the program command block. The in-position status is confirmed before advancing to the next block. Set the in-position width in the parameters.
- (e) Any G function (G72 to G89) in the 09 group is cancelled (G80) by the G00 command.
- (f) The feedrate is controlled to the shortest time, in a range not exceeding the rapid traverse rate of each axis. (Interpolation type)
The axis can also be moved at the respective rapid traverse rate of each independent axis, using parameter settings. In this case, the path is not a straight line toward the end point. (Non-interpolation type)
The positioning time does not change in either case.
- (g) When no number following the G address, this is treated as G00.

(Example)

Interpolation type (movement in a straight line to the end point)

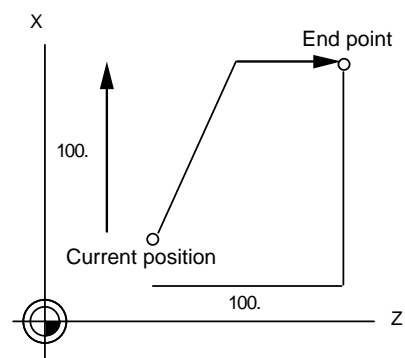
G00 G91 X100. Y100. ;



(Example)

Non-interpolation type (each axis moves at the rate of the respective parameter)

G00 G91 X100. Y100. ;



6. POSITIONING AND INTERPOLATION FUNCTIONS

6.2 Unidirectional Positioning; G60

6.2 Unidirectional Positioning; G60

(1) Outline

Final positioning is always carried out by the G60 command in the one direction specified in the parameters.

Positioning excluding backlash can be carried out.

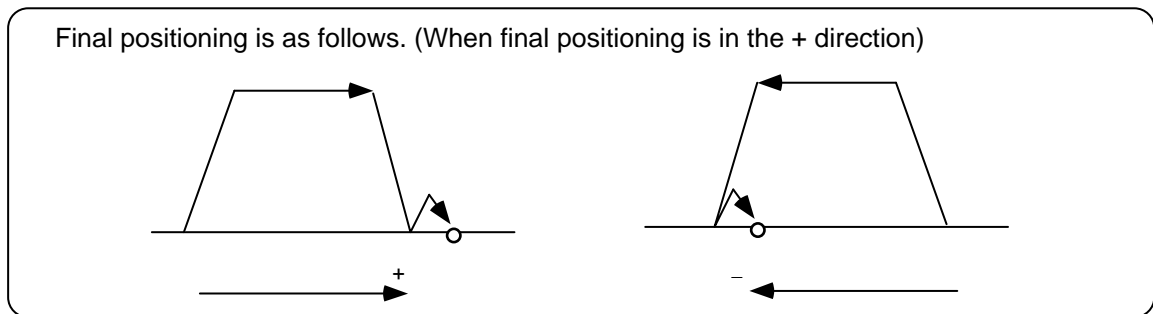
(2) Command format

G60 Xx1 Yy1 Zz1 ; (simultaneously with additional axis A, B, C, U, V and W also possible) (x1, y1, z1 are numerical values indicating position data)

With the above command, the tool moves by rapid traverse rate from the final position to a position separated only by the unidirectional positioning shift amount (parameter setting), and then moves further to the final position and completes the positioning.

Refer to the section on "Rapid traverse rate" for details on the NC rapid traverse rate.

Because the actual rapid traverse rate differs according to the machine, refer to the machine specifications.



(3) Detailed description

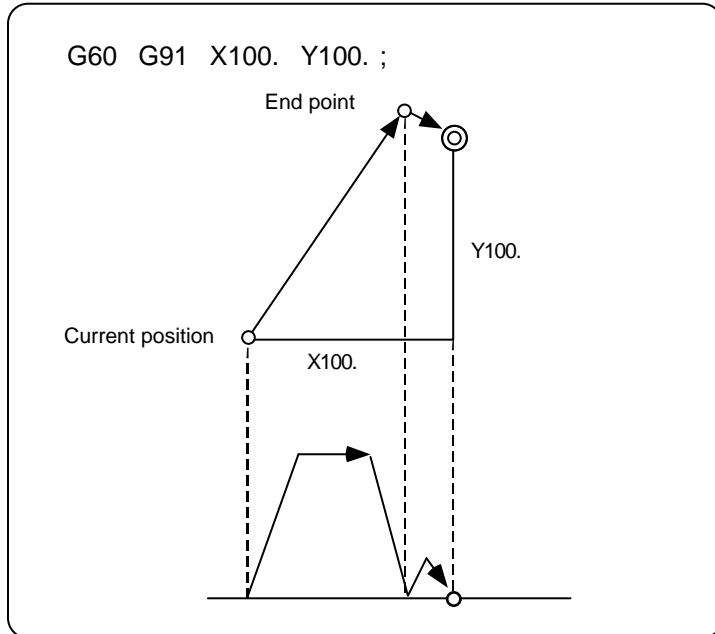
- The unidirectional positioning shift amount for the final positioning as well as the final positioning direction is set in parameters.
- After the tool has moved at the rapid traverse rate to the position separated from the final position by an amount equivalent to the unidirectional positioning shift amount, it moves to the final position in accordance with the rapid traverse setting where its positioning is completed.
- The above positioning operation is performed even when Z-axis commands have been designated for Z-axis cancel and machine lock. (Display only)
- When the mirror image function is ON, the tool will move in the opposite direction as far as the middle position due to the mirror image function but the unidirectional positioning shift amount operation during its final advance will not be affected by mirror image.
- The tool moves to the end point at the dry run speed during dry run when the G0 dry run function is valid.
- Feed during unidirectional positioning shift amount movement with final positioning can be stopped by resetting, emergency stop, interlock, feed hold and rapid traverse override zero.
- The tool moves over the unidirectional positioning shift amount at the rapid traverse setting. Rapid traverse override is valid.
- Unidirectional positioning is not performed for the hole drilling axis during hole drilling fixed cycles.
- Unidirectional positioning is not performed for shift amount movements during the fine boring or back boring fixed cycle.
- Normal positioning is performed for axes whose unidirectional positioning shift amount has not been set by parameters.
- Unidirectional positioning is always a non-interpolation type of positioning.
- When the same position (movement amount of zero) has been commanded, the tool moves back and forth over the unidirectional positioning shift amount and is positioned at its original position from the final advance direction.

6. POSITIONING AND INTERPOLATION FUNCTIONS

6.2 Unidirectional Positioning; G60

(m) During high-accuracy control, the unidirectional positioning operation is also carried out with acceleration/deceleration before interpolation.

(Example)



1. Each of the axis rapid traverse rates is a value set in the parameters as the G00 speed.
2. The vector speed to the middle point is a value composed of the respective speeds and distances.
3. The unidirectional positioning shift amount of the distance between the middle point and end point is set independently for each axis in the parameters.

(4) Notes

- (a) If G60 is commanded in an NC not provided with this specification, it is regarded as the reference point return command. Refer to the section of "Automatic 1st Reference Point Return; G28/G29" for details of the reference point return.

6. POSITIONING AND INTERPOLATION FUNCTIONS
6.3 Linear Interpolation; G01

6.3 Linear Interpolation; G01

(1) Outline

This command is accompanied by coordinate words and a feedrate command. It makes the tool move (interpolate) linearly from its current position to the end point specified by the coordinate words at the cutting feedrate specified by address F. In this case, the feedrate specified by address F always acts as a linear speed in the tool nose center advance direction.

(2) Command format

G01 Xx1 Yy1 Zz1 Ff1 ; (simultaneously with additional axis A, B, C, U, V and W also possible)
(x1, y1, z1 are numerical values indicating position data, f1 is a numerical value indicating speed data.)

Linear interpolation is carried out by the commands above at the f1 speed. The path takes the shortest distance in a straight line toward the end point.

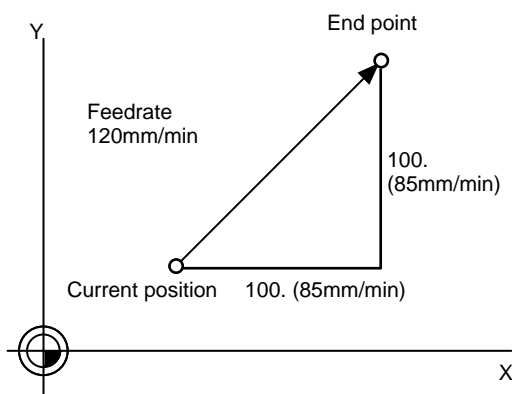
The NC f1 command value is a maximum 480m/min (when the input unit is 0.001mm), however this will differ according to the machine, so refer to the machine specifications.

(3) Detailed description

- (a) Once this command is issued, this mode is maintained until another G function (G00, G02, G03, G33, etc.) in the 01 group which changes the G01 mode is issued. Therefore, if the next command is also G01 and if the feedrate is the same, all that is required is to specify the coordinate words.
- (b) The feedrate of the rotation axis is commanded in °/min (decimal point position unit) or °/rev (per spindle rotation). (F300 = 300 °/min)
- (c) The G functions (G72 to G89) in the 09 group are cancelled (G80) by the G01 command.

(Example)

G01 G91 X100. Y100. F120 ;



- 1. The cutting feedrate becomes the command to the vector direction.
- 2. The composite speed of each axis becomes the proportional allocation of the respective command values.

- (d) The number of axes that can be simultaneously commanded depends on the specifications (number of simultaneous control axes). Axis combination within that range is random.
- (e) The feedrate is controlled so as not to exceed the cutting feedrate clamp.
If linear axis and rotation axis are commanded in the same block, this is regarded as a linear axis using degree (°) (1 ° = 1mm) units, and linear interpolation is carried out.

(4) Notes

- (a) The program error (P62) will occur if an F command is not issued in the first G01 command.

6.4 Circular Interpolation (designating center/radius); G02/G03

(1) Outline

(a) I, J and K designation circular interpolation

Using the movement command value issued in the program, this function moves the tool in an arc on the plane selected by the plane selection G code.

(b) R-designated circular interpolation

In addition to the arc center coordinate designation by the I, J and K of the previous item, this function can also issue arc commands by directly designating the arc radius.

(2) Command format

(a) I, J and K designation circular interpolation

<u>G02/G03</u>	<u>Xx1 Yy1</u>	<u>Ii1 Jj1</u>	<u>Ff1</u> ; (additional axis A, B, C, U, V and W possible)
Arc rotation direction	End point coordinate value	Arc center value	Feedrate

(b) R-designated circular interpolation

In addition to the arc center coordinate designation by the I, J and K of the previous item, this function can also issue arc commands by directly designating the arc radius.

<u>G02/G03</u>	<u>Xx1 Yy1</u>	<u>Rr1</u>	<u>Ff1</u> ; (additional axis A, B, C, U, V and W possible)
Arc rotation direction	End point coordinate value	Arc radius	Feedrate

(3) Detailed description

(a) I, J and K designation circular interpolation

- 1) G02 (or G03) is held until another G command (G00, G01 or G33) in the 01 group that changes this mode is issued.
- 2) Circular interpolation operations are carried out at the f1 rate by the commands above. The path is toward the end point, following an arc path centered at the position designated by distance i (X axis direction) and distance j (Y axis direction) in respect to the start point.
- 3) The following information is necessary to carry out circular interpolation.

a. Plane selection

The arc plane is designated by G17, G18 and G19.

- G17 XY plane
- G18 ZX plane
- G19 YZ plane

The arc plane always follows G17, G18 and G19.

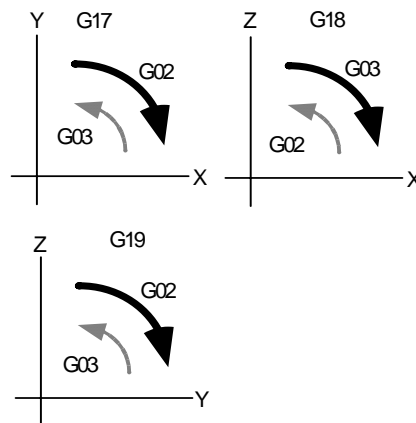
An alarm occurs if two addresses that do not match the plane are commanded.

The axes that configure the plane can be designated in the parameters. Refer to the section on plane selection.

b. Rotation direction

The arc rotation direction is designated by G02 or G03.

- G02 Clockwise (CW)
- G03 Counterclockwise (CCW)



6. POSITIONING AND INTERPOLATION FUNCTIONS
6.4 Circular Interpolation (designating center/radius); G02/G03

c. Arc end point coordinates

Issued by addresses X, Y and Z. Absolute value and incremental value combined use possible.

d. Arc center coordinates

Issued by addresses I, J and K. Absolute value and incremental value combined use possible.

It is best to command with decimal points attached.

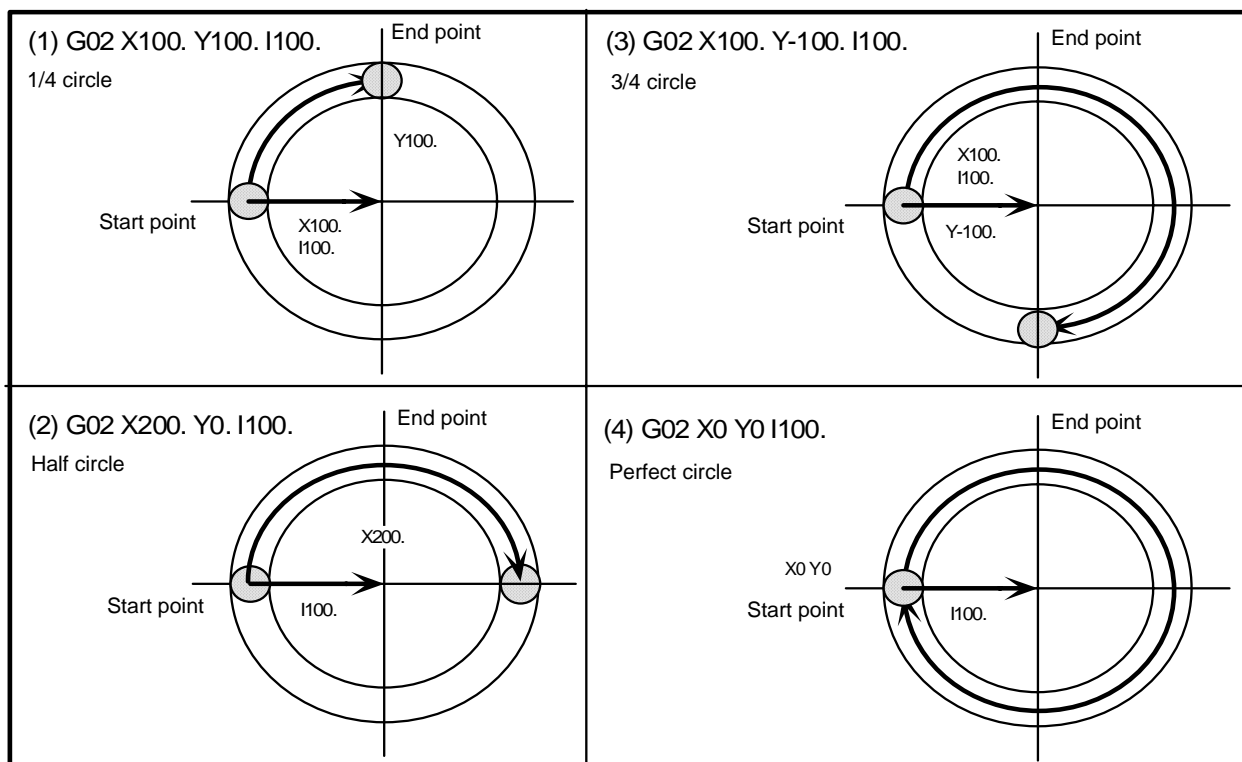
e. Feedrate

Issued by address F.

4) Notes related to circular interpolation

- a. In arc operations, clockwise (G02) and counterclockwise (G03) are defined as: In a right-hand coordinate system, looking in the minus direction from the plus direction of the coordinate axis that is perpendicular to the target plane.
- b. If the end point coordinates are all omitted, or the end point and start point are in the same position, commanding the center using I, J and K is the same as commanding a 360° arc (perfect circle).
- c. When the R command, and the I, J and K commands are omitted from the arc block, the program error "PR0071 Can't calculate arc center" will occur.
By setting the parameter (#110136 CIR to G1 no CENT OP), linear interpolation can be applied up to only the end point coordinate value of that block. Note that the modal is an arc modal.

5) **(Example)** Examples of a 1/4 circle, half circle, 3/4 circle and perfect circle are given.



- a. The axes that can be simultaneously commanded are the two axes of the selected plane.
- b. The feedrate is constantly controlled to be a rate that follows the circumference.
- c. Circular interpolation can be commanded within a 360° rotation.
- d. The maximum radius value can be designated up to six digits above the decimal point.

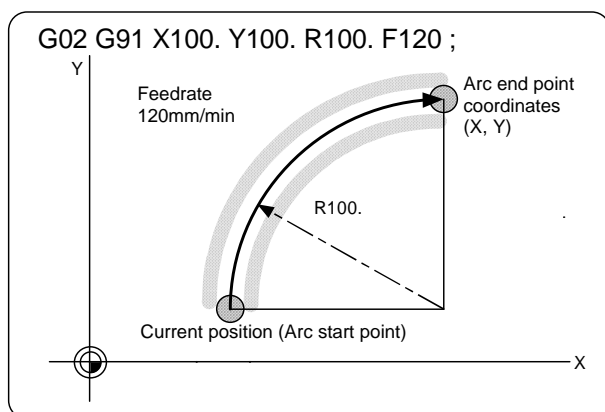
6. POSITIONING AND INTERPOLATION FUNCTIONS

6.4 Circular Interpolation (designating center/radius); G02/G03

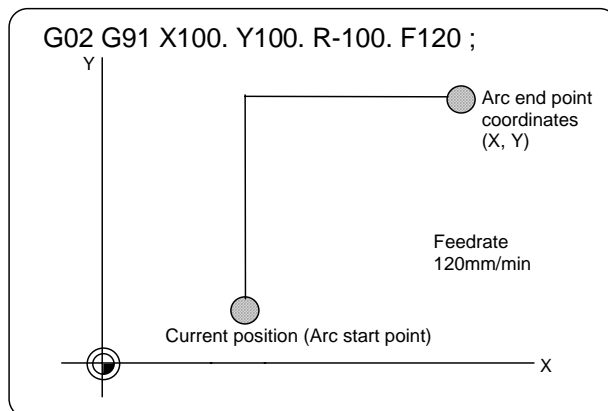
(b) R-designated circular interpolation

- 1) G02 (or G03) is held until another G command (G00, G01 or G33) in the 01 group that changes this mode is issued.
- 2)
 - a. The arc rotation direction is designated by G02 or G03.
 - b. The arc plane is designated by G17, G18 and G19.
The arc plane always follows G17, G18 and G19. The alarm (PR0070) occurs if two addresses that do not match the plane are commanded.
 - c. The arc center is on the perpendicular line that bisects the line segment connecting the start and end points. The intersection of the arc center and the designated radius circle that has the start point as its center is the center coordinate of the arc command.
 - d. The symbol of the command program R value is an arc command of 180° or less for the plus direction, and an arc command exceeding 180° for the minus direction.

(Example 1)



(Example 2)



- The axes that can be simultaneously commanded are the two axes of the selected plane.
- The feedrate is constantly controlled to be a speed that follows the circumference.
- The following conditions are necessary in R-designated circular interpolation commands.

$$L/2 \leq r$$

The alarm (PR0071) occurs if $L/2 - r > (\text{parameter: Arc error})$.

L: Straight line segment from the start point to the end point

3) Notes

- a. When R-designation and I, J and K designation are simultaneously designated in the same block, arc command by the R designation is given priority.
- b. For perfect circle commands (start point and end point match), if an R-designated arc command is issued, it is instantly completed and there is no operation. Therefore, use I, J and K designation arc commands.
- c. When the R command, and the I, J and K commands are omitted from the arc block, the program error "PR0071 Can't calculate arc center" will occur.
By setting the parameter (#110136 CIR to G1 no CENT OP), linear interpolation can be applied up to only the end point coordinate of that block. Note that the modal is an arc modal.

6.5 Hypothetical Axis Interpolation; G07

(1) Outline

By having one of the circular interpolation axes with spiral interpolation or with helical interpolation including the linear axis as the hypothetical axis (the axis which does not actually move) and by distributing the pulses, it is possible to provide interpolation (sine or cosine interpolation) in which the helical or spiral interpolation is viewed from the side (hypothetical axis). G07 is the command which sets the hypothetical axis.

(2) Command format

G07 X0 ; (X axis command cancel ON)
G07 X1 ; (X axis command cancel OFF)

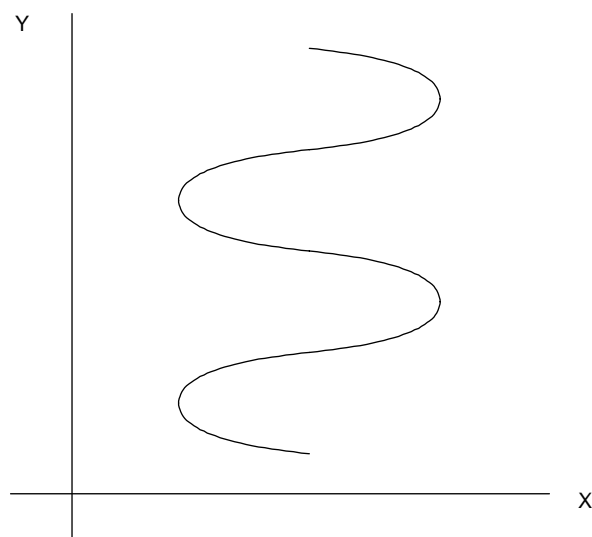
The designation of the axis to carry out axis command cancel is applicable to all of the NC axes.

(3) Detailed description

- Helical interpolation and spiral interpolation are the interpolation functions which can be used for hypothetical axis interpolation.
- The " α " axis in the "G07 α 0 ; to G07 α 1 ;" program commands is the hypothetical axis. Consequently, when the " α " axis is commanded separately between these program commands, the dwell state is established until the pulse distribution to the hypothetical axis is completed.
- The hypothetical axis is valid only with automatic operation; it is not valid with manual operation.
- The protection functions (such as interlock and stored stroke limit) are also valid for the hypothetical axis.
- Handle interrupt is valid for the hypothetical axis as well. The axis will move by an amount equivalent to the handle interrupt.

(f)

Example G07 Z0 ;
 G18 G02 X50. Z0. Y100. K30. P3 ;



To carry out hypothetical axis interpolation, it is necessary to combine and add helical interpolation.

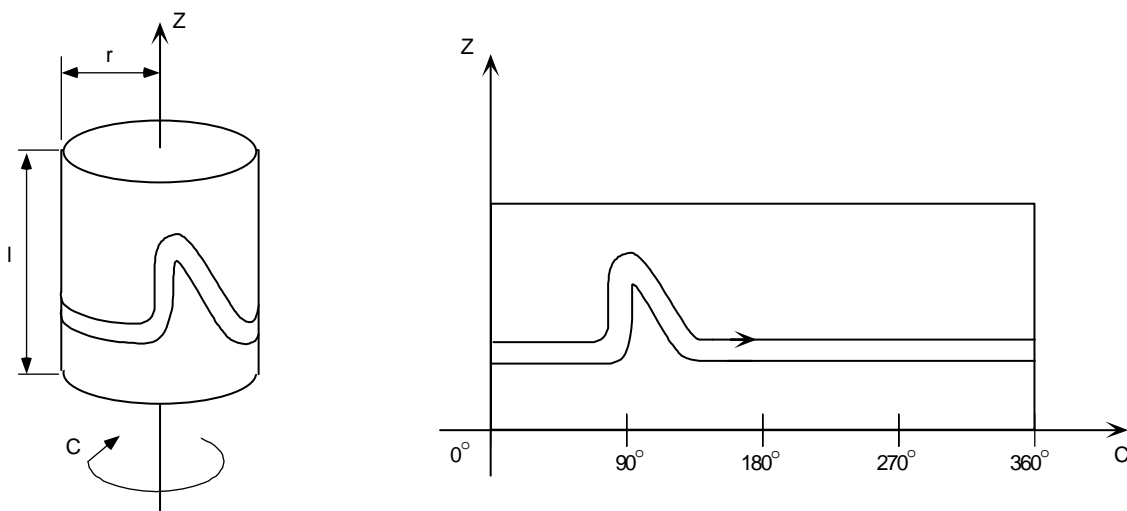
6.6 Cylindrical Interpolation; G07.1

(1) Outline

The cylindrical interpolation function is for machining the side face of a cylindrical workpiece. Because programming can be carried out on the plane developed on the cylinder side face, cylinder groove machining programs can be easily created.

(2) Command format

G07.1	Cc1	; Cylinder interpolation mode ON
G07.1	C0	; Cylinder interpolation mode cancel
C is the rotation axis name, c1 is the cylinder radius.		



The rotation axis movement amount commanded in angle is converted internally in the CNC to the linear axis distance on the circumference. After conversion, linear and cylindrical interpolation are carried out between the other axes. After interpolation, the linear axis distance is converted back into the rotation axis movement amount.

(3) Detailed description

- Cylindrical interpolation commands cannot be issued in the same block as other G commands. (An error will occur).
- The direction of the rotation axis can be set in the parameters.
- Feedrate designation
In the cylindrical interpolation mode, the commanded feedrate is the rate of the axis on the plane that developed the cylinder side face.
- Clamp speed designation
During the cylindrical interpolation mode, this is the speed on the cylindrical expansion plane proportional to the angle speed.

6. POSITIONING AND INTERPOLATION FUNCTIONS

6.6 Cylindrical Interpolation; G07.1

(e) Circular interpolation (G02, G03)

1) Plane selection

The plane selection setting will not return to the previous setting even if the cylindrical interpolation mode is canceled.

(Example) To carry out circular interpolation with Z axis and C axis

- Set the C axis for the machining parameter parallel axis 1I (X axis parallel axis).

The circular interpolation command at this time is as follows:

```
G18 Z0 C0;  
G02/03 Z25 C90 R10;
```

- The C axis can also be used for the machining parameter parallel axis 1J (Y axis parallel axis).

Note that in this case, the circular interpolation command is as follows:

```
G19 C0 Z0;  
G02/03 Z25 C90 R10;
```

2) Radius designation

The radius command can be designated with the R designation or the I, J designation.

Note that the R unit and the rotation axis side plane I, J unit must be designated as the length on the expansion plane.

(f) Radius compensation

To carry out radius compensation in the cylindrical interpolation mode, command the plane selection in the same manner as circular interpolation.

However, carry out startup and cancel in the cylindrical interpolation mode when carrying out radius compensation.

If the radius compensation is valid, correct compensation will not be carried out when the cylindrical interpolation mode is set.

(g) Positioning

An error will occur if positioning (including commands that create rapid traverse cycles, such as G28 and G80 to G89) is carried out in the cylindrical interpolation mode. Note that only G0 rapid traverse is possible.

Set the mode to cylindrical interpolation cancel mode when positioning.

(h) Coordinate system setting

The workpiece coordinate system (G92, G54 to G59) and local coordinate system (G52) cannot be commanded in the cylindrical interpolation mode.

(i) The cylindrical interpolation (G07.1) cannot be commanded in the positioning mode (G00).

(j) The fixed cycle cannot be commanded in the cylindrical interpolation mode.

(k) Rotation axis control parameters

Only the rotary-type rotation axis can be designated as the rotation axis for cylindrical interpolation.

(l) Inch inputs are not possible.

When initial inch is set, the cylindrical interpolation mode will not be entered. (It will be ignored.)

(m) Restart search

When restart search to a block in the cylindrical interpolation mode is carried out, the error "OPW0133 Restart Search Error \$1" will occur.

(n) Reset modal retaining

The modal information of the cylindrical interpolation mode is not retained at NC reset. Even if resetting is applied in the cylindrical interpolation mode with retaining modal at NC reset valid, the cylindrical interpolation mode will turn OFF.

6. POSITIONING AND INTERPOLATION FUNCTIONS

6.6 Cylindrical Interpolation; G07.1

(4) Notes

- (a) The cylindrical interpolation mode cannot be set again during the cylindrical interpolation mode. To set the mode again, cancel the cylindrical interpolation mode once.
- (b) One rotation axis can be simultaneously interpolated. Thus, two or more rotation axes cannot be commanded with the G07.1 command.
- (c) During the cylindrical interpolation mode, the movement amount of the rotation axis commanded as an angle is converted into a distance on the circumference once. Then, after carrying out the linear and circular interpolation operations between the other axes, the amount is reconverted into an angle.
Thus, if the cylinder radius is small, etc., the actual movement amount may differ from the commanded value. Note that the error at this time is not cumulated.

$$\text{Actual movement amount} = \left\{ \frac{\text{MOVE}}{2 \times 2\pi} \times \left(\text{Command value} \times \frac{2 \times 2\pi}{\text{MOVE}} \right) \right\}$$

MOVE : Movement amount per rotation axis rotation (parameter)

r : Workpiece radius

() : Rounded off with minimum setting unit

6. POSITIONING AND INTERPOLATION FUNCTIONS
6.7 Helical Interpolation; G02/G03

6.7 Helical Interpolation; G02/G03

(1) Outline

With orthogonal three axes, it is possible for the 3rd axis to carry out linear interpolation synchronized with the rotation of the arc, while the circular interpolation is simultaneously carried out by the G02/G03 commands on the plane selected by the plane selection G code (G17, G18, G19).

(2) Command format

<u>G17</u>	<u>G02/G03</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Zz1</u>	<u>Ii1</u>	<u>Jj1</u>	<u>Pp1</u>	<u>Ff1</u> ;
Arc plane	Arc rotation direction	Arc end point coordinates		Linear axis end point	Arc center coordinates		Number of pitches	Feedrate

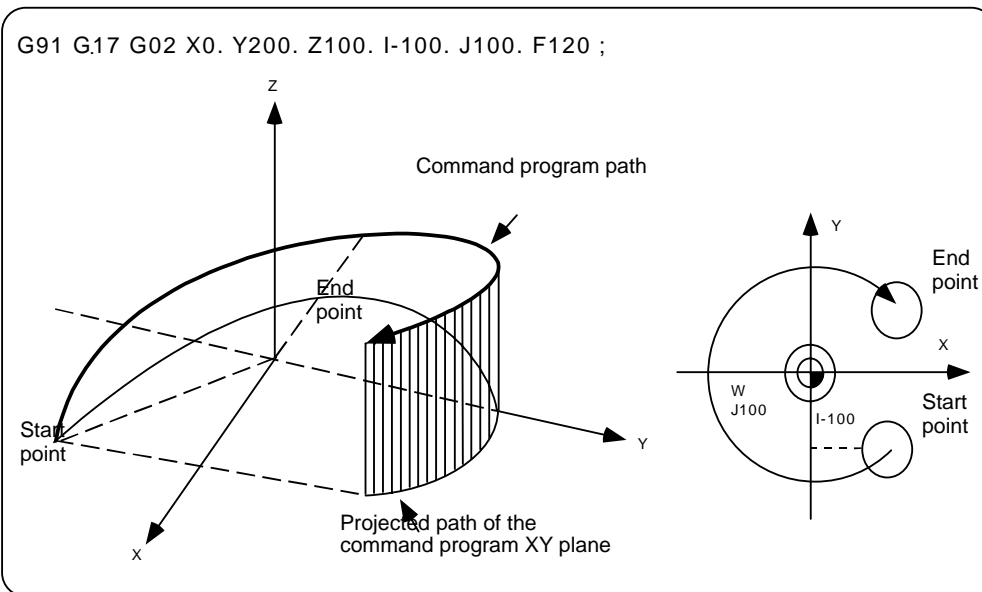
Or

<u>G17</u>	<u>G02/G03</u>	<u>Xx2</u>	<u>Yy2</u>	<u>Zz2</u>	<u>Rr2</u>	<u>Ff2</u> ;
Arc plane	Arc rotation direction	Arc end point coordinates		Linear axis end point	Arc radius	Feedrate

The arc center coordinates and radius should be commanded with a decimal point.

(3) Detailed description

(Example)



- (a) Command a linear axis (or a multiple number of axes) not including the arc axis for the circular interpolation command.
- (b) Command the combined speed of the X, Y and Z axis components for the feedrate F.
- (c) Pitch L is determined from the following formula:

$$L = z1 / ((2\pi \cdot p1 + \theta) / 2\pi)$$

$$\theta = \theta_e - \theta_s = \arctan(y_e/x_e) - \arctan(y_s/x_s) \quad (0 \leq \theta < 2\pi)$$

Where, x_s and y_s are the coordinates of the start point
 x_e and y_e are the coordinates of the end point

- (d) Address P may be omitted when the pitch number is 0.
- (e) The command range for pitch number P is 0 to 9999.
- (f) The pitch number (P command) cannot be designated using an R-designated arc.

6. POSITIONING AND INTERPOLATION FUNCTIONS

6.7 Helical Interpolation; G02/G03

(g) Plane selection

The selection of the arc plane for helical interpolation is determined by the plane selection mode and axis addresses, as is the case for circular interpolation. The basic method of commanding helical interpolation is to command an circular interpolation plane with a plane selection G code (G17, G18, G19) and to command two axes for circular interpolation and the addresses of the 3 axes for linear interpolation (axis perpendicular to the arc plane).

- XY plane arc, Z axis line
Command the addresses of the 3 axes X, Y and Z in the G02 (G03) mode and G17 (plane selection G code) mode.
- ZX plane arc, Y axis line
Command the addresses of the 3 axes Z, X and Y in the G02 (G03) mode and G18 (plane selection G code) mode.
- YZ plane arc, X axis line
Command the addresses of the 3 axes Y, Z and X in the G02 (G03) mode and G19 (plane selection G code) mode.
Plane selection of an additional axis can also be carried out, as is the case for circular interpolation.
- UY plane arc, Z axis line
Command the addresses of the 3 axes U, Y and Z in the G02 (G03) mode and G17 (plane selection G code) mode.

In addition to the above basic command methods, the command methods given after the following program examples are available. Refer to the section on "Plane selection" for the arc planes selected with these command methods.

- (h) The axis combination that can be simultaneously commanded depends on the specifications. The combination within that range is random.
- (i) The feedrate is constantly controlled to be a speed that follows the circumference.

6. POSITIONING AND INTERPOLATION FUNCTIONS
6.8 Spiral Interpolation; G02.1/G03.1

6.8 Spiral Interpolation; G02.1/G03.1

(1) Outline

This function carries out interpolation that smoothly joins the start and end points in a spiral. This interpolation is carried out for arc commands such as those in which the start point and end point are not on the same circumference.

(2) Command format

<u>G17</u>	<u>G02.1/G03.1</u>	<u>Xx1</u> <u>Yy1</u>	<u>li1</u> <u>Jj1</u>	<u>Pp1</u>	<u>Ff1</u> ;
Rotation plane	Arc rotation direction	End point coordinates	Arc center value	Number of pitches	Feedrate

Circular interpolation operations are carried out at the f1 speed by the commands above. The path is toward the end point, following a spiral arc path centered at the position designated by distance i (X axis direction) and distance j (Y axis direction) in respect to the start point. The arc plane is designated by G17, G18 and G19.

- G17 XY plane
- G18 ZX plane
- G19 YZ plane

The arc rotation direction is designated by G02.1 or G03.1.

- G02.1 . . . Clockwise (CW)
- G03.1 . . . Counterclockwise (CCW)

P: This designates the number of pitches (rotation speed).
 p1 = 0 is less than one rotation and can be omitted.
 p1 = 1 is more than one rotation and less than two rotations.
 The command range is from 0 to 9999.

F: Tool path direction speed

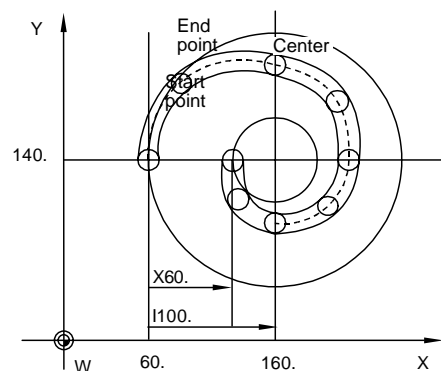
(Note) The spiral interpolation function can be specified by G69G02 or G69G03 in the M2/M0 format. For the details, see "End Point Error Check Clear".

(3) Detailed description

- (a) The arc rotation direction G02.1 is the same as G02, and G03.1 is the same as G03.
- (b) There are no R-designated arcs in spiral interpolation.
- (c) Conical cutting, tapered thread-cutting and other such machining operations can be conducted by changing the start point and end point radius and commanding the linear axis simultaneously.
- (d) Normally the spiral interpolation is automatically enabled with the arc commands (G02, G03) when the difference between the start point radius and the end point radius is less than the parameter setting value.

(e) (Example)

```
G91 G17 G01 X60. F500 ;
Y140. ;
G2.1 X60. Y0 I100. P1 F300 ;
G01 X-120 ;
G90
G17 G01 X60. F500 ;
Y140. ;
G2.1 X120. Y140. I100. P1 F300 ;
G01 X0 ;
```



- (f) The axis combination that can be simultaneously commanded depends on the specifications. The combination within that range is random.
- (g) The feedrate is the constant tangential speed.

6. POSITIONING AND INTERPOLATION FUNCTIONS

6.8 Spiral Interpolation; G02.1/G03.1

- (h) Simultaneous control by combining with tool radius compensation (G41, G42) is not possible.
- (i) The arc plane always follows G17, G18 and G19. The plane arc control is carried out by G17, G18 and G19, even if designated by two addresses that do not match the plane.

(Example)

G91 G17 G01 X60. F500 ;

Y140. ;

G2.1 X60.0 Z100.0 I100. P1 F300 ;

G01 X-120 ;

→ Because this is the G17 plane, arc control is not carried out by X-Z.

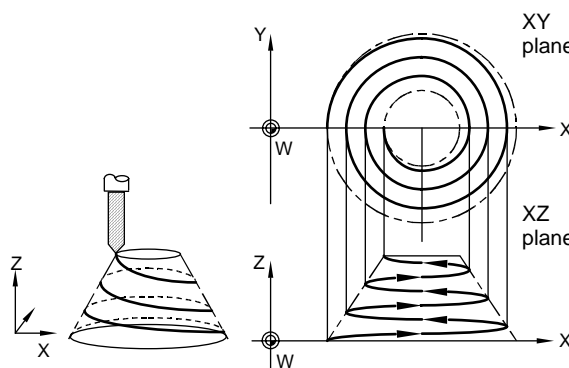
→ Arc control is carried out by X-Y.

- (j) When an axis designation other than the spiral interpolation plane is simultaneously designated, other axes are also interpolated in synchronization with the spiral interpolation.

(Example)

G17 G91 G2.1 X100. Z150. I150. P3 F500 ;

In these examples, the interpolation is truncated cone interpolation.



6. POSITIONING AND INTERPOLATION FUNCTIONS

6.9 NURBS Interpolation; G06.2

6.9 NURBS Interpolation; G06.2

(1) Outline

NURBS interpolation is an interpolation function that operates the command output in the NURBS format from the CAD/CAM directly with the NURBS definition in the CNC.

Directly exponential interpolation of a free line defined with the NURBS format in the CAD/CAM is possible without developing the curve into fine segments, so a very smooth interpolation path is achieved.

The shape of the NURBS curve can be flexibly controlled with the number of stages, control point, weight and knot vector. The shape can be freely edited by changing the control point position, changing the weight, and by changing the relative interval between each knot of the knot vector. However, generally manual editing is usually impossible, and a dedicated CAD/CAM system, etc., must be used to edit the NURBS curve or generate the NURBS interpolation program.

Manually editing the NURBS interpolation program generated with a CAD/CAM system, etc., is not recommended.

(2) Command format

<u>G06.2</u>	<u>Pp</u>	<u>Kk</u>	<u>Xx Yy Zz</u>	<u>Rr</u>	<u>Ff ;</u>
NURBS interpolation ON	Number of stages (can be omitted)	Knot	Control point coordinate value	Control point weight (can be omitted)	Interpolation speed (can be omitted)
		Kk	Xx Yy Zz		
			:		
			:		

The NURBS interpolation mode is entered with the G06.2 command. The number of stages, the control point coordinate value, the control point weight and the knot to determine the NURBS curve shape are commanded in succession.

(3) Detailed description

- G06.2 belongs to the G code group 1. The G06.2 mode is entered when NURBS interpolation is commanded. When turning the NURBS interpolation OFF (canceling the interpolation), and not issuing a NURBS interpolation command in succession, issue another group 1 command (G0, G1, etc.). The NURBS interpolation mode is not held when either initializing or retaining the modal at NC reset. (Initialization is carried out following the control parameter #320004 Initial G00.)
- When turning NURBS interpolation ON immediately after turning NURBS interpolation OFF, the G06.2 code can be omitted. Note that in this case, the number of stages P and interpolation speed F cannot be commanded.
- P indicates the number of stages in the NURBS curve. When the number of stages is m, NURBS curve is a (m-1) dimension curve. The P setting value is 2, 3, or 4. Any other value will cause an error. If the setting is omitted, the default value 4 will be set. Designate the number of stages in the first block (same block as G06.2 command).
The NURBS curve will be as follow according to the value set for the number of stages P.
 - P2 ... Linear (one-dimension curve)
 - P3 ... Two-dimension curve
 - P4 ... Three-dimension curve
- X, Y and Z indicate the coordinate values of the control point, and R indicates the weight of the control point. The weight is unique to each control point, and is commanded in the same block as the control point coordinate values. If the weight is omitted, the default value 1.0 will be set. When a large weight is set, the NURBS curve will be generated closer to the control point, and when a small weight is set, the NURBS curve will be farther from the control point.

6. POSITIONING AND INTERPOLATION FUNCTIONS
6.9 NURBS Interpolation; G06.2

- (e) K indicates the knots. When the a number of stages is m and the number of control points is n, the NURBS curve requires (n+m) knots. The arrangement of these (n+m) knots is called the knot vector. To command the knot vector, command the first knot simultaneously with the first control point, the second knot simultaneously with the second control point and following as a pair in one block. After commanding the blocks for the number of control points (n), command the remaining m knots in the independent blocks respectively. The number of control points is identified by these blocks containing only knots, so the control point does not need to be commanded. NURBS interpolation is completed when the blocks containing only knots are commanded for the number of stages.
- (f) The command range of the number of stages, weight and knots is as follows.

Meaning	Address	Range	Number of digits valid after decimal point
Number of stages	P	2, 3, 4	0
Weight	R	0.0001 to 99.9999	4
Knot	K	-214748 to 214748	4

(4) Notes

- (a) NURBS interpolation cannot be commanded for a rotary-type rotation axis. An error will occur if commanded.
- (b) Command the control point for the axis (X, Y or Z) carrying out NURBS interpolation with the control point in the first block (G06.2 block). If an axis not commanded in the first block is commanded in the second and following blocks, an alarm will occur.
- (c) The first control point (G06.2 block coordinate value) should be commanded as the start point of the NURBS curve. Thus, command so that it matches the end point of the previous block. An alarm will occur if the points do not match.
- (d) The command range of the weight is 0.0001 to 99.9999. Even if the decimal point is omitted, the value will be handled as a value with a decimal point.
If "1" is commanded, the result will be the same as "1.0".
- (e) The knot cannot be omitted, and must be commanded in each block. An alarm will occur if omitted.
- (f) As with the weight, up to four digits after the decimal point can be commanded for the knot. However, even if the decimal point is omitted, the value will be handled as a value with a decimal point.
- (g) Command the knot with a monotone increment. If the value is smaller than the previous block, an alarm will occur.
- (h) There is no particular order to command the addresses.
- (i) During single block operation, the operation will stop at the position corresponding to that block's knot of the NURBS curve shape instead of the control point.

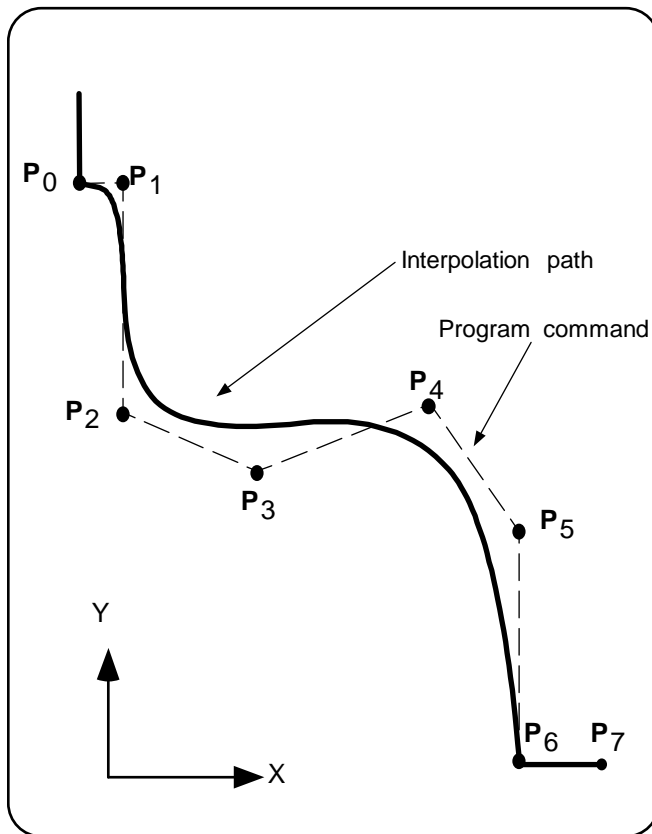
6. POSITIONING AND INTERPOLATION FUNCTIONS
6.9 NURBS Interpolation; G06.2

(5) Program example

An example of a program with four stages (three-dimension curve) and seven control points is shown below.

Control point : P0 P1 P2 P3 P4 P5 P6

Knot vector : [0.0 0.0 0.0 0.0 1.0 2.0 3.0 4.0 4.0 4.0 4.0]



```

:
:
:
G90 G01 X0 Y120. F3000 ;
Y100. ; ..... P0
G6.2 P4 X0 Y100. R1. K0 ; ..... P0
X10. Y100. R1. K0 ; ..... P1
X10. Y60. R1. K0 ; ..... P2
X60. Y50. R1. K0 ; ..... P3
X80. Y60. R1. K1. ; ..... P4
X100. Y40. R1. K2. ; ..... P5
X100. Y0 R1. K3. ; ..... P6
K4. ;
K4. ;
K4. ;
K4. ;
G01 X120. ; ..... P7
:
:
:

```

6. POSITIONING AND INTERPOLATION FUNCTIONS
6.10 Thread Cutting; G33

6.10 Thread cutting; G33

(1) Outline

The thread cutting command (G33) enables thread cutting with a commanded number of thread leads or number of ridges per inch. When the leads are designated, the command value unit is [mm/rev] or [inch/rev], but the unit is always [ridges/inch] when designating the number of ridges. Multiple thread screws, etc., can also be machined by designating the thread cutting start angle (0 to 360°).

- (a) Uniform lead thread cutting (lead designation)
 The tool feed is controlled in synchronization with the spindle rotation using the synchronization signal from the spindle encoder as the base, and the designated lead threads are cut. This makes it possible to conduct constant-lead straight thread-cutting and tapered thread-cutting.
- (b) Inch thread cutting (number of ridge designation)
 If the number of ridges per inch in the long axis direction is assigned, the feed of the tool synchronized with the spindle rotation will be controlled, which means that constant-lead straight thread-cutting and tapered thread-cutting can be performed.
- (c) Continuous thread cutting
 When the lead or shape changes during the threads, continuous thread cutting can be realized by commanding thread cutting commands in succession.

(2) Command format

- (a) Constant lead thread cutting (lead designation)

<u>G33</u>	<u>Zz1</u>	<u>Ff1</u>	<u>Qq1</u>	; (Normal lead thread cutting commands)
Thread command	Thread end point	Thread lead	Shift angle	

- Zz1 : Thread cutting direction axis address (X, Y, Z, α) and thread length
- Ff1 : Lead of long axis (axis which moves most) direction.
- Qq1 : Thread cutting start shift angle (0 to 360°)

<u>G33</u>	<u>Zz1</u>	<u>Ee1</u>	<u>Qq1</u>	; (Precision lead thread cutting commands)
Thread command	Thread end point	Thread lead	Shift angle	

- Zz1 : Thread cutting direction axis address (X, Y, Z, α) and thread length
- Ee1 : Lead of long axis (axis which moves most) direction.
- Qq1 : Thread cutting start shift angle (0 to 360°)

- (b) Inch thread cutting (number of ridge designation)

<u>G33</u>	<u>Zz1</u>	<u>Ee1</u>	<u>Qq1</u>	;
Thread command	Thread end point	Number of threads	Shift angle	

- Zz1 : Thread cutting direction axis address (X, Y, Z, α) and thread length
- Ee1 : Number of ridges per inch in direction of long axis (axis which moves most) (decimal point command can also be assigned)
- Qq1 : Thread cutting start shift angle (0 to 360°)

(3) Detailed description

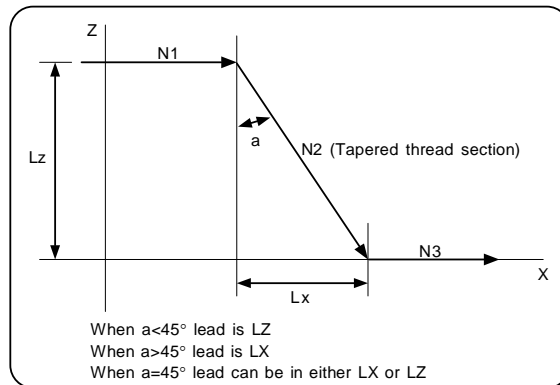
- (a) Uniform lead thread cutting (lead designation)

- 1) By using the E command, the leads can be commanded in a smaller unit than the F command.
 The E command is also used to command the number of ridges for inch thread cutting, but it can also be used to designate the precision leads by turning the control parameter "Precision thrd cut E" ON.

6. POSITIONING AND INTERPOLATION FUNCTIONS

6.10 Thread Cutting; G33

- 2) Tapered threads can be machined by commanding several axis addresses. In this case, the lead in the long axis direction is commanded.



- 3) Thread lead command range

Metric command		
Min. setting unit (mm)		Lead command range (mm/rev)
0.001	E command	0.00001 to 999.99999
	F command	0.0001 to 999.9999

Inch command		
Min. setting unit (inch)		Lead command range (inch/rev)
0.0001	E command	0.0000001 to 9.9999999
	F command	0.000001 to 99.999999

- 4) Thread lead command unit
(FA on the Modal Display screen is not updated)

Metric command		
Min. setting unit (mm)		Lead command range (mm/rev)
0.001	E command	E1=0.00001 (E1.=1.00000)
	F command	F1=0.0001 (F1.=1.0000)

Inch command		
Min. setting unit (inch)		Lead command range (inch/rev)
0.0001	E command	E1=0.0000001 (E1.=1.0000000)
	F command	F1=0.000001 (F1.=1.000000)

- 5) Shift angle command unit

Metric command		
Min. setting unit (°)		Shift angle command unit (°)
1.	Q command	Q1=0.001 (Q1.=1.000)

Inch command		
Min. setting unit (°)		Shift angle command unit (°)
1.	Q command	Q1=0.0001 (Q1.=1.0000)

Command the shift angle as an integer between 0° and 360°. Decimal values will be ignored even if commanded.

- 6) The constant surface speed control function should not be used for taper thread cutting commands or scrolled thread cutting commands.
- 7) The spindle speed should be kept constant throughout from the rough cutting until the finishing.
- 8) If the feed hold function is employed during thread cutting to stop the feed, the thread ridges will lose their shape. For this reason, feed hold does not function during thread cutting.
 If the feed hold switch is pressed during thread cutting, block stop will result at the end point of the next movement block that does not contain thread cutting.
 (The "Auto operation pause signal ON" warning will not appear.)
- 9) When thread cutting is started, the feedrate determined by the lead and spindle rotation speed and the cutting feed clamp speed will be compared. If the speed exceeds the cutting feed clamp speed, an error will occur.
- 10) In order to protect the lead during thread cutting, an actual feedrate may sometimes exceed the cutting feed clamp rate.

6. POSITIONING AND INTERPOLATION FUNCTIONS
6.10 Thread Cutting; G33

- 11) An illegal lead is normally produced at the start of the thread cutting and at the end of that because of servo system delay and other such factors.
 Therefore, it is necessary to command a thread length which is determined by adding the illegal lead lengths to the required thread length.

- 12) The spindle speed is subject to the following restriction:

$$1 \leq R \leq \frac{\text{Maximum feedrate}}{\text{Thread lead}}$$

Where R ≤ Permissible speed of encoder (r/min)
 R = Spindle speed (r/min)
 Thread lead = mm or inches
 Maximum feedrate = mm/min or inch/mm
 (this is subject to the restrictions imposed by the machine specifications).

- 13) Spindle override is invalid during thread cutting, and will be fixed to 100% regardless of the external signal. If thread cutting is executed when an override value other than 100% is selected, the 100% override will be applied during thread cutting. The selected override value will be applied when thread cutting is completed.
- 14) Single block is invalid during thread cutting. Block stop will be applied at the end point of the next movement block that does not contain thread cutting.
- 15) If the operation mode is changed during execution of G33, block stop will be applied at the end point of the next movement block that does not contain thread cutting.
- 16) The thread cutting command waits for the single rotation sync signal of the rotary encoder and starts movement. Note that when using multiple systems, if a system issues the thread cutting command while another system is executing thread cutting, movement will start without waiting for the rotary encoder single rotation sync signal. This will cause the thread ridges to deviate, so do not issue the thread cutting command when using multiple systems.
- 17) Thread cutting is executed in synchronization with the spindle selected with the R register (R1839: command spindle selection). Note that if multiple spindles are selected, the spindle with the smallest number will be used as the thread cutting spindle.
(Example 1) When the R1839 value is "4" (bit 2 is set to 1) → The 3rd spindle will be the thread cutting spindle.
(Example 2) When the R1839 value is "5" (bit 0 and bit 2 are set to 1) → The 1st spindle will be used as the thread cutting spindle.

(b) Inch thread cutting (number of ridge designation)

- 1) Number of thread ridge command range (The direction with the most movement is used as the reference)

Metric command	
Command unit (mm)	Number of ridge command range (ridge/inch)
0.001	0.01 to 999999.99

Inch command	
Command unit (inch)	Number of ridge command range (ridge/inch)
0.0001	0.0001 to 9999.9999

- 2) Number of thread ridge command unit

Inch command	
Command unit (inch)	Lead command unit (ridge/inch)
0.0001	E1=1 (E1=1.0000)

Metric command	
Command unit (mm)	Lead command unit (ridge/inch)
0.001	E1=1 (E1=1.00)

- 3) The number of ridges in the long axis direction is assigned as the number of ridges per inch.
- 4) The E command is also used to command the leads for precision thread cutting, but it can also be used to designate the number of ridges by turning the control parameter "Precision thrd cut E" OFF.

6. POSITIONING AND INTERPOLATION FUNCTIONS

6.10 Thread Cutting; G33

- 5) The E command value should be set within the lead value range when the lead is converted.
(Lead value (inch) = 1/ridge)
- 6) See (a) Uniform lead thread cutting for other details.

(4) Restrictions

- (a) Dry run can be validated for thread cutting by turning the control parameter "Dry run in thread cut" ON. Note that the feedrate in dry run will not be synchronized to the spindle rotation. The dry run signal is checked at the start of thread cutting, and any changes during thread cutting are ignored.
- (b) If thread cutting is commanded during tool radius compensation, the tool radius compensation will be temporarily canceled, and thread cutting will be executed. Any movement commands for axes other than the compensation axis will not be temporarily canceled.
- (c) If a group 1 modal command is issued in the same block, the G code commanded last will have the priority.
- (d) If group 9 (fixed cycle) is commanded in the same block, the thread cutting command will have the priority.
- (e) If G93 (inverse time feed) is commanded during the thread cutting modal, or if thread cutting is commanded during the G93 modal, the program error "G93 mode error" will occur.

7. FEED FUNCTIONS

7.1 Rapid Traverse Rate

(1) Detailed description

- (a) The rapid traverse rate can be independently set for each axis.
- (b) The rapid traverse rate is valid in the following G codes: G00, G27, G28, G29, G30 and G60.
- (c) When positioning, the path from the start point to the end point is interpolated by a straight line. With the parameters it is also possible to select non-interpolation type positioning that moves at the maximum speed of each axis. The time taken for positioning is the same with either type. Note that in reference point return, the interval between from the middle point to the reference point is only non-interpolation type. (Interpolation type is invalid.)
- (d) An override can be applied to the rapid traverse rate by an external signal dependent on PLC specifications.

up to 100% is one step

- F0, F1, 50%, 100%

F0: Can be set in the parameters at a constant speed for each axis.

F1: Can be set in the parameters at a constant percent.

- (e) Setting range of the rapid traverse rate

Min. setting unit	A, B	B', C
Metric input system	1 to 480000 (mm/min, °/min)	1 to 100000 (mm/min, °/min)
Inch input system	1 to 18897 (inch/min)	1 to 3937 (inch/min)

Minimum setting range A: 0.01mm (0.001inch)

Minimum setting range B: 0.001mm (0.0001inch)

Minimum setting range B': 0.001mm (0.0001inch)

Minimum setting range C: 0.0001mm (0.00001inch)

7. FEED FUNCTIONS

7.2 Cutting Feedrate

7.2 Cutting Feedrate

(1) Outline

This function designates the feedrate of the cutting command. The feed amount per spindle rotation or feed amount per minute is designated.

(2) Command format

F f1 ;

(3) Detailed description

- (a) Once set, that value is stored in the memory as a modal. The feedrate modal value is cleared to zero only by a restart.
- (b) The maximum value of the cutting feedrate is clamped by the parameters (setting range is the same as the cutting feedrate) of the cutting feed clamp speed.
- (c) Setting range of the cutting feedrate

Min. setting unit	A, B	B', C
Metric input system	1 to 480000 (mm/min, °/min)	1 to 100000 (mm/min, °/min)
Inch input system	1 to 18897 (inch/min)	1 to 3937 (inch/min)

Minimum setting range A: 0.01mm (0.001inch)
 Minimum setting range B: 0.001mm (0.0001inch)
 Minimum setting range B': 0.001mm (0.0001inch)
 Minimum setting range C: 0.0001mm (0.00001inch)

- (d) G code commands in which the cutting feed is valid
Refer to the interpolation function specifications for information on other commands (G01, G02, G03, G02.1, G03.1, G33, etc.).
- (e) Number of designated valid digits in the cutting feedrate
F9-digit direct designation (integer part: 6 digits, decimal part: 3 digits)
- (f) Refer to the specifications for acceleration/deceleration before interpolation for information on the cutting feed clamp speed during acceleration/deceleration before interpolation.

(4) Notes

- (a) A program error will occur if there is no F command in the first cutting command (G01, G02, G03, G02.1, G03.1) after restarting.

(5) Others

- (a) Feedrate designation and effects on control axes
It has already been mentioned that a machine has a number of control axes. These control axes can be divided into linear axes which control linear movement and rotation axes which control rotation movement.
The feedrate is designed to designate the displacement speed of these axes, and the effect exerted by this feedrate on the tool movement speed which affects cutting differs between the time when control is exercised over the linear axes and when it is exercised over the rotation axes.
The displacement amount for each axis is designated separately by a value corresponding to the respective axis. The feedrate is not designated for each axis but designated as a single value. Therefore, when two or more axes are to be controlled simultaneously, it is necessary to understand how this will work for each of the axes involved.
The designation of the feedrate is described with the following related items.

1) When controlling linear axes

Even when only one machine axis is to be controlled or there are two or more axes to be controlled simultaneously, the feedrate which is designated by the F code functions as a linear speed in the tool advance direction.

When only linear axes are to be controlled, it is sufficient to designate the cutting feedrate in the program.

The designated feedrate is broken down into components corresponding to the movement amounts, and each of these components constitutes the feedrate for each axis.

7. FEED FUNCTIONS

7.2 Cutting Feedrate

2) When controlling rotation axes

When rotation axes are to be controlled, the designated feedrate functions as the rotation speed of the rotation axes or, in other words, as an angular speed.

Consequently, the cutting feedrate in the tool advance direction, or in other words the linear speed, varies according to the distance between the center of rotation and the tool. This distance must be considered when designating the feedrate in the program.

(Note) When the circular interpolation function is used and the tool is moved along the circumference by the linear control axis, the rate in the tool advance direction, or in other words the tangential direction, will be the feedrate designated in the program.

3) When linear and rotation axes are to be controlled at the same time

The NC system is operated in exactly the same way whether linear or rotation axes are to be controlled.

When a rotation axis is to be controlled, the numerical value assigned by the coordinate words (A, B, C) is the angle and the numerical values designated by the feedrate (F) are all handled as linear speeds. In other words, 1° of the rotation axis is handled as being equivalent to 1mm of the linear axis.

Consequently, when both linear and rotation axes are to be controlled simultaneously, the components of the numerical value designated by F — each component corresponding to each axis — will be the same as for item 1) above (When controlling linear axes).

In this case, however, although both the size and direction of the speed components based on linear axis control do not vary, the direction of the speed components based on rotation axis control will change along with the tool movement (their size will not change).

This means, as a result, that the composite tool advance direction feedrate will vary along with the tool movement.

7. FEED FUNCTIONS
7.3 Feed per Minute; G94

7.3 Feed per Minute; G94

(1) Outline

By issuing a G94 command, the commands from that block can be commanded as feedrate per minute (mm/min, inch/min) using an F code.

(2) Command format

G94 Ff1 ;

(3) Detailed description

- (a) Feed per minute is a modal command. Once commanded, it is valid until feed per rotation (G95) is commanded.
- (b) Command range

Metric input (mm)

Min. setting unit		(A) 0.01mm	(B) 0.001mm	(B') 0.001mm	(C) 0.0001mm
F command unit (mm/min)	No decimal point With decimal point	F1 =1mm/min F1.=1mm/min	F1 =1mm/min F1.=1mm/min	F1 =1mm/min F1.=1mm/min	F1 =1mm/min F1.=1mm/min
Command range (mm/min)		0.01 to 480000.00	0.001 to 480000.000	0.001 to 100000.000	0.0001 to 100000.0000

Inch input (inch)

Min. setting unit		(A) 0.001inch	(B) 0.0001inch	(B') 0.0001inch	(C) 0.00001inch
F command unit (inch/min)	No decimal point With decimal point	F1 =1inch/min F1.=1inch/min	F1 =1inch/min F1.=1inch/min	F1 =1inch/min F1.=1inch/min	F1 =1inch/min F1.=1inch/min
Command range (inch/min)		0.001 to 18897.6378	0.0001 to 18897.6378	0.0001 to 3937.0078	0.00001 to 3937.00787

- 1) 1mm/min (1inch/min) or less cannot be commanded without a decimal point.
When carrying out 1mm/min (1inch/min) or less commands, use a decimal point.
 - 2) With the parameter "F cmd mgf type /min" settings, the feedrate can be magnified by 1/10 and 1/100 using the F command commanded with no decimal point.
 - 3) By turning the parameter "Initial synchr feed" OFF, the initial status after restart can be set to asynchronous feed (feed per minute).
 - 4) The F command unit is common to all systems.
- (c) By turning the parameter "Initial synchr feed" OFF , the initial status after restart can be set to asynchronous feed (feed per minute).

7. FEED FUNCTIONS
7.4 Feed per Rotation; G95

7.4 Feed per Rotation; G95

(1) Outline

By issuing a G95 command, the command from that block can be commanded as feedrate per spindle rotation (mm/rev, inch/rev) using an F code. A rotary encoder must be installed on the spindle to use this command.

(2) Command format

G95 Ff2 ;

(3) Detailed description

- (a) The G95 command is a modal command. Once commanded, it is valid until G94 (feed per minute) is commanded.
- (b) The minimum command unit and command range of the feedrate designation F are as follows.

Metric input (mm)

Min. setting unit		(A) 0.01mm	(B) 0.001mm	(B') 0.001mm	(C) 0.0001mm
F command unit (mm/rev)	No decimal point	F1 = 0.1	F1 = 0.01	F1 = 0.01	F1 = 0.001
	With decimal point	F1. = 1	F1. = 1	F1. = 1	F1. = 1
Command range (mm/rev)		0.001 to 99999.999	0.0001 to 9999.9999	0.0001 to 999.9999	0.00001 to 999.99999

Inch input (inch)

Min. setting unit		(A) 0.001inch	(B) 0.0001inch	(B') 0.0001inch	(C) 0.00001inch
F command unit (inch/rev)	No decimal point	F1 = 0.01	F1 = 0.001	F1 = 0.001	F1 = 0.0001
	With decimal point	F1. = 1	F1. = 1	F1. = 1	F1. = 1
Command range (inch/rev)		0.001 to 3937.007	0.0001 to 393.7007	0.0001 to 39.3700	0.00001 to 39.37007

- 1) 1mm/min (1 inch/min) or less cannot be commanded without a decimal point.
- 2) The F command unit is common to all systems.

- (c) With the parameter "F cmnd mgf type /rot" settings, the feedrate can be magnified by 1/10 and 1/100 using the F command commanded with no decimal.
- (d) By turning the parameter "Initial synchr feed" ON, the initial status after restart can be set to the synchronous feed (feed per rotation) mode.
- (e) The execution speed (actual machine movement speed) for feed per rotation is as follows.

$$FC = F * N * OVR \dots \dots \dots \text{(expression 1)}$$

FC : Effective speed (mm/min, inch/min)
 F : Command speed (mm/rev, inch/rev)
 N : Spindle rotation speed (r/min)
 OVR : Cutting feed override

When several axes are commanded simultaneously, the execution speed FC by expression 1 operates in the vector direction of that command.

- (f) The FC of the setting and display unit "Position display" screen displays the execution speed (mm/min or inch/min) converted from the command speed, spindle speed and cutting feed override to the feedrate per minute.
- (g) When the execution speed above exceeds the cutting feed clamp speed, the speed is clamped at that speed.
- (h) During machining lock high-speed processing, the value set in parameter "#320004 Machine lock rapid " becomes valid regardless of command speed and spindle speed. When not carrying out high-speed processing, the feedrate is the same as without machining lock.
- (i) During dry run, the feedrate becomes feed per minute, and moves at the manual feedrate (mm/min, inch/min, °/min).
- (j) The fixed cycle G84 (tap cycle) and G74 (reverse tap cycle) are executed following a previously designated feed mode.

(4) Notes

- (a) The operation warning "OPW 112 Spindle stop" will occur if the spindle speed is zero during feed per rotation execution.
 If the speed is slow (9r/min or less), a warning may be output. However, the operation will not be obstructed.

7.5 Inverse Time Feed; G93

(1) Outline

During inside cutting when machining curved shapes with radius compensation applied, the machining speed on the cutting surface becomes faster than the tool center feedrate. Therefore, problems such as reduced accuracy may occur.

This reduced accuracy can be prevented with inverse time feed. This function can, in place of normal feed commands, issue one block of machining time (inverse) in F commands. The machining speed on the cutting surface is constantly controlled, even if radius compensation is applied in to the machining program that expresses the free curve surface with fine segment lines. Note that when the calculated machining time exceeds the cutting feed clamp speed, the F command value in the inverse time feed follows the cutting feed clamp speed.

(2) Command format

G93 ;

Inverse time feed (G93) is a modal command. Once commanded, it is valid until feed per minute or feed per rotation is commanded.

G00 Xx1 Yy1 ;

G93 ; → Inverse time feed mode ON

G01 Xx2 Yy2 Ff2 ; → In inverse time feed mode

G02 Xx3 Yy3 Ii3 Jj3 Ff3 ;

G94 (G95) ; → Inverse time feed mode OFF

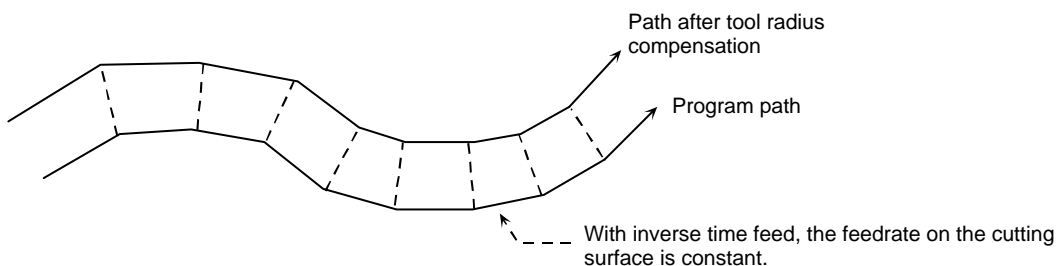
(3) Detailed description

(a) Inverse time feed (G93) is a modal command. Once commanded, it is valid until feed per minute (G94) or feed per rotation (G95) is commanded, or until a reset (M02, M30, etc.) is executed.

(b) Command method of F command values in inverse time feed

	Metric command (G21)	Inch command (G20)
During linear mode (G01)	Cutting feedrate [mm/min] Cutting distance [mm]	Cutting feedrate [inch/min] Cutting distance [inch]
During arc mode (G02, G03) (G02.1, G03.1)	Cutting feedrate [mm/min] Start point arc radius [mm]	Cutting feedrate [inch/min] Start point arc radius [inch]

Handling of F command	Unmodal (command per block)
When F0 is commanded	Program error (no F command)
When there is no F command	Program error (no F command)
Command range (min ⁻¹)	0.001~99999.999



7. FEED FUNCTIONS

7.5 Inverse Time Feed; G93

(c) Example of program

When inverse time feed is used during radius compensation

Asynchronous feed

```
N01 G90 G00 X-80. Y80;
N02 G01 G41 X0 Y0 D11 F500;
N03 X100.;
N04 G02 Y-200. R100.;
N05 G03 Y-400. R100.;
N06 G02 Y-600. R100.;
N07 G01 X0 F500;
N08 Y0;
N09 G40 X-80. Y80.;
N10 M02;
```

Inverse time feed

```
N01 G90 G00 X-80. Y80;
N02 G01 G41 X0 Y0 D11 F500;
N03 X100.;
N04 G93 G02 Y-200. R100.;
N05 G03 Y-400. R100.;
N06 G02 Y-600. R100.;
N07 G94 G01 X0 F500;
N08 Y0;
N09 G40 X-80. Y80.;
N10 M02;
```

Comparison of asynchronous feed and inverse time feed (when tool radius is 10.0mm)

Location	Asynchronous feed		Inverse time feed	
	Feedrate of tool center	Feedrate of cutting point	Feedrate of tool center	Feedrate of cutting point
N04	F500	F450	F550	F500
N05	F500	F550	F450	F500
N06	F500	F450	F550	F500

↑
The block seam protrudes due to the cutting speed change at the block seam.

↑
The feedrate follows the command regardless of the tool radius.

- (d) The initial modal after a restart is G94 (feed per minute) or G95 (feed per rotation).
- (e) The feedrate of the block inserted in tool radius compensation, C axis normal line control and corner R/C is the same speed as the feedrate of the block immediately before it.
- (f) The feed function is clamped at the maximum cutting speed. Consequently, the feed may be slower than the commanded speed.
- (g) If an extremely slow speed such as F0.001 is designated, an error will occur in the machining time.

(4) Notes

- (a) The F command in G93 modal is unmodal. Issue an F command for each block. The program error "PR0062" will occur in blocks with no F command.
- (b) The program error "PR0062" will occur when F0 is commanded.
- (c) An F command is necessary when G93 to G94 or G95 is commanded. The program error "PR0062" will occur if there is no F command.
- (d) In the inverse time feed mode, the program error will occur when the commands below are issued.

G12	Inside cutting CW
G13	Inside cutting CCW
G31 to G31.1	Skip
G33	Thread cutting
G34 to G36, G37.1	Special fixed cycle
G37	Automatic tool length measurement
G73 to G89	Fixed cycle
G96	Constant surface speed control ON

(e) The program error will occur if inverse time feed (G93) is commanded in the following modes.

G33	Thread cutting
G73 to G89	Fixed cycle
G96	Constant surface speed control

- (f) Even if the control parameter "1-digit F feed valid" is ON, the F1-digit feed command will not be valid during the G93 modal.
- (g) The dwell command in the inverse time feed modal is a time designation (dwell per second).

7. FEED FUNCTIONS
7.6 Synchronized Tapping; ,R/H

7.6 Synchronized Tapping; ,R/H

(1) Outline

This function carries out tapping while carrying out synchronized control of the spindle and servo. High-accuracy tap depth tapping can be carried out without a floating tap.

(2) Command format

(a) Tap pitch designation

<u>G84</u> <u>(G74)</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Zz1</u>	<u>Rr1</u>	<u>Pp1</u>	<u>Ff1</u>	<u>Ss1</u>	<u>, R1/Hh ;</u>
Tapping command	Hole position data		Hole machining data			Pitch	Spindle speed	Synchronized method

(b) Designation of the number of tap threads

<u>G84</u> <u>(G74)</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Zz1</u>	<u>Rr1</u>	<u>Pp1</u>	<u>Ee1</u>	<u>Ss1</u>	<u>, R1/Hh ;</u>
Tapping command	Hole position data		Hole machining data			Number of threads	Spindle speed	Synchronized method

The address data is as follows.

- G84 : Forward tap G74: Reverse tap
- X, Y : Drilling coordinate position
- Z : Hole bottom position
- R : Hole R position
- P : Dwell time at hole bottom
- F : Z axis feed amount (tap pitch) per spindle rotation
- E : Number of tap threads per Z axis inch
- S : Spindle rotation speed
- ,R/H : Synchronous method selection
 (,R0: asynchronous method ,R1: synchronous method
 H0: asynchronous method Hh: (a)synchronous method (by the parameters))
 (When ,R and H are commanded in the same block, the H command is ignored.) (h>0)

(3) Detailed description

(a) The NC control status is as follows when the synchronous tapping command (G74, G84) is carried out.

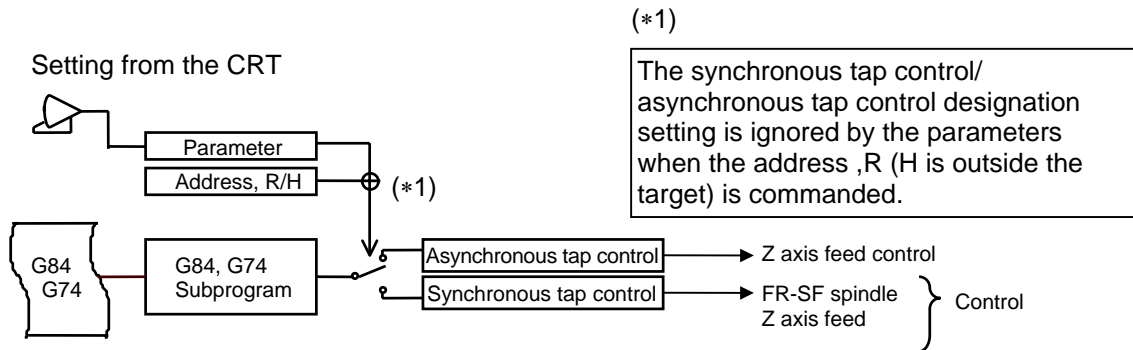
1. Cutting override 100% fixed
2. Feed hold invalid
3. IN TAPPING MODE signal output
4. Block seam deceleration command invalid
5. Single block invalid

(Note) With plane selection, the synchronous tapping cycle can be carried out for axes other than the Z axis.

7. FEED FUNCTIONS

7.6 Synchronized Tapping; ,R/H

(b) The tapping control method is shown in the following drawing.



(c) Synchronous/asynchronous control is judged by ",R command" and parameters or "H command" and parameters.

,R command (or H command)

	No ,R command No H command	,R0 H=0	,R1 H>0
Synchronous tap parameter: ON	Synchronous method (*1)	Asynchronous method	Synchronous method (*2)
Synchronous tap parameter: OFF	Asynchronous method	Asynchronous method	(A)Synchronous method (*3)

*1) Retract override parameter is referred to.

*2) Retract override parameter is not referred to. (address H)

*3) Asynchronous method during H command

(d) Tap return override

With this function, an override can be applied to the retract from the hole bottom to the R point during synchronous tap.

1) Retract override valid/invalid

The "#3900=1" command of the fixed cycle subprogram validates the retract override. (Only validated during synchronous tapping.)

2) Retract override setting

The override is set in the basic specification parameter "tapovr" of the machine parameters. The setting range is from 0 to 999%, but when set below 100% a 100% clamp is applied.

3) Speed clamp when the retract override is valid

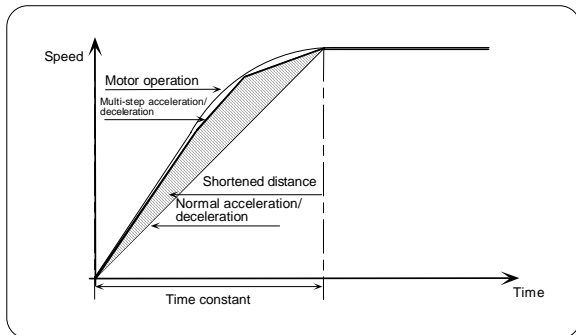
In situations such as when the speed with a retract override applied exceeds the cutting clamp speed of the hole drilling axis or the spindle tap maximum speed (tap maximum rotation speed of the selected gear level), the hole drilling axis feedrate and spindle rotation speed is clamped by the cutting clamp speed of the hole drilling axis or the spindle tap maximum rotation speed.

7. FEED FUNCTIONS

7.6 Synchronized Tapping; ,R/,H

(4) Multi-step acceleration/deceleration

When acceleration/deceleration is carried out in the synchronous tap cycle, the acceleration rate is calculated with the speed/time. However, in actual operation, if the motor characteristics provide a constant output and a set rotation speed is reached, the torque will decrease. Thus, the positioning time may increase because a longer time constant is set. To reduce this, the spindle and hole drilling axis acceleration/deceleration pattern can be divided into multiple steps so that the spindle acceleration/deceleration pattern is similar to the acceleration/deceleration pattern for the speed loop. This will shorten the time to reach the rotation speed commanded for the spindle.



(a) Command method

Select time constant/constant inclination acceleration/deceleration or multi-step acceleration/deceleration with the parameter "S-tap Slope/Time".

(b) Command format

```
G84   Xx  Yy  Zz  Rr  Pp  Ff  Ss  ,R1  Hh  ,Ss ;
(G74)                               (Ee)
```

Basically, the command format is the same as for the normal time/constant inclination acceleration/deceleration command. With multi-step acceleration/deceleration, the spindle rotation speed for retracting can be commanded with the address (,S).

Address	Details	Command range
,S	Spindle rotation speed for retracting <ul style="list-style-type: none"> • The data is held as modal information. Thus, even if the synchronous tap is canceled, the S command will be held. Note that this can be canceled with the parameter (#330106 Tap back speed CLR). 	0 to 99999 (r/min)

- When ,S is commanded, if the value is smaller than the S command, the S command will be valid during retract.
- When ,S (≠ 0) is commanded, the tap retract override will be invalid.
- The ,S command is canceled with reset (modal not held) or rewind. It is not canceled with reset (modal held).
- If the ,S command exceeds the maximum rotation speed for multi-step acceleration/deceleration retract (stmax), it will be clamped at the maximum rotation speed for multi-step acceleration/deceleration retract.
- When ,S0 is commanded, the command will be ignored and the tap retract override will be validated.
- If S is commanded during an asynchronous (,R0) command, the S command will be ignored.
- The retract spindle rotation speed can be changed even when S is commanded in the G84 modal.

Note that when ,S is commanded independently (with no positioning command), "PRW0004 Unused address exists" will occur.

7. FEED FUNCTIONS

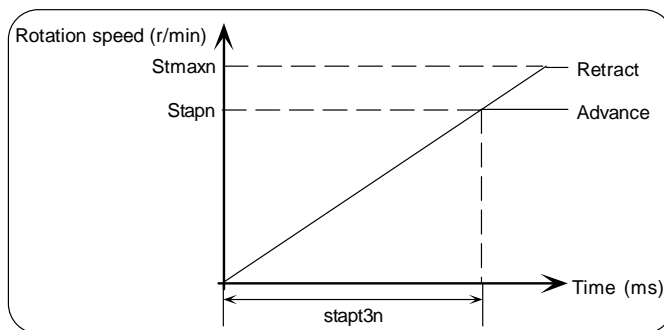
7.6 Synchronized Tapping; ,R/,H

- (c) Maximum rotation speed and time constants
 The maximum rotation speed and time constants for acceleration/deceleration are set for each step.

1st step		Maximum rotation speed		Time constant	
	Gear 00	#200089	Stap11	#200093	Stapt11
	Gear 01	#200090	Stap12	#200094	Stapt12
	Gear 10	#200091	Stap13	#200095	Stapt13
	Gear 11	#200092	Stap14	#200096	Stapt14
2nd step		Maximum rotation speed		Time constant	
	Gear 00	#200097	Stap21	#200101	Stapt21
	Gear 01	#200098	Stap22	#200102	Stapt22
	Gear 10	#200099	Stap23	#200103	Stapt23
	Gear 11	#200100	Stap24	#200104	Stapt24
3rd step		Maximum rotation speed for advancing	Maximum rotation speed for retracting	Time constant	
	Gear 00	#200014	Stap1	#200109	Stmax1
	Gear 01	#200015	Stap2	#200110	Stmax2
	Gear 10	#200016	Stap3	#200111	Stmax3
	Gear 11	#200017	Stap4	#200112	Stmax4

1) Cases for constant inclination acceleration/deceleration

- When 1st step's maximum rotation speed = 0, and
 2nd step's maximum rotation speed = 0, and
 3rd step's maximum rotation speed > 0, and
 3rd step's time constant > 0

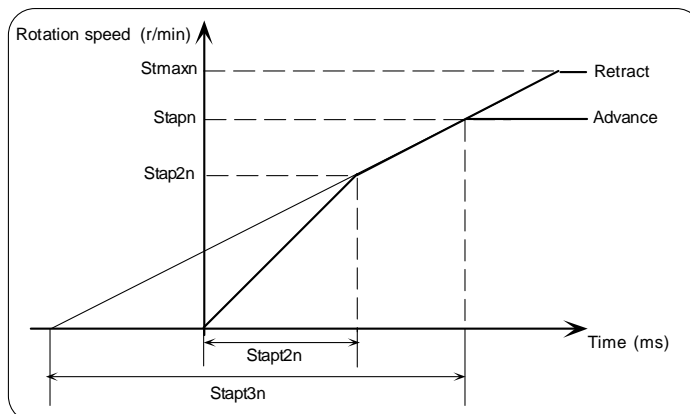


7. FEED FUNCTIONS

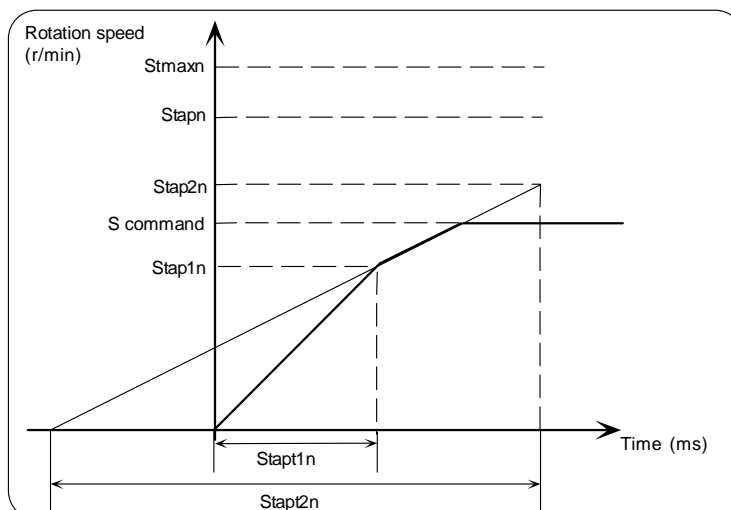
7.6 Synchronized Tapping; ,R/,H

2) Cases for two-step acceleration/deceleration

- When 1st step's maximum rotation speed = 0, and
 $0 < 2\text{nd step's maximum rotation speed} < 3\text{rd step's maximum rotation speed}$, and
 $0 < 2\text{nd step's time constant} < 3\text{rd step's time constant}$



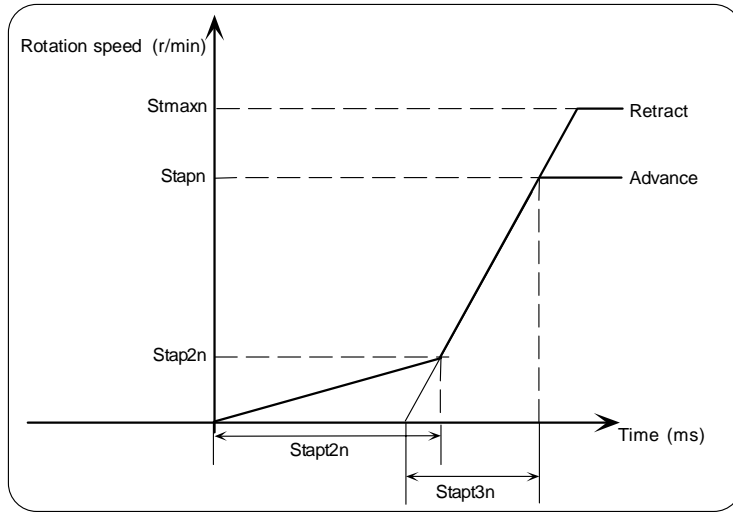
- 1st step's maximum rotation speed $\neq 0$, and
 $0 < 2\text{nd step's maximum rotation speed} < \text{commanded rotation speed} < 3\text{rd step's maximum rotation speed}$, and
 $0 < 2\text{nd step's time constant} < 3\text{rd step's time constant}$



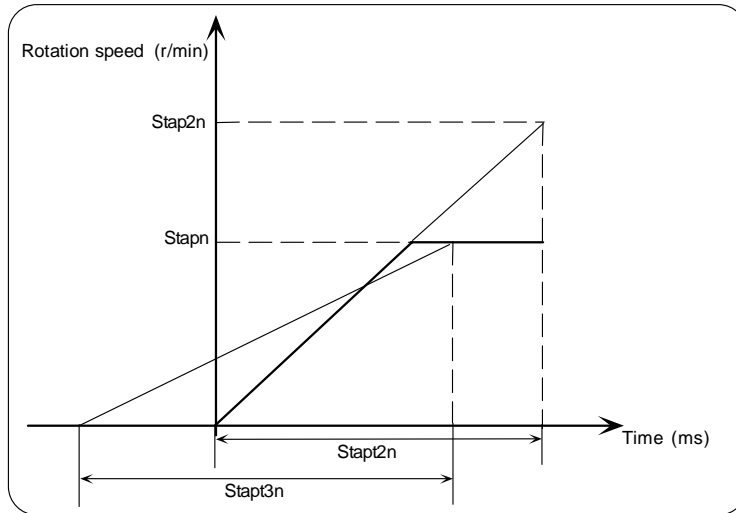
7. FEED FUNCTIONS

7.6 Synchronized Tapping; ,R/,H

- When 1st step's maximum rotation speed = 0, and $0 < 2\text{nd step's maximum rotation speed} < 3\text{rd step's maximum rotation speed}$, and $0 < 3\text{rd step's time constant} < 2\text{nd step's time constant}$



- When 1st step's maximum rotation speed = 0, and $0 < 3\text{rd step's maximum rotation speed} < 2\text{nd step's maximum rotation speed}$, and $0 < 2\text{nd step's time constant} < 3\text{rd step's time constant}$



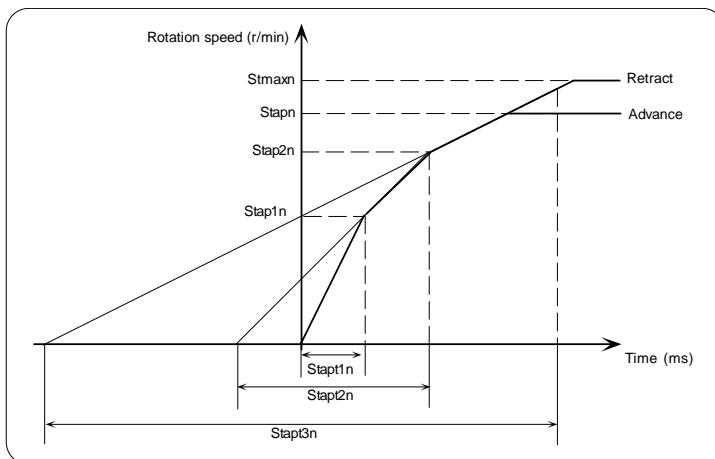
The axis will accelerate to the 3rd step's maximum rotation speed at the 2nd step's time constant

7. FEED FUNCTIONS

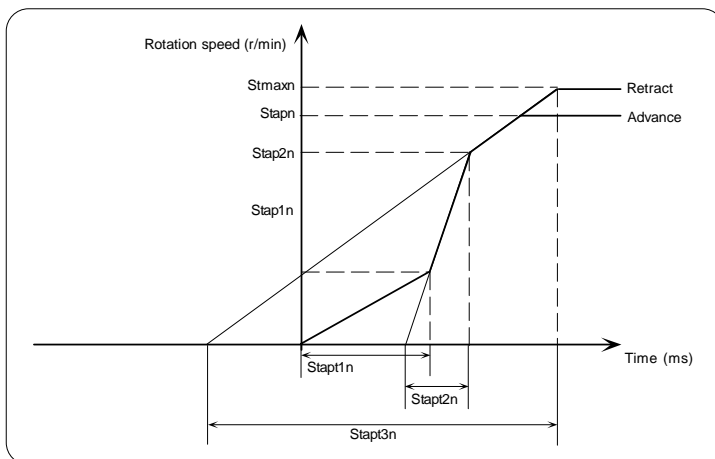
7.6 Synchronized Tapping; ,R/,H

3) Cases for three-step acceleration/deceleration

- $0 < 1\text{st step's maximum rotation speed} < 2\text{nd step's maximum rotation speed} < 3\text{rd step's maximum rotation speed}$, and
 $0 < 1\text{st step's time constant} < 2\text{nd step's time constant} < 3\text{rd step's time constant}$

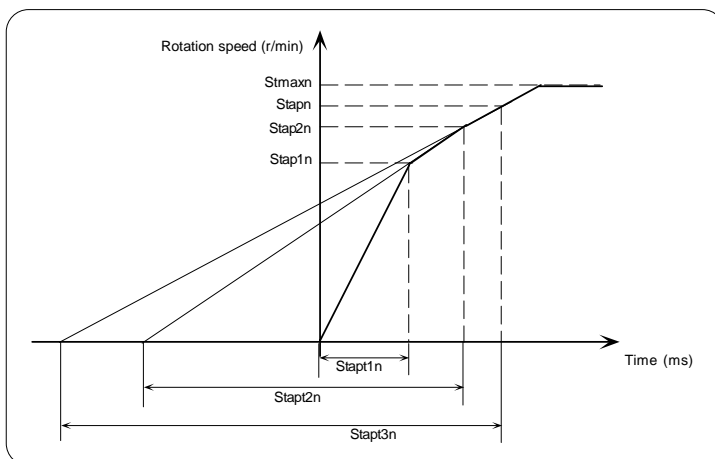


- $0 < 1\text{st step's maximum rotation speed} < 2\text{nd step's maximum rotation speed} < 3\text{rd step's maximum rotation speed}$, and
 $0 < 2\text{nd step's time constant} < 1\text{st step's time constant} < 3\text{rd step's time constant}$



(d) Override for retracting

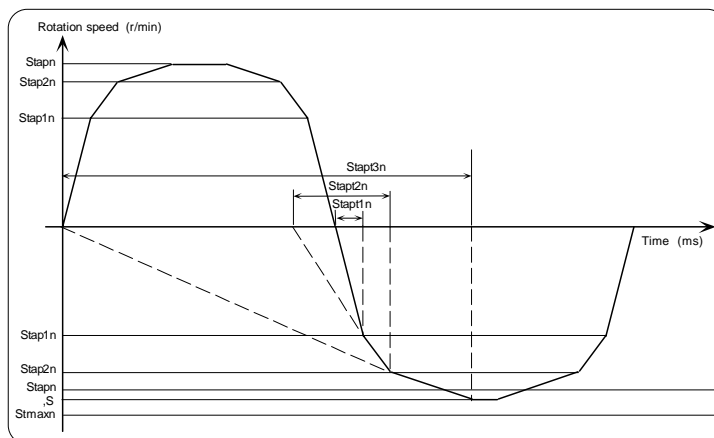
During multi-step acceleration/deceleration retract override, the 3rd step's maximum rotation speed becomes the maximum rotation speed for multi-step acceleration/deceleration retracting (stmax1 to stmax4). The inclination follows the 3rd step's maximum rotation speed and time constant.



7. FEED FUNCTIONS

7.6 Synchronized Tapping; ,R/,H

- (e) Spindle rotation speed command for retracting (,S)
 The maximum spindle rotation speed for retracting can be commanded with the address (,S).
 When ,S<S_{maxn} is established, the 3rd step's maximum rotation speed is clamped by ,S.
 The inclination in this case is determined by Stap_n and Stapt_{3n}.



(5) Restrictions

- (a) When both F and E are commanded simultaneously in the G84, G74 command block, the prioritized address will be determined by the input unit.
 Metric input ····· Pitch (F) has the priority. (The number of ridges is ignored.)
 Inch input ····· The number of ridges (E) has the priority. (The pitch is ignored.)
- (b) During synchronous tapping, the hole drilling axis's cutting feedrate and spindle rotation speed are clamped by the hole drilling axis's maximum cutting feedrate or the spindle maximum rotation speed.
- (c) During execution of G84, G74, the hole drilling axis's cutting feedrate is synchronized with the spindle rotation speed. However, the cutting feedrate modal (F modal) will not change at this time.
- (d) The forward run (M3), reverse run (M4) and stop (M5) commands are ignored during the synchronous tap cycle modal. (The M code and MF will not be output.)
 The spindle's rotation speed will be synchronized with the hole drilling axis movement, and will rotate in the forward or reverse direction. There is no need to issue the spindle forward run or reverse run command before the fixed cycle command.
- (e) If the S command is issued during the synchronous tap cycle modal, the program error "PR0033 Format error" will occur. To change the spindle rotation speed, cancel the synchronous tap cycle modal with G80, and then issue the G84 command.

[Example]

- 1) G84 Z-10. R-2. F1. S1000;
 X-10.;
 S2000; ← "PR0033 Format error"
 Y-10.;
- 2) G74 Z-10. R-2. F1. S2000;
 X-10.;
 S2000; ← "PR0033 Format error"
 Y-10.;
- 3) G74 Z-10. R-2. F1. S2000;
 G74 Z-12. R-2. F1. S1500; ← "PR0033 Format error"

7. FEED FUNCTIONS

7.6 Synchronized Tapping; ,R/,H

- (f) If there is no S command in the first G84 command block, synchronous tapping will be executed with the previously commanded S command (S modal). (The gears will not be changed in the G84 command block.)
- (g) The other restrictions are the same as the regular fixed cycle.
- (h) The spindle will not rotate while the miscellaneous function is locked.
- (i) If F/E are not commanded in the G84, G74 command block, the program error "PR0183 No pitch/thread number" will occur. Note that the previous F/E values will be used if the parameter (#320024 Synchronized tapping) is turned ON. In this case, the F/E values will be initialized when the NC is reset.

[Example] Parameter OFF

```
G84Xx Yy Zz Rr Ff Ss ,R1;  
Xx;  
G80;  
G84Xx Yy Zz Rr ,R1; ← "PR0183 No pitch/thread number"  
G80;
```

Parameter ON

```
G84Xx Yy Zz Rr Ff Ss ,R1;  
Xx;  
G80;  
G84Xx Yy Zz Rr ,R1; ← Operates with previous f  
G80;
```


7.7 Dwell (Time designation); (G94) G04

(1) Outline

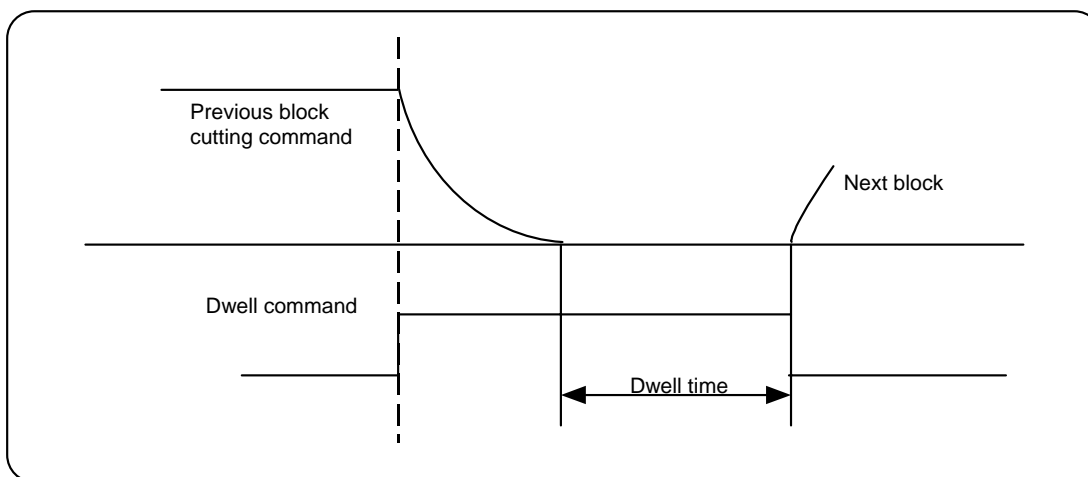
When G04 is commanded, the machine temporarily pauses, and goes to a standby status for the time commanded in the program. With this function, the start of the next block can be delayed. When the feed mode is feed per minute (G94), the operation becomes time designated. By adding a multi-step skip function, the remaining dwell time can be canceled.

(2) Command format

<p><u>(G94)</u> <u>G04</u> <u>Xx1</u> ; Besides address X, designation is also possible with address P.</p> <p>Asynchronous Dwell Time</p>
--

(3) Detailed description

- (a) The dwell time designation x1 range is from 0.001 to 99999.999s.
- (b) The designation with address P can be changed over between decimal point valid/invalid by the parameter "#320007 G04P decim pt valid". When the decimal point is invalidated by the parameter setting, commands below the decimal point with address P are ignored.
- (c) When a cutting command is in the previous block, the dwell command starts calculating the dwell time after the machine has decelerated and stopped. When it is commanded in the same block as an M, S, T or B command, the calculation starts simultaneously.
- (d) The dwell function is valid during interlock.
- (e) The dwell function is valid even for machine lock but it can be completed immediately by the control parameter "#320004 Machine lock rapid".
- (f) By the control parameter "#320005 G04 time fixed", the time designation (dwell per second) can be set, regardless of the feed mode.



(g) Examples of programs

- 1) When input unit is 0.001mm.

(Example 1) G04 X500 ; 0.5s. dwell time

(Example 2) G04 X5000 ; 5s. dwell time

(Example 3) G04 X5. ; 5s. dwell time

G04 P5000 ; 5s. dwell time

G04 P12.345 ; 0.012s. dwell time

(12.345s. when the decimal point is validated by the parameter settings.)

7.8 Dwell (Rotation designation); (G95) G04

(1) Outline

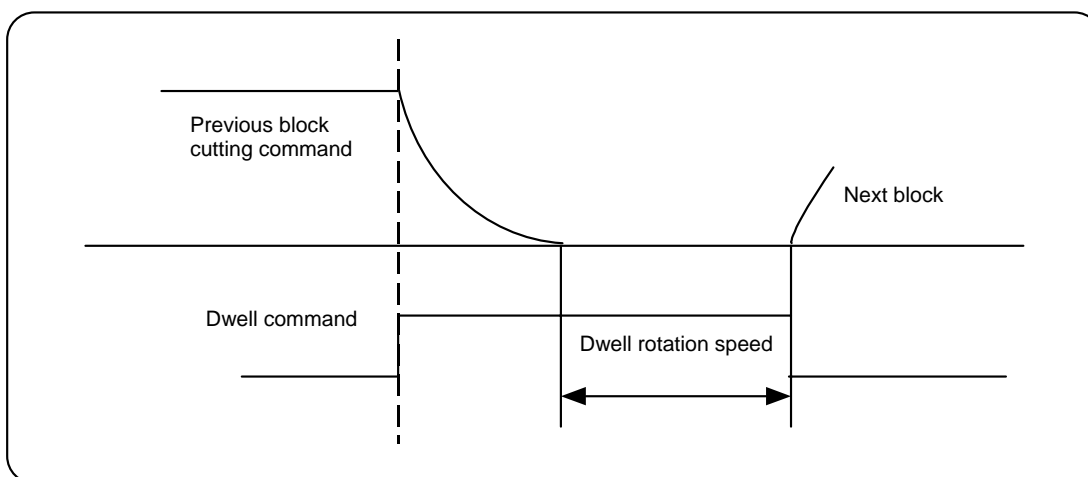
When G04 is commanded, the machine temporarily pauses, and goes to a standby status for the spindle rotation speed commanded in the program. With this function, the start of the next block can be delayed. When the feed mode is feed per rotation (G95), the operation becomes rotation speed designated. By adding a multi-step skip function, the remaining dwell time can be canceled.

(2) Command format

<p><u>(G95)</u> <u>G04</u> <u>Xx1</u> ; Besides address X, designation is also possible with address P.</p> <p>Synchronous Dwell Rotation</p>

(3) Detailed description

- (a) The dwell rotation designation x1 range is from 0.001 to 99999.999rev.
- (b) The designation with address P can be changed over between decimal point valid/invalid by the parameter "#320007 G04P decim pt valid". When the decimal point is invalidated by the parameter setting, commands below the decimal point with address P are ignored.
- (c) When a cutting command is in the previous block, the dwell command starts calculating the dwell time after the machine has decelerated and stopped (Refer to the figure below). If it is commanded in the same block as an M, S, T or B command, the control is started simultaneously.
- (d) The dwell function is valid even for machine lock but it can be completed immediately by the control parameter "#320004 Machine lock rapid".
- (e) The dwell also stops during spindle stop. It is resumed when the spindle starts to rotate again.
- (f) By the control parameter "#320005 G04 time fixed", the time designation (dwell per second) can be set, regardless of the feed mode.
- (g) This function cannot be used unless a position detection encoder has been mounted on the spindle.



- (h) If commanded in the same block as another unmodal command, the latter command will have the priority.
- (i) If commanded in the same block as the group 1 modal command (G00 to G03.1, G33), the dwell will be executed and the modal will be updated.
- (j) If commanded in the same block as a modal command that involves axis movement, such as Group 4 (G22, G23), Group 7 or 8 (G40 to G44, G49), Group 9 (G73 to G89), Group 11 (G50, G51), Group 15 (G40.1 to G42.1), Group 16 (G68, G69) or Group 19 (G50.1, G51.1), the dwell will have the priority and the other command will be ignored.
- (k) If commanded in the same block as a modal command that does not have an axis address, such as plane selection, absolute/incremental command, inch/metric changeover, feed modal, workpiece coordinate system changeover, or G61 to G64, both commands will be valid.
- (l) If the dwell is commanded during the inverse time feed mode (G93), this will be a time designation.

7. FEED FUNCTIONS

7.9 F1-digit Feed

7.9 F1-digit Feed

(1) Outline

When the control parameter "1-digit F feed valid" is ON, by designating a one-digit value after address F, the feedrate preregistered in the parameters and corresponding to the value can be commanded. The F commands include F0, and F1 to F9.

When F0 is commanded, the rapid traverse rate is set, and the same feedrate as G00 is applied. (The G modal does not change.)

When F1 to F9 is commanded, the set feedrate corresponding to the setting becomes the commanded rate.

When a value F10 or more is commanded, it will be interpreted as the normal F5-digit cutting feedrate.

The F1-digit command is valid only in the G01, G02, G03, G02.1 or G03.1 modal.

The F1-digit command can be used in the fixed cycle.

(2) Detailed description

(a) If a F1-digit command is issued in the automatic operation mode, the feedrate can be incremented and decremented by turning the manual pulse generator.

This speed change is valid when the external input (PLC) "F1-digit speed change valid signal" is selected.

(b) The feedrate increment/decrement amount follows the expression below.

$$\pm\Delta F = \frac{FMn}{K} \times (\text{number of manual pulse generator pulses}) \times (\pm 1)$$

Where, "+" expresses an increment, and "-" expresses a decrement.

FMn : This is the clamp speed. Two types of speeds (FM1 and FM2) can be set.

FM1: Maximum speed for F1 to F4

FM2: Maximum speed for F5 to F9

K : Operation constant (This is the number of FMn divisions, and is the calculated constant of the increment/decrement speed per scale of the manual handle pulse generator.)

(Note 1) The manual pulse generator is also used for the conventional axis movement, such as handle interrupt.

Select the usage purpose with the control parameters.

(c) The speeds corresponding to F1 to F9, K and FMn can be set on the Process param screen. The increment/decrement value range is F1 to F9 ... 0 to FMn

However, if the feedrate is 0 or exceeds the FM1 (FM2 for F5 to F9), the operation warning "OPW0167 F1 feed rate zero" will occur.

(3) Operation method

(a) Validate the F1-digit.

(b) Set FMn and K.

	Setting range
K	0 to 32767
FM1 (FM2)	1 to 60000 (mm/min)

(c) Set F1 to F9.

	Setting range
F1 to F9	1 to 60000 (mm/min)

7. FEED FUNCTIONS

7.9 F1-digit Feed

(4) Special notes

- (a) Use of both the F1-digit command and normal cutting feedrate command is possible in the (3)-(a) state.

(Example 1)

F0..... Rapid traverse rate
 F1 to F9..... F1-digit
 F10 or more .. Normal cutting feedrate command

- (b) F1 to F9 are invalid in the G00 mode. The rapid traverse rate will be applied.
- (c) If F0 is used in the G02, G03, G02.1 or G03.1 mode, the program error "PR0121 In F0 arc modal mode" will occur.
- (d) When F1. to F9. (with decimal point) is commanded, this will be the 1mm/min to 9mm/min feedrate instead of the F1-digit command.
- (e) When used with a metric or degree command, the feedrate set corresponding to F1 to F9 will be the command speed (mm/min or °/min).
- (f) When used with an inch command, one-tenth of the feedrate set corresponding to F1 to F9 will be the command speed (inch/min).
- (g) The number of manual pulse generator pulses is one pulse per scale, regardless of the magnification.
- (h) During the F1-digit command, the "in F1-digit command" signal and F1-digit number are output as PLC signals.
- (i) The speed designated with the E command is valid for the speed at the corner rounding or corner chamfering.
- (j) When both the F1-digit command and inverse time feed command are present, the inverse time feed command will have the priority. When the inverse time feed is canceled (G94), the F (speed) command is invalidated, so the F command must be issued again.

(5) F1-digit and G command

- (a) 01 group G command in same block as F1-digit

	Actual feedrate	Modal speed display	G modal
G0F0 F0G0	Rapid traverse rate	0	G0
G0F1 F1G0	Rapid traverse rate	1	G0
G1F0 F0G1	Rapid traverse rate	0	G1
G1F1 F1G1	Actual feedrate	1	G1

- (b) The F1-digit and unmodal command can be issued in the same block. In this case, the unmodal command is executed, and the F1-digit modal is updated simultaneously.

(6) Example of operation constant K setting

To set the one scale of the manual pulse generator to ±10mm/min.

For FM1 = 15000mm/min:

$$\Delta F = 10 = \frac{15000}{K}$$

With the above expression, K = 1500.

Thus, when the handle is turned by one scale, the feedrate will be F (1 to 4) ±10 [mm/min].

7. FEED FUNCTIONS

7.9 F1-digit Feed

(7) Conditions for validating manual pulse generator

The following conditions must be established to validate the manual pulse generator.

- Must be in cutting feed (F1 to F9).
- Must be in automatic start.
- F1-digit must be valid.
- The machine side manual handle valid switch must be ON.
- The mode must be the MDI mode, memory mode, HD operation or FD operation.
- The machine must not be locked. (Machine lock rapid traverse)
- Dry run or feed hold must not be in execution.

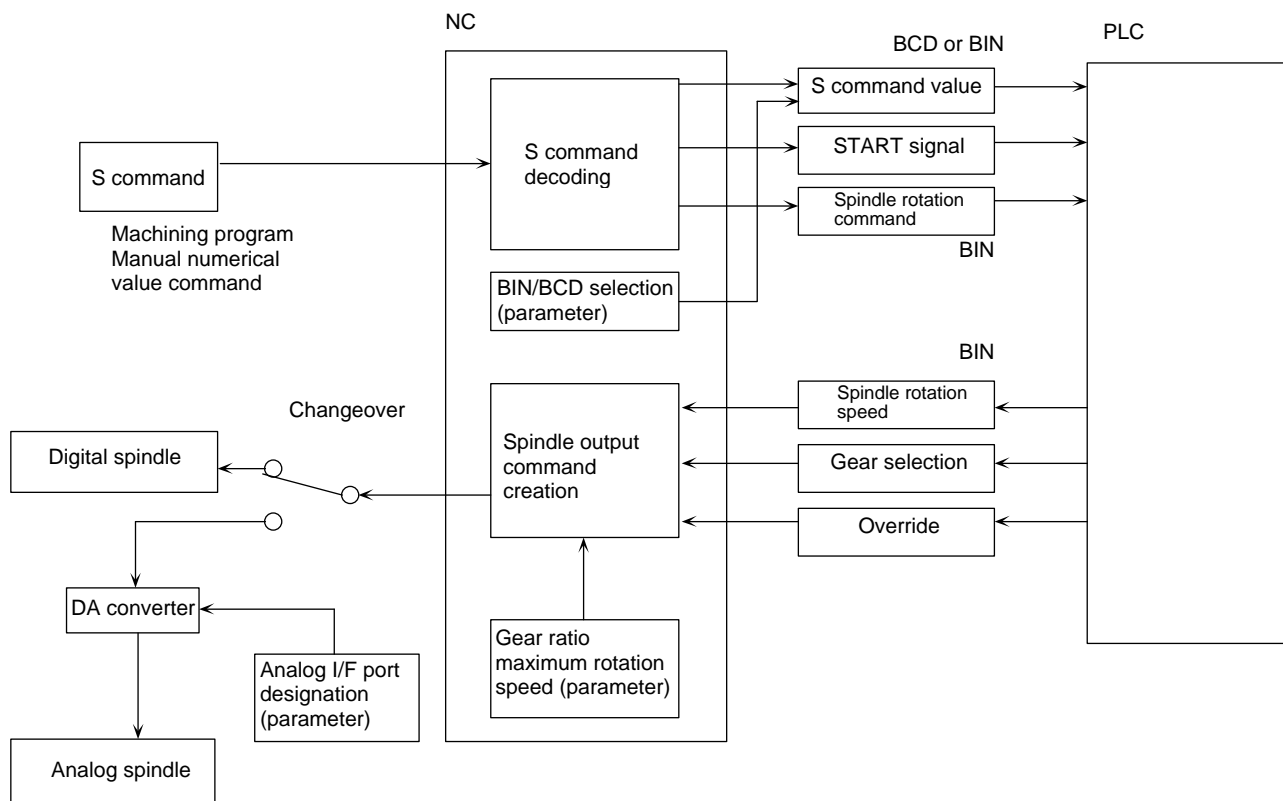
Note that this cannot be used if the handle specifications are not provided.
The first handle is used.

8. SPINDLE FUNCTIONS

8.1 Spindle Control Function

(1) Outline

With this function, the spindle speed is determined for the S command issued by the automatic operation or manual numerical value command, allowing for the override and gear ratio, and the spindle is rotated. The outline of the spindle control is shown below.



(2) Detailed description

- (a) When the spindle override is the code method, the designation is in 10% units from 50% to 120%, but when the override is the numerical value setting method, designation in 1% units from 0 to 200% is possible. The override is invalid during spindle stop input ON, tapping mode and thread cutting mode.
- (b) The number of gear levels can be commanded up to four levels.
- (c) The spindle maximum rotation speed can be set for each gear.

(3) Notes

- (a) The processing sequence and completion sequence for the S command must be grouped in the PLC.
- (b) There is no rotation speed designation by the code designation method.

8. SPINDLE FUNCTIONS

8.2 S Code Output

8.2 S Code Output

(1) Outline

When the eight-digit numbers (S00000000 to S ± 99999999) after the address S are commanded, a 32-bit binary data with symbol attached and START signal, or an 8-digit BCD signal and START signal are output to the PLC.

(2) Command format

S ± xxxxxxxx ;

(3) Detailed description

- (a) Numbers with up to eight digits can be commanded.
- (b) Only one set of S commands can be commanded in one block. If more than one set of S commands is commanded in one block, the last S command is validated.
- (c) The PLC must be built into the NC to use this function. The processing sequence and completion sequence for all S commands must be grouped in the PLC.
- (d) Either binary output or BCD output is selected with the parameters.
- (e) When commanding spindle rotation speed, the spindle rotation speed must be commanded from the PLC as shown in the drawing in Section 8.1.

8. SPINDLE FUNCTIONS
8.3 Spindle Position Control (Spindle C axis control)

8.3 Spindle Position Control (Spindle C axis control)

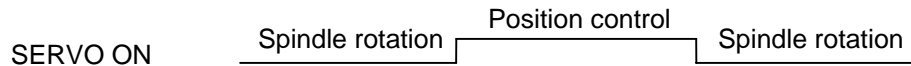
(1) Outline

With this function, servo control is carried out on the spindle amplifier and spindle motor, and the spindle position can be controlled.

(2) Detailed description

(a) When controlling the spindle position

1) The SERVO ON signal from the PLC is turned ON.



- During servo OFF : Spindle (position control not possible)
Reference point return incomplete status
- During servo ON : Spindle position control

2) After that, the command is carried out with the axis name when positioning control is set.

(Example) G00 C100. ;

(b) The coordinate system must be established by a reference point return when the spindle is changed over to position control.

The reference point can be established using one of the following three methods.

Reference point return method	Zero point return parameter setting	Spindle parameter setting
Method where the position during servo ON becomes the reference point	Axis with no reference point	The spindle does not carry out an automatic reference point return
Method using the automatic reference point return function of the spindle		The spindle carries out an automatic reference point return
Dog-type reference point return	Dog-type reference point return	The spindle does not carry out an automatic reference point return

- (c) A spindle amplifier bus-connected with the control unit and a position detector are necessary to carry out the spindle position command.
- (d) The spindle can also be set so that it does not carry out an automatic reference point return by an external signal.
- (e) The resolution depends on the position detector resolution. Refer to the sections on "Minimum setting units" and "Speed commands" for the setting units, speed, etc.
- (f) A program error will occur if a position control command is carried out during servo OFF or orientation.

9. TOOL FUNCTIONS

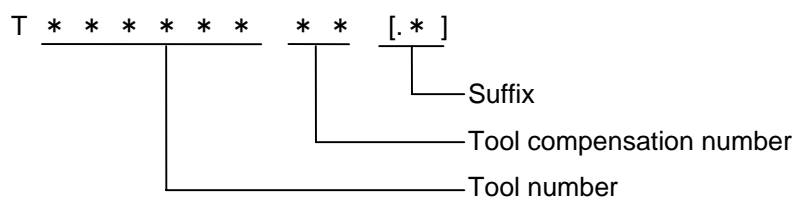
9.1 T Commands

(1) Outline

The tool number and tool compensation number can be commanded using the T commands (a number up to eight digits following address T). Only one set of T commands can be commanded in one block.

(2) Command format

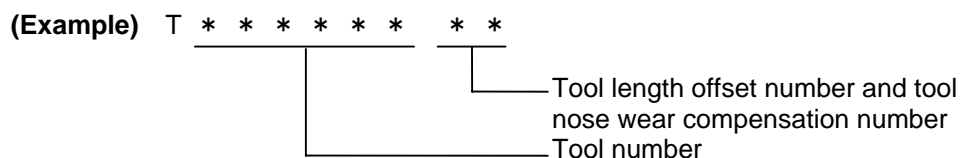
(Example) When the T command has eight digits and the tool compensation number command has two digits



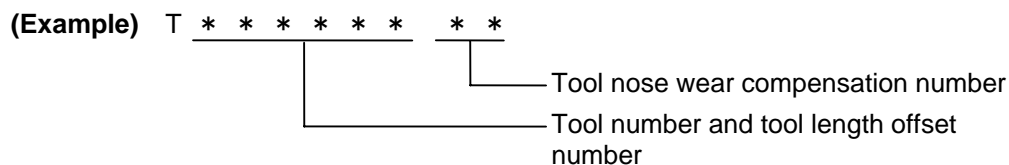
(3) Detailed description

- (a) The number of digits in the T command and the number of digits in the tool compensation number below it are determined by the parameters.
- (b) If the number of digits in the tool compensation number is 0, all of the digits can be used for the tool number command.
A suffix (omission possible) can be commanded at the end of the T command.
- (c) The tool compensation consists of tool length offset and tool nose wear compensation. One of the following three methods can be selected by the parameters for designating these compensation numbers.
(In this example, the T command has eight digits, and the tool compensation number has two digits. There is no suffix.)

- 1) Method by which the compensation numbers of the tool length and tool nose wear are designated by the T command low-order digits.
 High-order digits The tool number is designated.
 Low-order digits Both the tool length offset and tool nose wear compensation are designated by the same compensation number.



- 2) Method by which the tool number and tool length offset number are designated by the T command high-order digits.
 High-order digits The tool number and tool length offset number are designated by the same compensation number.
 Low-order digits The tool nose wear compensation number is designated.



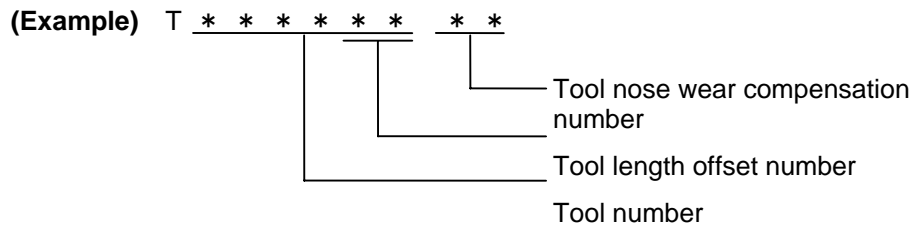
9. TOOL FUNCTIONS

9.1 T Commands

- 3) Method by which the tool length offset number is designated by the last two digits of the T command high-order digits

High-order digits The tool number is designated by all the high-order digits.
The tool length offset number is designated by the last two digits of the high-order digits.

Low-order digits The tool nose wear compensation number is designated.



- (d) The tool number, START signal and suffix are output to the PLC (the tool compensation number is not output) by the T command. The suffix use is random, but can be used to designate, etc., the cutting edge of the tool having two cutting edges.
- (e) Whether the tool number is output as a 32-bit binary value or as a BCD output is set by the parameters.
- (f) The processing sequence and completion sequence for all T commands must be composed on the PLC side.

10. MISCELLANEOUS FUNCTIONS

10.1 Miscellaneous Functions

(1) Outline

When the number up to eight digits (M00000000 to M99999999) after the address M are commanded, a 32-bit binary data and START signal, or an 8-digit BCD signal and START signal are output to the PLC.

(2) Command format

Mm1	Mm2	Mm3	Mm4 ;
-----	-----	-----	-------

m1, m2, m3, m4: Number up to eight digits (00000000 to 99999999)

(3) Detailed description

- (a) Up to four sets of M commands can be simultaneously commanded in one block. If five or more sets of M commands are commanded in one block, the last four M commands are validated.
- (b) Separate from the above signals, respective dedicated individual signals are also output for the following signals.
 - M00 Program stop
 - M01 Optional stop
 - M02 Program end
 - M30 Program end
- (c) To use this function, the processing sequence and completion sequence for all M commands from M00000000 to M99999999 must be grouped in the PLC side (excluding M98, M99). In this case, because the code signals and START signals of all the M commands in the same block are simultaneously transferred from the CNC to the PLC, high-speed machine control can be carried out by the PLC processing sequence.
- (d) The output signals are either an 8-digit BCD code and START signal or a 32-bit binary data with symbol attached and START signal. Either BCD output or binary output is set in the parameters.

10. MISCELLANEOUS FUNCTIONS

10.2 M Individual Output

10.2 M Individual Output

(1) Outline

These functions are output to the PLC when M00, M01, M02 and M30 are commanded during automatic operation, or by a manual numerical value command, and are turned OFF by a miscellaneous function completed signal and reset signal from the PLC.

(2) Command format

Machining program	M individual output	Meaning	Reply to the CNC
M00	M00	Program stop	Fin1 or Fin2
M01	M01	Optional stop	Fin1 or Fin2
M02	M02	Program end	Reset
M30	M30	Program end	Reset

(3) Detailed description

- (a) As pre-read of M00, M01, M02 and M30 is prohibited, the next block is not read into the pre-read buffer.
- (b) When there is a movement command/dwell in the same block, the M individual output is output after the movement command/dwell is completed.

1) Program stop: M00

- When this miscellaneous function is read, reading to the next block is stopped. The NC functions only stop the program reading. Whether the machine side functions such as spindle rotation and coolant are stopped differ according to the machine.
- The restart is carried out by the automatic start signal.
- Whether a reset is carried out by M00 depends on the machine specifications.

2) Optional stop: M01

- The program reading stops when the optional stop signal of the machine operation board is turned ON, and this M01 command is read.
- Restart is carried out by the miscellaneous function completed signal, not by the automatic start signal. (When restarting with the automatic start button, it is necessary to group the sequence in the user PLC.)
- The M01 command is ignored if the optional stop signal is OFF.

3) Program end: M02 or M30

- This command is normally used in the last block where machining is completed.
- Normally this command is reset as a reply after it is output. The contents of the command position display counter are not cleared by this reset, but the modal command and compensation amount are canceled. Refer to the section on the NC reset for details.
- The restart is carried out by the automatic start signal.

10.3 2nd Miscellaneous Functions

(1) Outline

By commanding an address code (with a number up to eight digits) that does not overlap with the axis name from any of A, B, C, H, U, V, W, X, Y or Z, a 32-bit binary data and START signal, or a BCD signal and START signal are output to the PLC.

(2) Detailed description

- (a) The address to be used is defined by the parameters.
- (b) A command with a decimal point is also possible. If the decimal point is omitted, the output magnification can be set with the parameters.
- (c) A processing sequence and completion sequence are necessary for all 2nd miscellaneous functions.
- (d) The same address cannot be used in the axis name and 2nd miscellaneous function.
- (e) If A is designated in the 2nd miscellaneous function address, the following functions cannot be used.
 - Linear angle command
 - Geometric command
- (f) Whether to output a binary data or BCD signal is set in the parameters.

11. TOOL COMPENSATION

11.1 Tool Length Offset; G43/G44

(1) Outline

By this command, the position of the end point of the movement command is changed to the position that was offset the previously set tool offset amount only, and movement control is carried out.

Using this function, the distance difference, due to the tool length, between the program coordinate position and the actual machine tool tip position is offset, and the program and operation can be made more efficient.

(2) Command format

- (a) Offsetting is carried out for the rotation tool.
- (b) In 1-axis offset, a random axis is offset. Because the relation between the offset number and offset axis is not assigned beforehand, which axis is to be offset must be designated when the command is issued.

The offset number is designated by an H command.

G43 Zz1 Hh1 ;	Besides the Z axis, tool length offset can be carried out for the all axes (such as X and Y) that can be controlled by the system.
Selection axis	
Offset number	
G44 ;	Tool length offset cancel

- (c) The offset is canceled by the following G commands.

G44 ;

G43 H0 ;

(3) Detailed description

(a) Tool length offset movement amount

- 1) The movement amount is calculated using the expression below when the G43 or G44 tool length offset start command or G44 tool length offset cancel command is assigned.

G43 Zz Hh1 ; $z + \lambda h1$ (Offset in +direction by a value equivalent to offset amount)

G44 Zz ; Offset amount cancel

$\lambda h1$: Offset amount of offset number h1

The actual end point of the coordinates is offset from the programmed movement command end point coordinates by an amount calculated by the above expression, regardless of whether absolute value or incremental value commands are used.

- 2) The G44 (tool length offset cancel) mode is established when the power is turned ON and after the reset and M02 command have been executed.

11. TOOL COMPENSATION

11.1 Tool Length Offset; G43/G44

(b) Offset numbers

- 1) The offset amounts differ according to the offset types.
 - Type A (When the option parameter = 1)
When G43 Hh1; is commanded, offset amount $\lambda h1$ commanded by offset number h1 will be common to tool length offset amount, tool radius compensation amount, shape offset amount and wear offset amount.
 - Type B (When the option parameter = 0)
When G43 Hh1; is commanded, offset amount $\lambda h1$ commanded by offset number h1 will be the shape offset amount + wear offset amount.
- 2) The valid range of the offset numbers depends on the specification configuration (number of offset sets).
Refer to "11.5 Number of compensation sets".
- 3) An alarm occurs when the offset number commanded exceeds the range of the specifications.
- 4) The tool length offset is canceled by commanding H0.
- 5) An offset number commanded in the same block as G43 will subsequently remain valid as a modal.

(Example)

```
G43  Zz1  Hh1  ; Tool length offset is carried out by h1.
      ⋮
G45  Xx1  Yy1  Hh6  ;
      ⋮
G44  Zz2  ;          Tool length offset is canceled.
      ⋮
G43  Zz2  ;          Tool length offset is carried out again by h1.
```

- 6) When G43 is further commanded while G43 is still modal, offset carried out by an amount equivalent to the difference in the data of the offset number.

(Example)

```
G43  Zz1  Hh1  ; z1 + ( $\lambda h1$ ) movement
      ⋮
G43  Zz2  Hh2  ; z2 + ( $\lambda h2 - \lambda h1$ ) movement
```

11. TOOL COMPENSATION

11.1 Tool Length Offset; G43/G44

(c) Tool length offset valid axes

- 1) When the control parameter (T leng offset Z fix) is ON, tool length offset applies at all times to the Z axis.
- 2) When the control parameter (T leng offset Z fix) is OFF, it depends on the axis address assigned in the same block as the G43 command. When two or more axes are simultaneously commanded, the sequence of priority is as follows: Z axis > Y axis > X axis.

(Example)

```
G43 Xx1 Hh1 ; +offset for X axis
  ⋮
G44 Xx2 ;
  ⋮
G43 Yy1 Hh2 ; +offset for Y axis
  ⋮
G44 Yy2 ;
  ⋮
G43 □ α1 Hh2 ; +offset for additional axis
  ⋮
G44 □ α2 ;
  ⋮
G43 Xx3 Yy3 Zz3 Hh3 ; +offset for Z axis
  ⋮
G44 ;
```

Additional axes are handled in accordance with the setting of the parameters.
When tool length offset is to be commanded with a rotation axis, set the name of the rotation axis into any of the parallel axes.

- 3) When there is no axis address in the same block as G43, the Z axis is validated.
- 4) The Z and X axes are offset when the following is commanded:

```
G43Zz1Hh1 ;
G43Xx1Hh2 ;
```

(d) Operations when other commands are assigned while tool length offset is modal

- 1) Automatic reference point return (G28 high-speed/dog-type, G30 high-speed)
If automatic reference point return is started for an axis in the tool length offset modal, the tool length offset will be canceled and the offset amount will be cleared when the reference point is reached.
- 2) Manual dog type reference point return A
If manual dog-type reference point return is started for an axis in the tool length offset modal, the tool length offset will be canceled and the offset amount will be cleared when the reference point is reached.
Note that the Modal screen display will be updated when the system is restarted.
- 3) Manual high-speed reference point return A
If manual high-speed reference point return is started for an axis in the tool length offset modal, the tool length offset will not be canceled when the reference point is reached.
Note that the tool length offset can be canceled with the parameter (110135 Cancel G43 MDL M-REF). In this case, the Modal screen display will be updated when the system is restarted.
- 4) When a movement command is assigned to the G53 machine coordinate system, movement follows to the machine position with the tool length offset amount canceled. When returning to one of the workpiece coordinate systems (G54 to G59), the tool returns to the coordinates which have been shifted again by an amount equivalent to the tool offset.

11. TOOL COMPENSATION

11.1 Tool Length Offset; G43/G44

(e) Operation during tool life management

If the tool length offset is commanded when the control parameter (T-life manage valid) is set, the actual offset amount becomes as follows according to the settings on the Tool Life Management screen.

1) Number offset method

When the tool length offset number is set on the Tool Life Management screen, the H (offset number) designation is substituted for the set number, and tool length offset is carried out.

(Example)

Tool Life Management screen setting: When the tool length offset number of tool number t1 is h2.

```
G43  Zz1  Hh1  ; Tool length offset is carried out for h1 by tool number λh2.  
      :  
G44  Zz2  ;
```

2) Offset amount direct offset method

When the tool length offset amount is set on the Tool Life Management screen, offset is carried out by the set offset amount instead of offset amount indicated by H (offset number).

(Example)

Tool Life Management screen setting: When the tool length offset number of tool number t1 is λ1.

```
G43  Zz1  Hh1  ; The tool length offset of λ1 amount is carried out.  
      :           (λh1 offset amount is not used.)  
G44  Zz2  ;
```

11. TOOL COMPENSATION

11.1 Tool Length Offset; G43/G44

(4) Example

(a) Combined operation with tool length measurement

(Example 1.) Tool length offset by combination with tool length measurement type I

```
G28 X0 Y0 Z0;
T01;
T02 M06;
G91 G00 G43
Z2.0 H01;
```

(Note)
When the tool compensation amount is $H01 = -450.000$, etc., it is set by the negative value.

(Example 2.) Tool length offset by combination with tool length measurement type II

```
G28 X0 Y0 Z0;
T01;
T02 M06;
G91 G54 G00 G43
Z2.0 H01;
```

(Note)
When the tool compensation amount is $H01 = +150.000$, etc., it is set by the positive value.

(b) When offsetting is carried out with the distance between the reference point and workpiece as a reference

```
G28 X0 Y0 Z0;
T01;
T02 M06;
G91 G00 G43
Z2.0 H01;
```

(Note)
When the tool compensation amount is $H01 = -450.000$, etc., it is set by the negative value.

11. TOOL COMPENSATION
11.1 Tool Length Offset; G43/G44

(5) Notes

To set the tool compensation amount (tool length and tool radius) during automatic operation, set during the single block stop.

The following will occur when the compensation amount is set during automatic operation.

State during automatic operation	When tool length offset amount is set	When tool radius compensation amount is set
During single stop	Whether to validate the setting from the next block or from the next H No. command can be set with parameter (110051 Op Tolcomp ofs valid).	Valid from the next block.
Other than the above	The set compensation amount is validated from the tool compensation command block several blocks ahead of the block being executed.	

11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

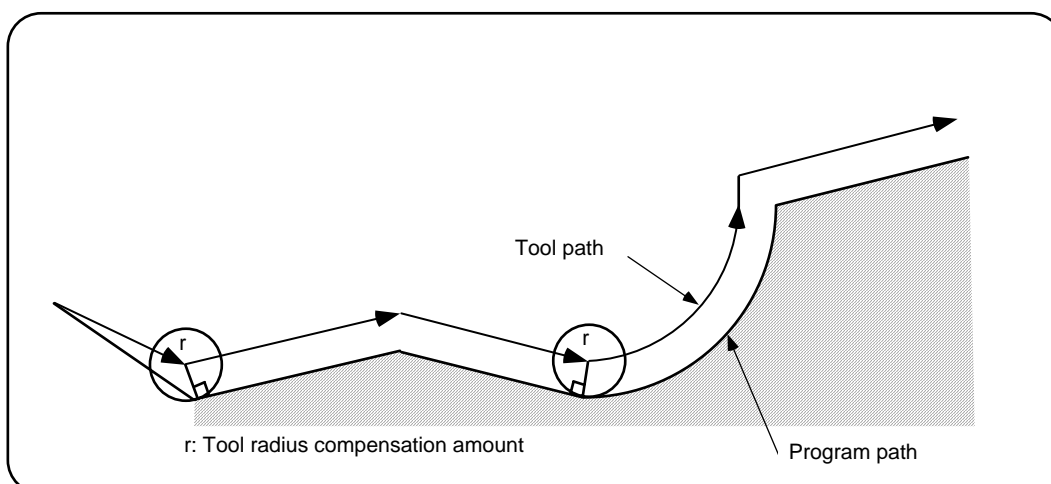
11.2.1 Outline

This commands the function that compensates the tool radius. This function compensates the actual tool center path in respect to the program path on the outside or inside by the tool radius amount, using a combination of G commands and D designations.

Because path calculation is carried out by the intersection calculation method, overcutting does not occur at the inside of corners.

Compensation control operation is carried out from the G41 or G 42 commanded blocks. In the tool radius compensation mode, up to five blocks of the program are pre-read, including blocks with no movement amount. Of those blocks, a tool radius interference check is carried out on the shape of up to three blocks with movement amount.

G code	Function
G38	Vector change during tool radius compensation
G39	Corner arc during tool radius compensation
G40	Tool radius compensation cancel
G41	Tool radius compensation left command
G42	Tool radius compensation right command



11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

11.2.2 Command format

<u>G17</u>	<u>G01</u>	<u>G41</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Dd1</u> ;
Compensation plane	Cutting	Compensation command	Movement axis	Movement axis	Compensation number

(1) Compensation plane

The compensation plane, movement axis and the next advance direction vector follow the plane selection command designated from G17 to G19.

- G17 XY plane, X, Y, I, J
- G18 ZX plane, Z, X, K, I
- G19 YZ plane, Y, Z, J, K

(2) Compensation command

Command format	Function	Remarks
G40X_Y_ ;	Tool radius compensation cancel	
G41X_Y_ ;	Tool radius compensation (left)	
G42X_Y_ ;	Tool radius compensation (right)	
G38I_J_ ;	Compensation vector change and hold	Command valid only in the radius compensation mode
G39X_Y_ ;	Corner changeover	

(3) Compensation number

- Number of compensations sets differs according to the model.
- D01 to Dn
- In tool radius compensation, H commands are ignored and only D commands are validated.

11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

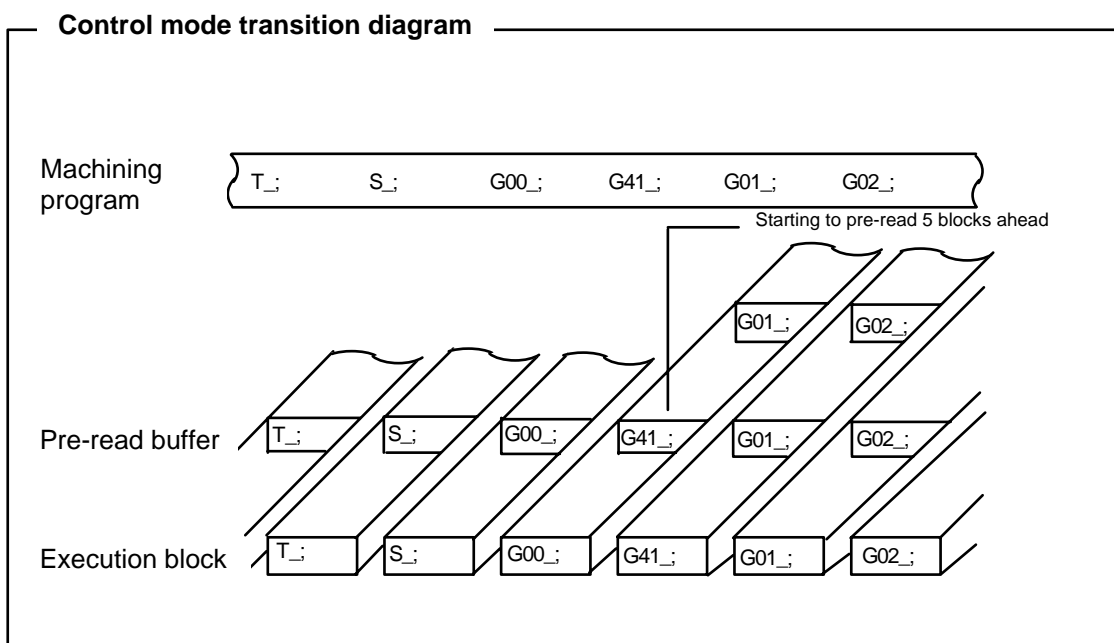
11.2.3 Tool radius compensation operations

(1) Tool radius compensation cancel status

- (a) The tool radius compensation cancel mode is established by any of the following conditions.
 - 1) After the power has been turned ON
 - 2) After the reset button on the CNC unit has been pressed
 - 3) After the M02 or M30 command with reset function has been executed
 - 4) After the compensation cancel command (G40) has been executed
- (b) The offset vectors are zero in the compensation cancel mode, and the tool center path matches the program path.
- (c) Programs including tool radius compensation must be terminated in the compensation cancel mode.

(2) Tool radius compensation start (start-up)

- (a) Tool radius compensation starts when all the following conditions are met in the compensation cancel mode.
 - 1) The G41 or G42 command has been issued.
 - 2) The compensation number of the tool radius compensation is $0 < D \leq$ maximum compensation number.
 - 3) The movement command is any command except an arc command.
- (b) At the start of compensation, regardless of continuous operation and single-block operation, three blocks of movement commands are always pre-read. If there are not three blocks of movement commands, maximum of five blocks are pre-read and compensation calculation is carried out.

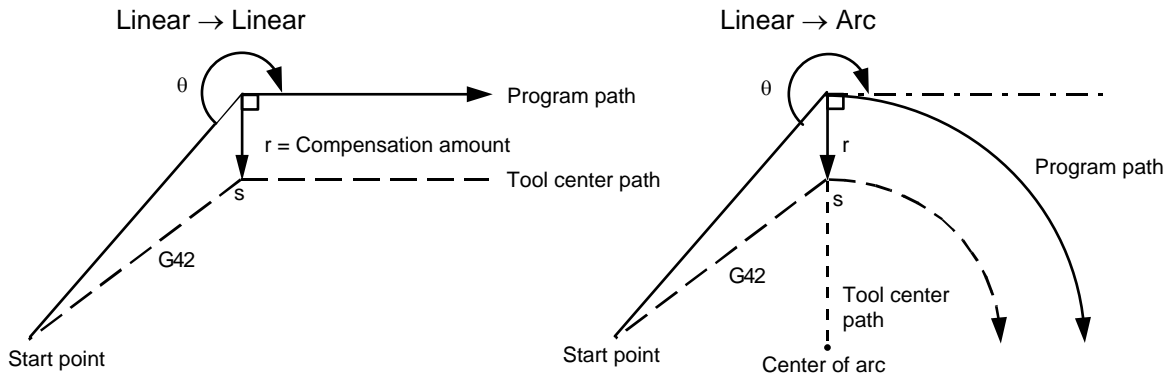


- (c) There are two ways of starting the compensation operation: type A and type B. The type can be selected or de-selected by the parameter. This type is used in common with the compensation cancel type.
- (d) In the following explanatory diagram, "S" denotes the single block stop point.

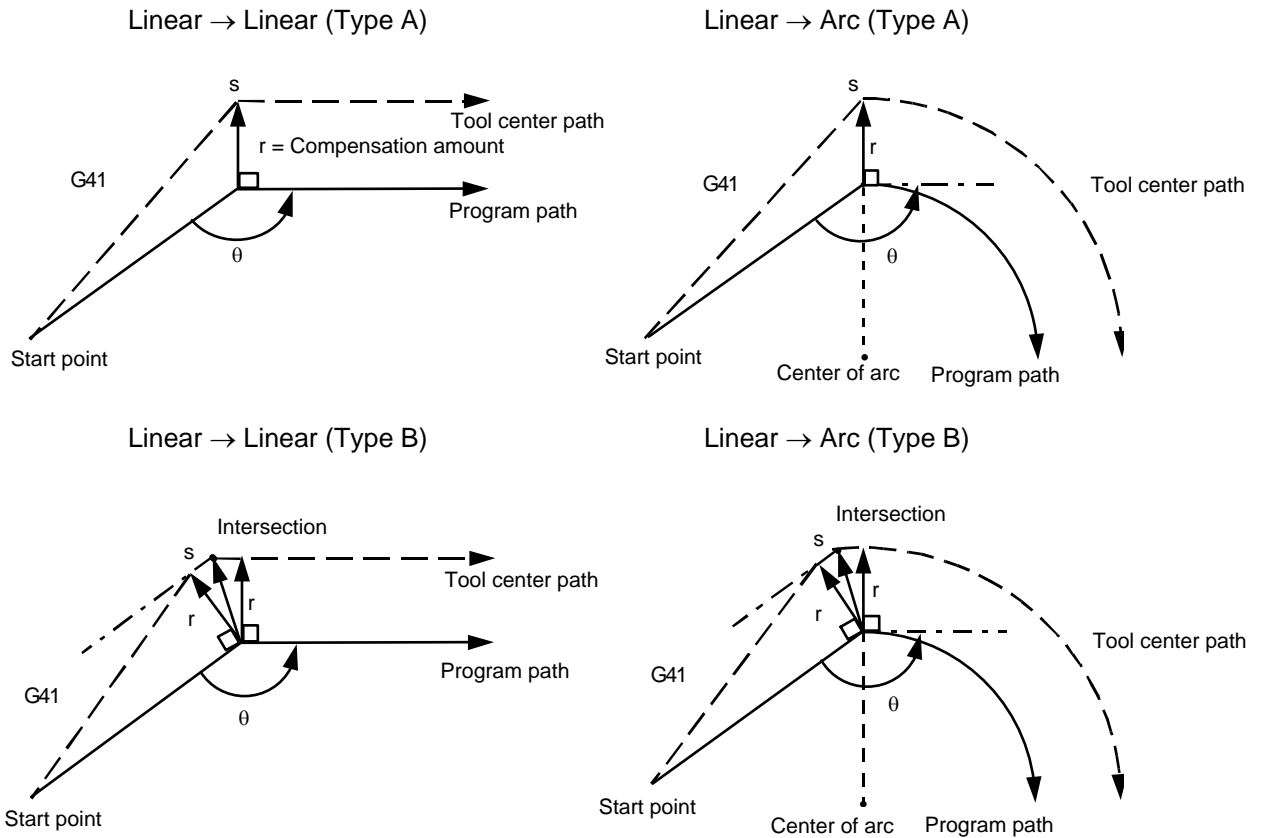
11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(3) Start operation for tool radius compensation

(a) Machining an inside corner



(b) Machining an outside corner (obtuse angle) (Type A or B can be selected by parameter)
 $[90^\circ \leq \theta < 180^\circ]$

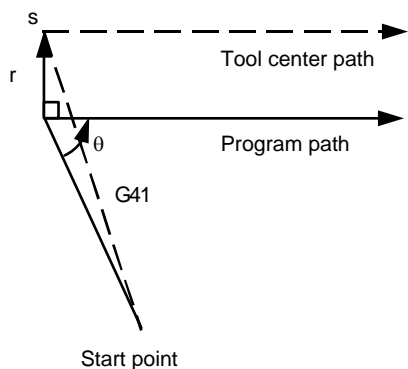


(Note) Cutting in may occur if the compensation amount is larger than the arc radius.

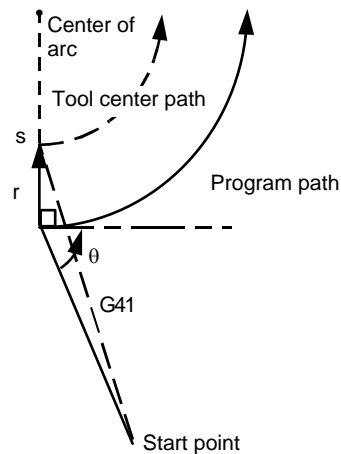
11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(c) Machining an outside corner(acute angle) (Type A or B can be selected by parameter)
 $[\theta < 90^\circ]$

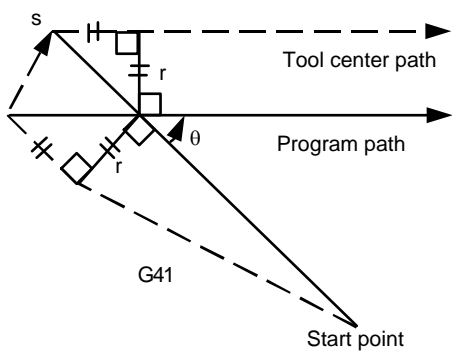
Linear → Linear (Type A)



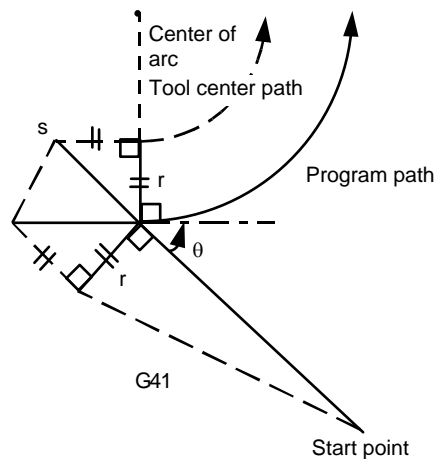
Linear → Arc (Type A)



Linear → Linear (Type B)



Linear → Arc (Type B)



(Note) An alarm will occur without the radius compensation starting up if cancel G40 is commanded without an axis movement command in the compensation plane after a G41/G42 command.

11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(4) Operations in compensation mode

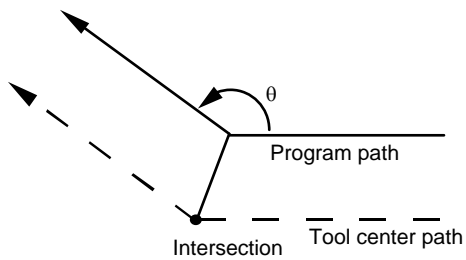
Compensation is valid both for positioning and for interpolation commands such as circular and linear interpolation.

Even if the same compensation command (G41/G42) is issued in a compensation mode, the command will be ignored.

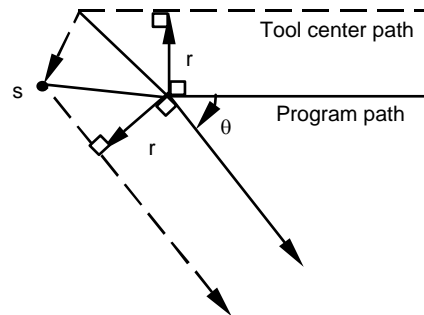
When 4 or more blocks not accompanying movement are commanded continuously in the compensation mode, overcutting or underwriting will result.

(a) Machining an outside corner

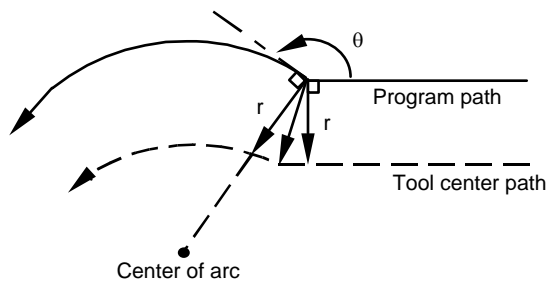
Linear → Linear ($90^\circ \leq \theta < 180^\circ$)



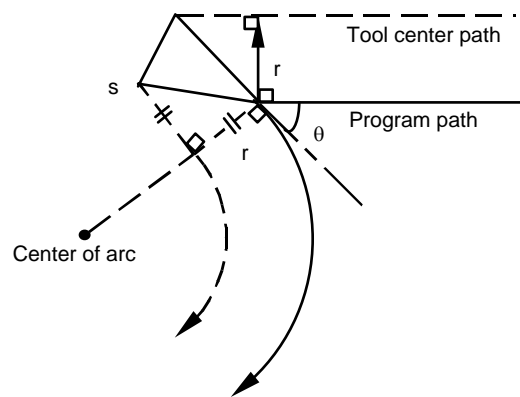
Linear → Linear ($0^\circ < \theta < 90^\circ$)



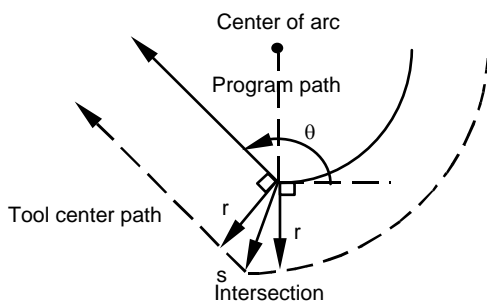
Linear → Arc ($90^\circ \leq \theta < 180^\circ$)



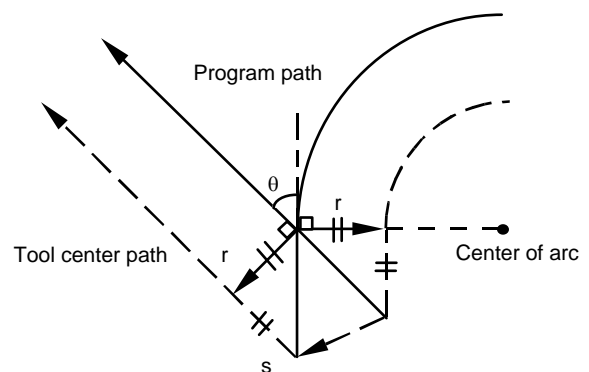
Linear → Arc ($0^\circ < \theta < 90^\circ$)



Arc → Linear ($90^\circ \leq \theta < 180^\circ$)

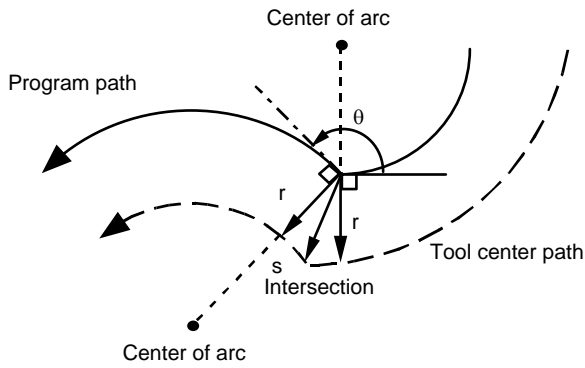


Arc → Linear ($0^\circ < \theta < 90^\circ$)

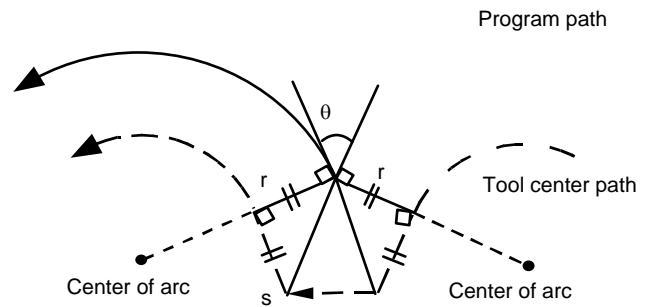


11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

Arc → Arc ($90^\circ \leq \theta < 180^\circ$)

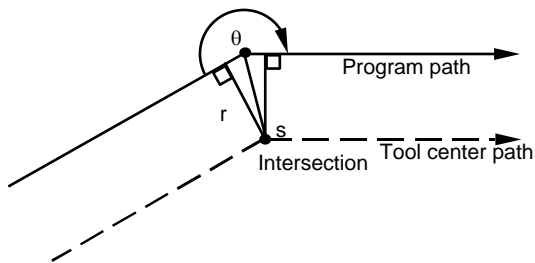


Arc → Arc ($0^\circ < \theta < 90^\circ$)

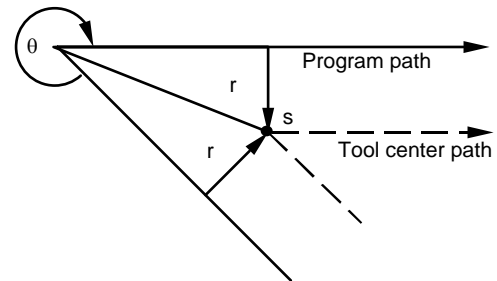


(b) Machining an inside corner

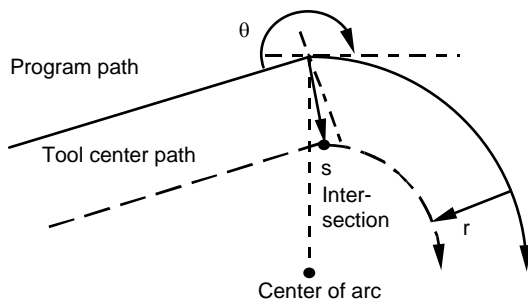
Linear → Linear (Obtuse angle)



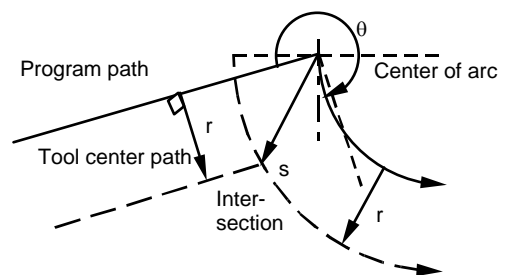
Linear → Linear (Acute angle)



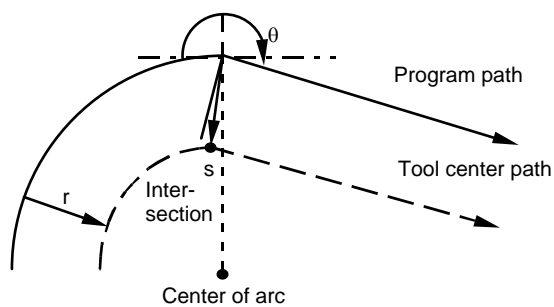
Linear → Arc (Obtuse angle)



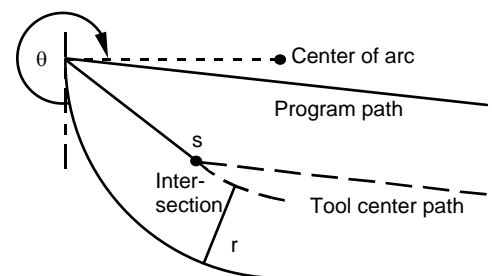
Linear → Arc (Acute angle)



Arc → Linear (Obtuse angle)

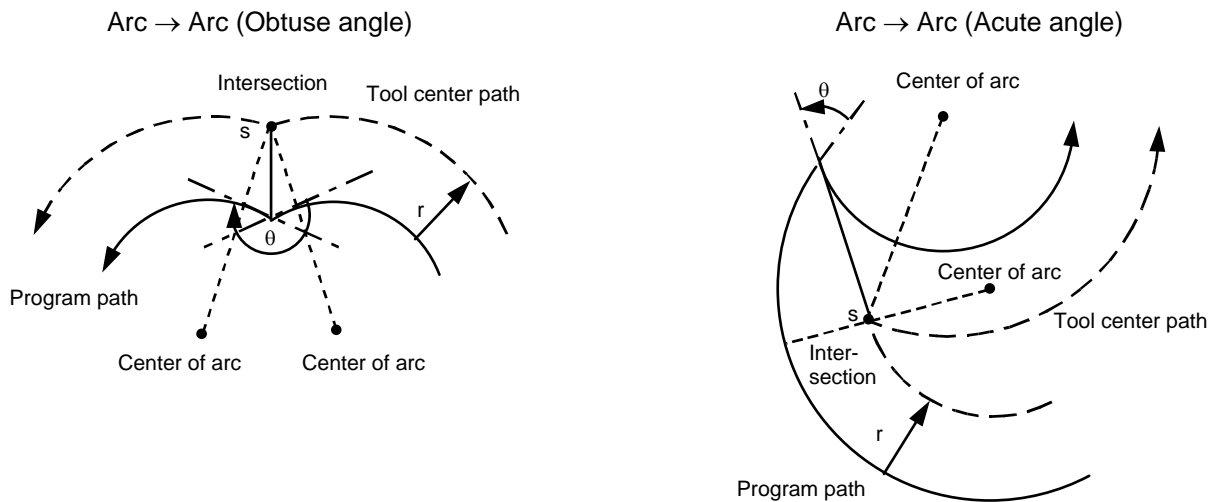


Arc → Linear (Acute angle)



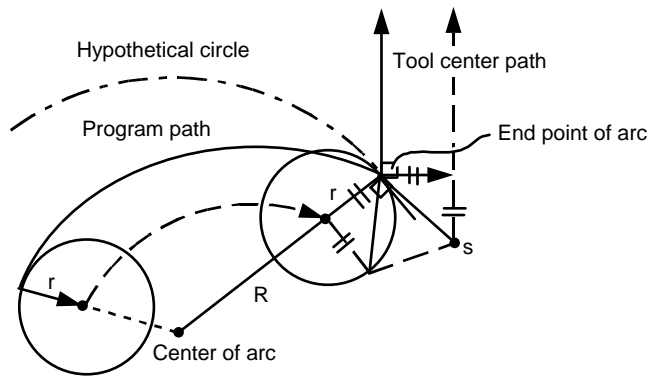
11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39



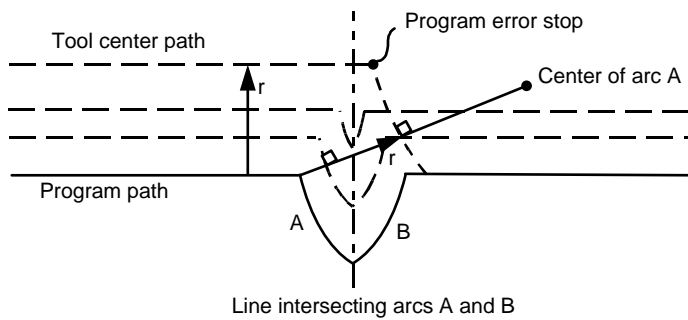
(c) When the arc end point is not on the arc

For a spiral arc command The interval from the start point to the end point of the arc command is interpolated as a spiral arc.
 For a normal arc command If the error after compensation is within the value of the parameters, it is interpolated as a spiral arc.



(d) When the inner intersection point does not exist

In an instance such as that shown in the figure below, the intersection point of arcs A and B may cause to exist due to the compensation amount.



11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(5) Tool radius compensation cancel

If either of the following conditions is met in the tool radius compensation mode, the tool radius compensation will be canceled.

- a. The G40 command has been executed.
- b. The compensation number (D00) has been executed.

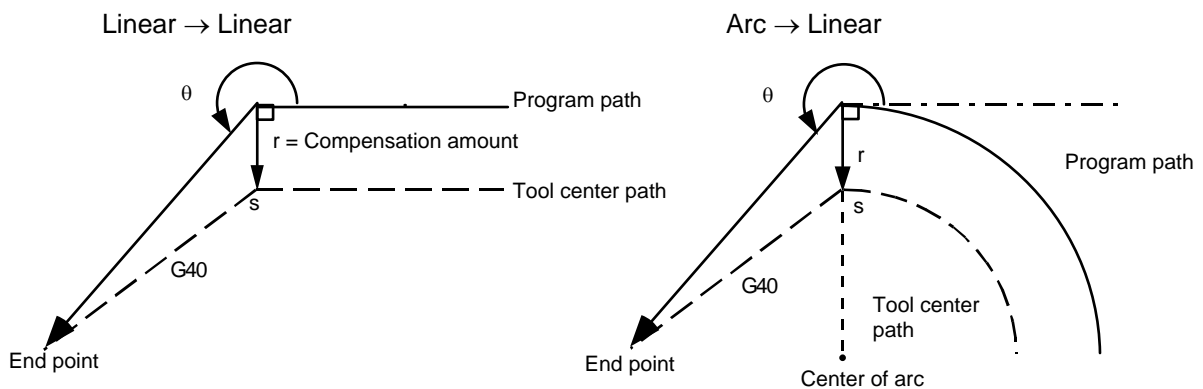
However, the movement command must be a command which is not an arc command.

If the compensation is canceled by an arc command, alarm results.

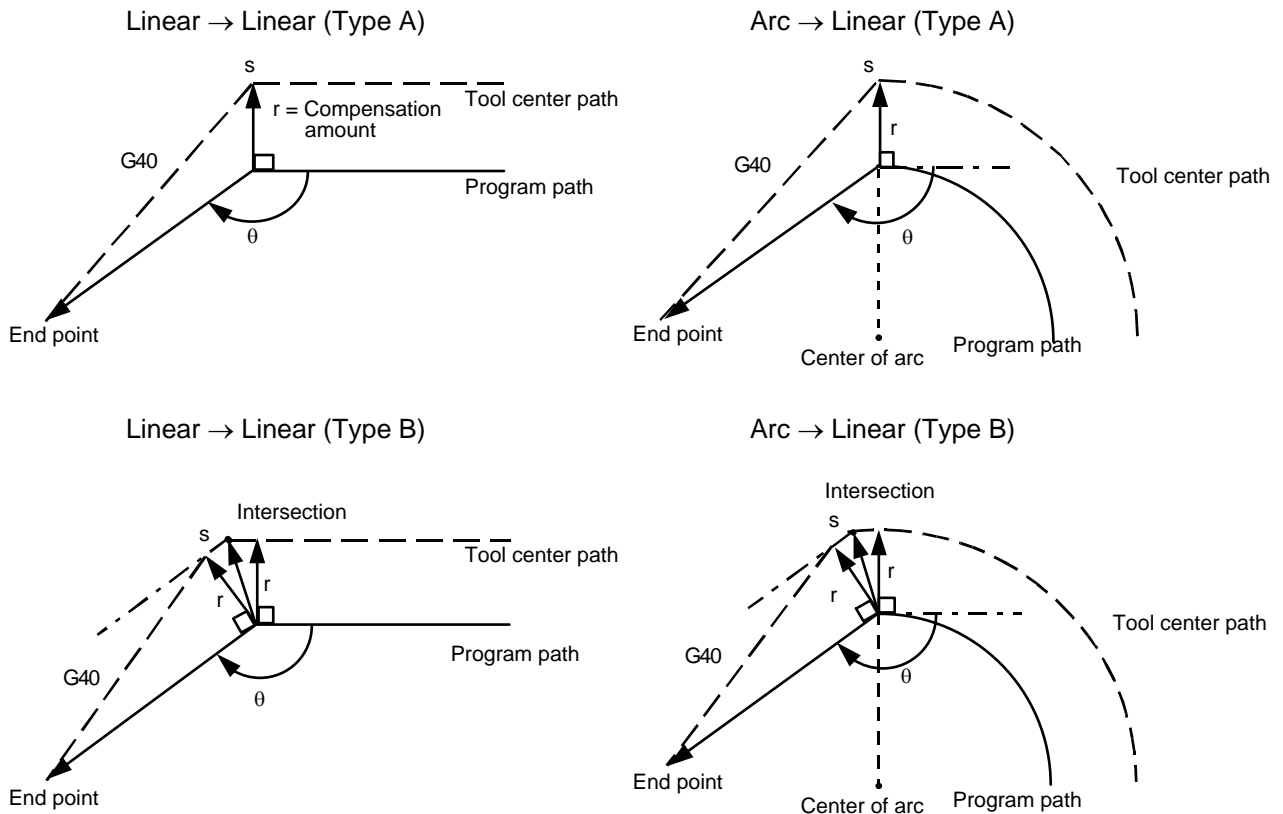
The cancel mode is established after the compensation cancel command has been read, 5-block pre-reading is suspended and 1-block pre-reading is applied instead.

(6) Tool radius compensation cancel operation

(a) Machining an inside corner



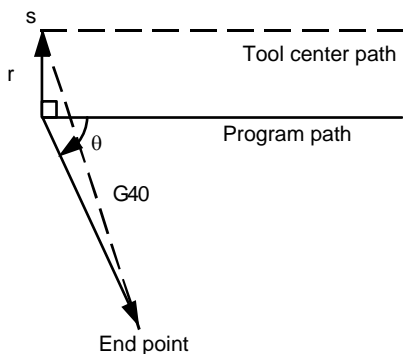
(b) Machining an outside corner (obtuse angle) (Type A or B can be selected by parameter)



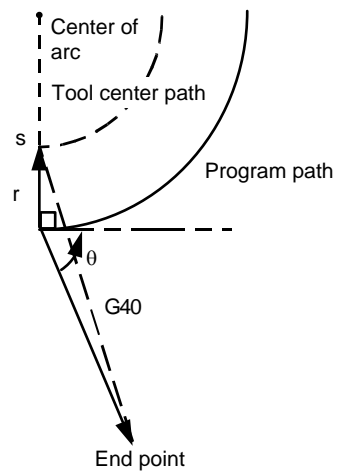
11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(c) Machining an outside corner (acute angle) (Type A or B can be selected by parameter)

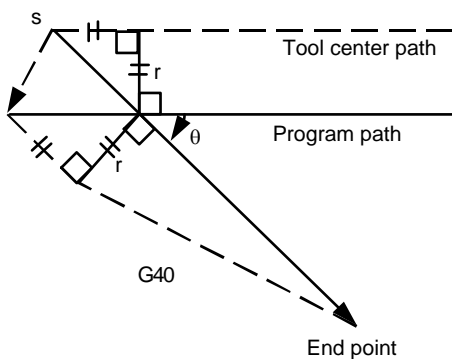
Linear → Linear (Type A)



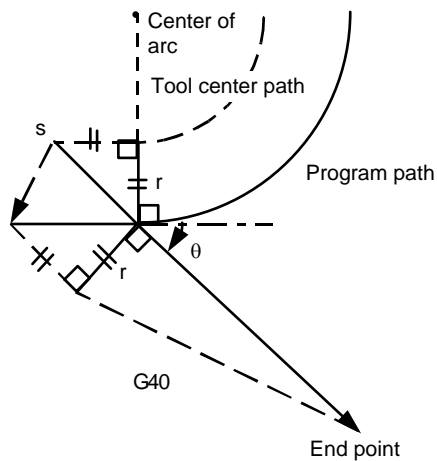
Arc → Linear (Type A)



Linear → Linear (Type B)



Arc → Linear (Type B)



11. TOOL COMPENSATION

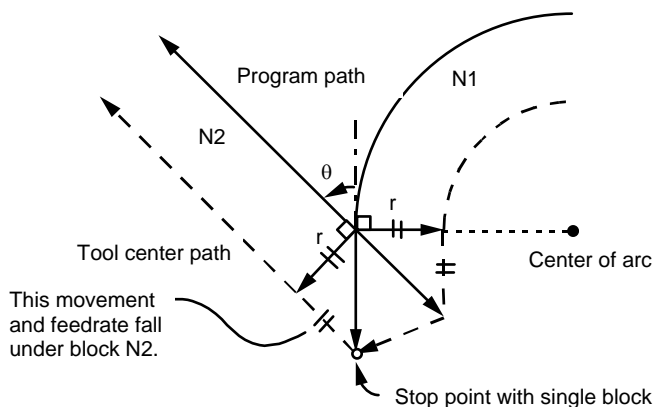
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(7) Corner movement

When a multiple number of compensation vectors are created at the seams between movement command blocks, the tool will move in a straight line between those vectors. This action is called corner movement.

This action is called corner movement.

The movement for turning the corner is related to the movement of the joining blocks. Thus, the operation in the single block operation is executed as a single block consisting of the previous block, the joining movement, and the corner movement. The remaining joining movement and the next block is executed as one block at the next start.



11. TOOL COMPENSATION

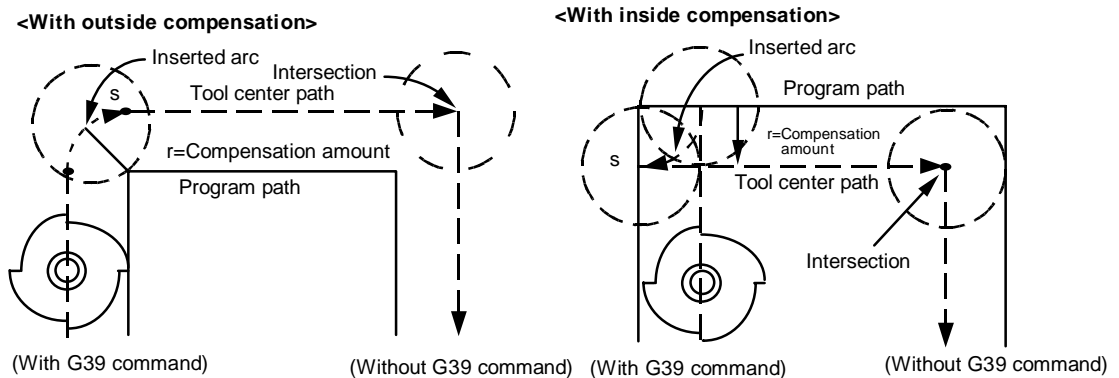
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

11.2.4 Other commands and operations for tool radius compensation

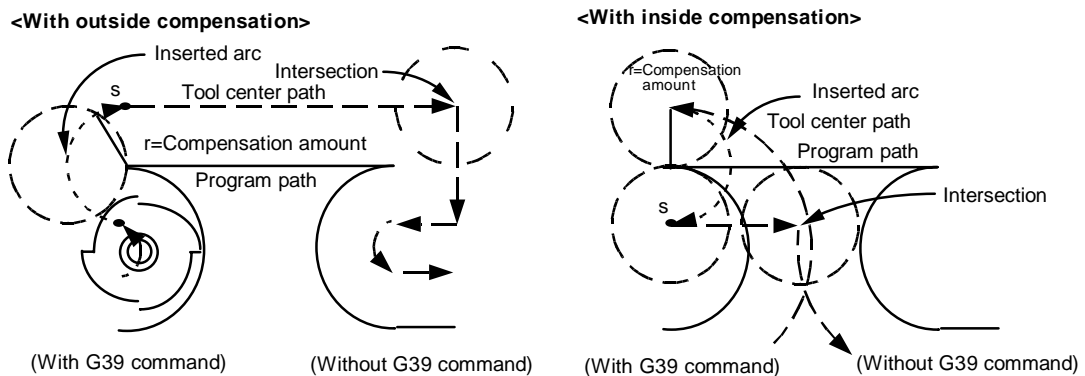
(1) Inserting a corner arc

When the G39 command (corner arc) is assigned without carrying out intersection calculation of the workpiece corner, the arc with the compensation amount as its radius is inserted. Cutting will occur during inside compensation.

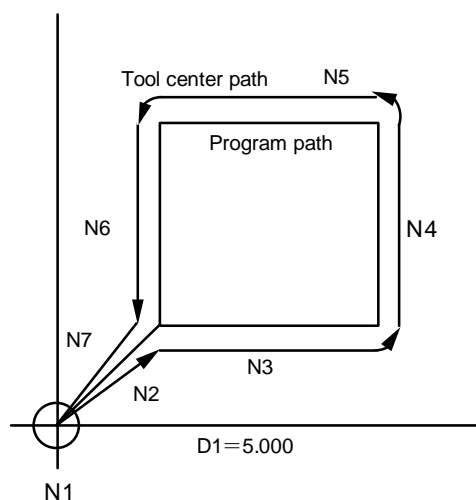
(a) For linear



(b) For arc



(Example)



```

N1 G28X0Y0;
N2 G91G01G42X20.Y20.D1F100;
N3 G39X40.;
N4 G39Y40.;
N5 G39X-40.;
N6 Y-40.;
N7 G40X-20.Y20.;
N8 M02.;
    
```

(Note) The corner arc will not be inserted when the G39 independent block is commanded.

11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(2) Changing/retaining the compensation vectors

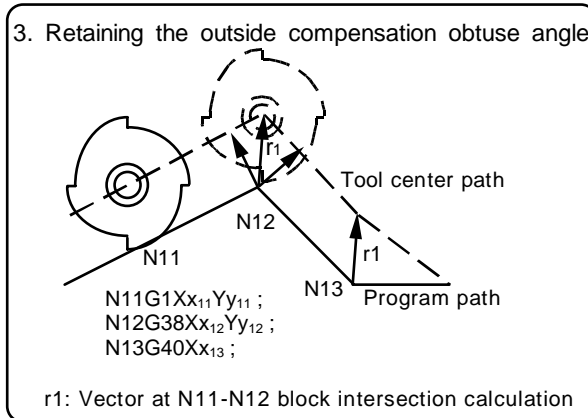
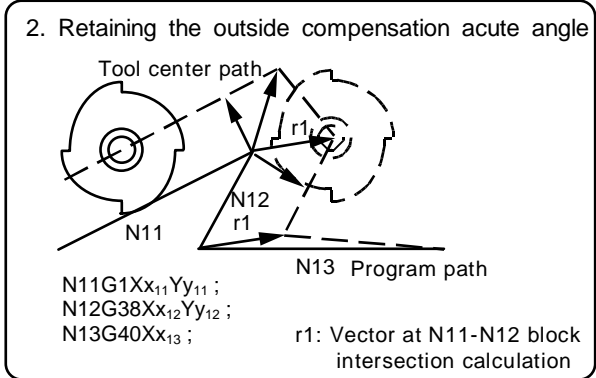
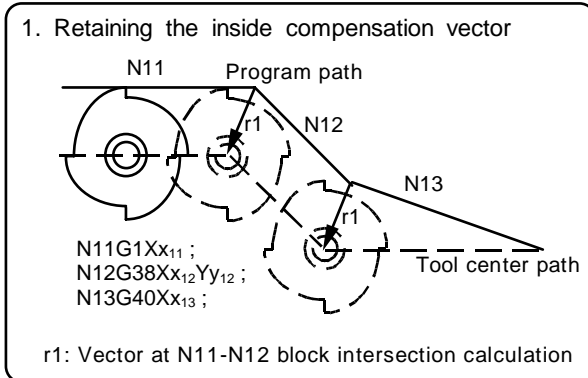
The compensation vectors can be changed or retained during tool radius compensation by the G38 command.

(a) Retaining the vectors:

When G38 is assigned in a block with a movement command, the vectors of the previous block are retained without the intersection being calculated at the end point of that block.

G38XxYy ;

This function can be used for pick feed, etc.

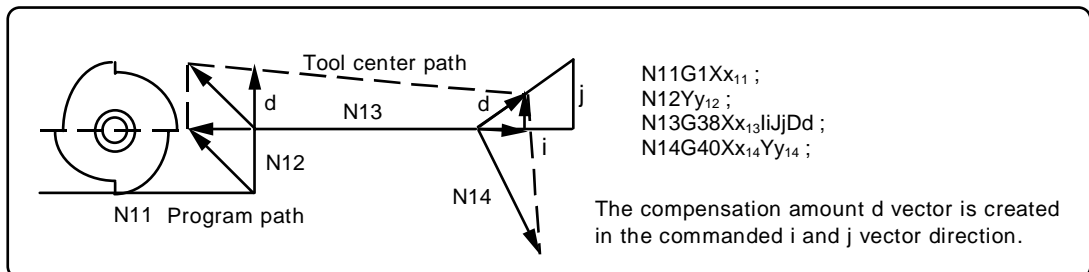


(b) Changing the vectors :

The direction of the new compensation vectors can be designated with I, J and K, while the compensation amount can be designated with D .

(These can be commanded in the same block as the movement command.)

G38 Ii Jj Dd ; (I, J and K depend on the selected plane)



(Note) If G38 is commanded in the same block as the arc block (G02/G03) I and J commands, I and J will be handled as the G38 vector, and "PR0070 Arc end point deviation large" will occur.

11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(c) Commanding G41/G42 and vector hold/change (G38) in the same block

When these are commanded in the same block, G41/G42 will be valid and G38 will be ignored as shown below.

Program command	Operation
G40→G41/G42+G38	G41/G42 commands are valid.
G40→G41/G42+G38+I, J	G41/G42I, J commands are valid.
G41/G42→G41/G42+G38	G41/G42 commands are valid.
G41/G42→G41/G42+G38+I, J	G41/G42I, J commands are valid.

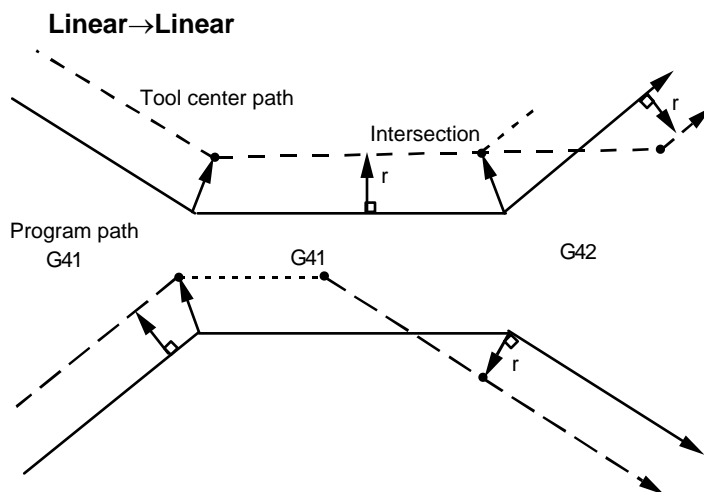
11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(3) Changing the compensation direction during tool radius compensation

- (a) The compensation direction is determined by the tool radius compensation commands (G41, G42) and the compensation amount sign.

Compensation direction G code	+	-
G41	Left-hand compensation	Right-hand compensation
G42	Right-hand compensation	Left-hand compensation

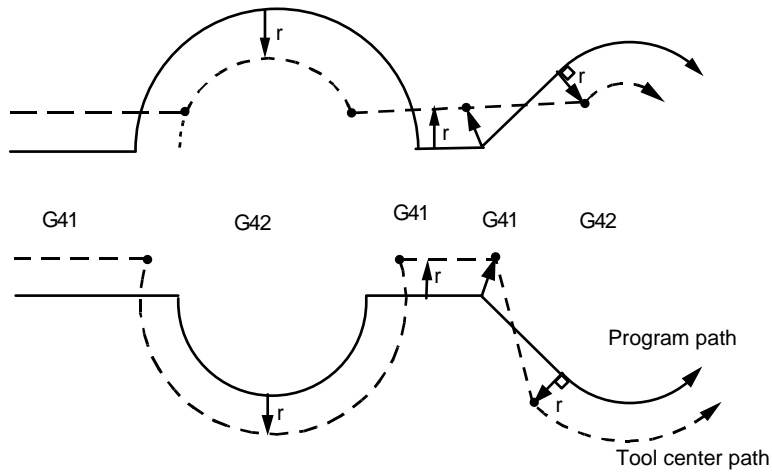
- (b) The compensation direction can be changed by changing the compensation command in the compensation mode without commanding the compensation cancel. However, no change is possible in the compensation start block and the following block. Refer to the "General cautions for tool radius compensation" for the operations when the sign is changed.



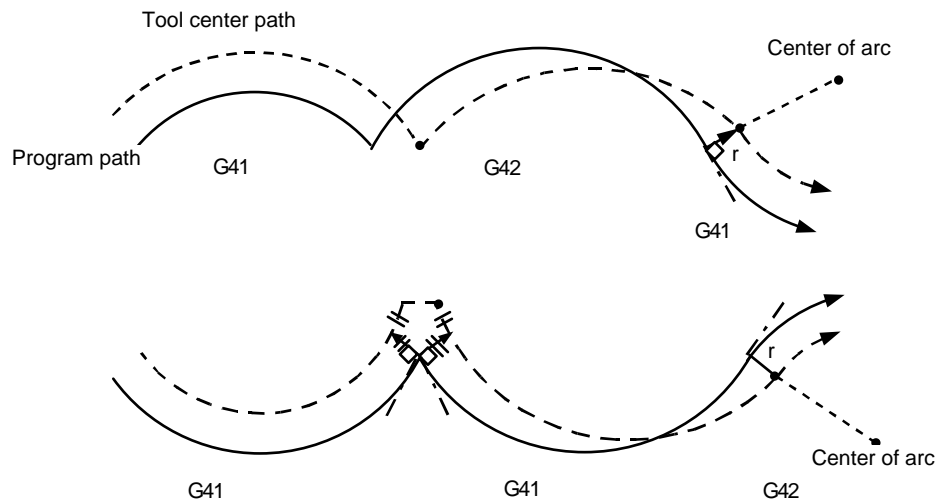
11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(c) When there is no intersection at compensation direction change

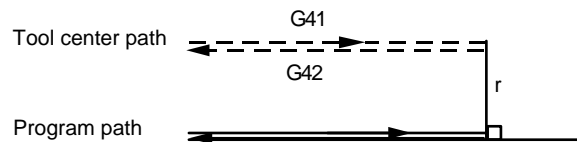
Linear→Arc



Arc→Arc

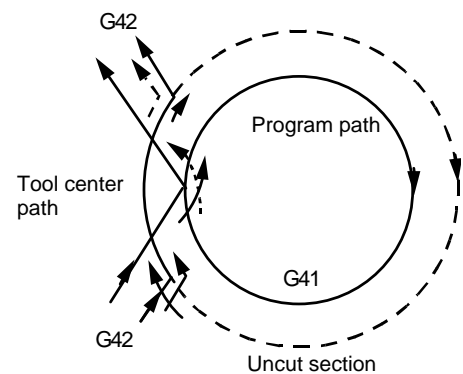


When returning with linear



(d) In the following case, the arc may become 360° or more.

- Compensation direction selection based on G41/ G42.
- I, J, K assigned in G40
 The tool center passes through an area where the arc is duplicated due to compensation. An uncut section will therefore result.



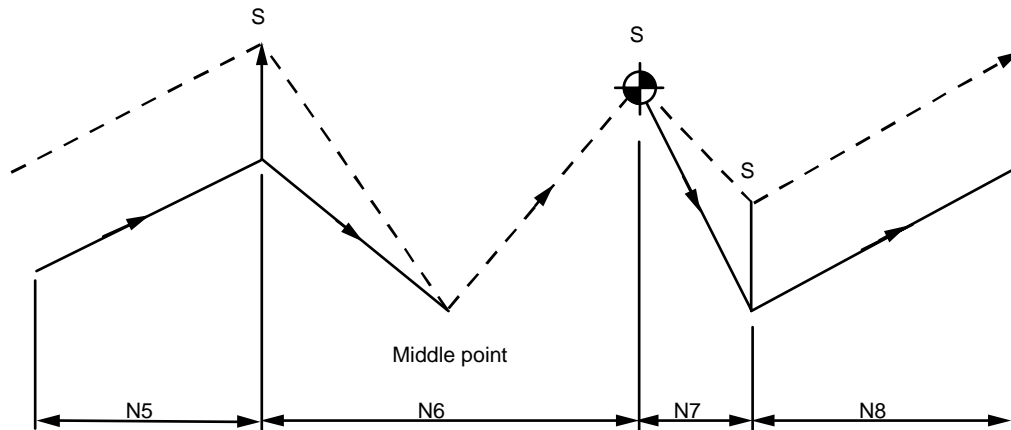
11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(4) Command for eliminating compensation vectors temporarily

When the following command is issued in the tool radius compensation mode, the compensation vectors are temporarily eliminated and a return is then made automatically to the compensation path. In this case, the compensation is not canceled, and the tool goes directly from the intersection vector to the point without vectors or, in other words, to the program commanded. When a return is made to the compensation path, it goes directly to the intersection.

(a) Reference point return command

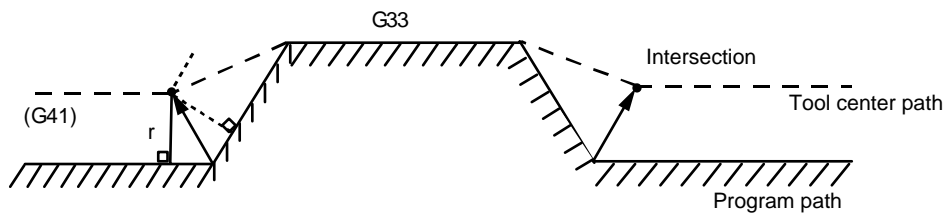


```
(G41)      ;
N5 G91 G01  X60.  Y30.  ;
N6 G28      X50.  Y-40. ;
N7          X30.  Y-60. ;
N8          X70.  Y40.  ;
           ;
```

← Compensation at middle point
(temporary vector : 0)
(Reference point when there is no middle point)

(b) G33 thread cutting command

Tool radius compensation does not apply to the G33 block.



(c) When G53 (basic machine coordinate system selection) is commanded, the compensation vector is eliminated temporarily.

(Note) The compensation vector does not change with the coordinate system setting (G92) command.

11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(5) Blocks without movement and M commands prohibiting pre-read

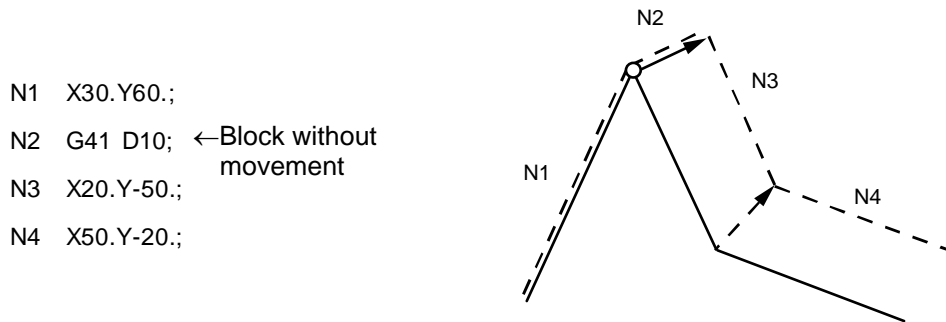
The following blocks are known as blocks without movement.

- a. M03 ; M command
- b. S12 ; S command
- c. T45 ; T command
- d. G04X500 ; Dwell
- e. G22X100. Y150. Z100. ; Machining prohibit area setting
- f. G10L10P01R50 ; Compensation amount setting
- g. G92X600. Y400. Z500. ; Coordinate system setting
- h. (G17) Z40. ; Movement but not on compensation plane
- i. G90 ; G code only
- j. G91X0 ; Zero movement amount

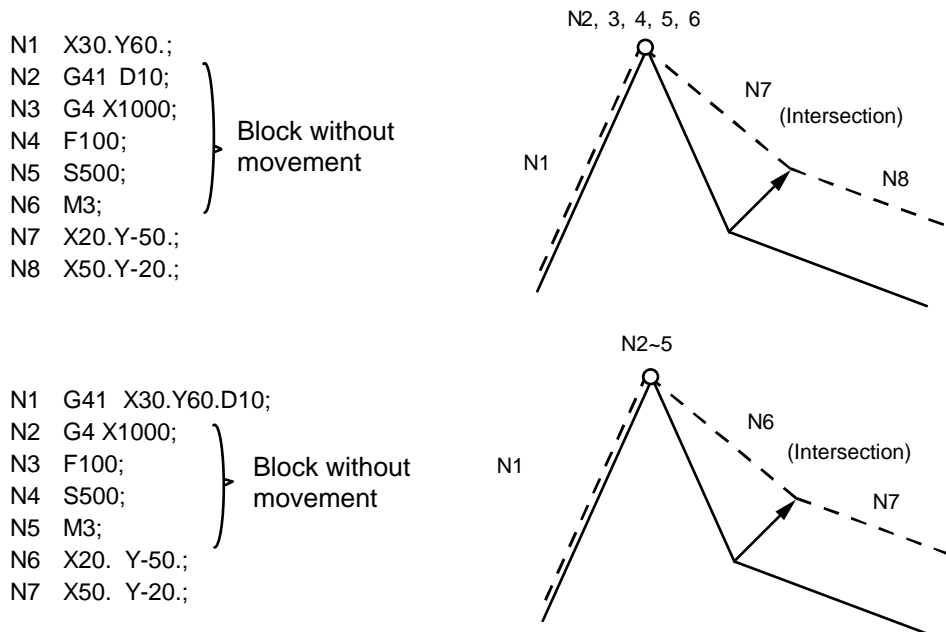
M00, M01, M02 and M30 are treated as pre-read prohibit M codes.

(a) When assigned at compensation start

Compensation is performed perpendicularly at the next movement block.



When 4 or more blocks without movement follow in succession and there is an M command prohibiting pre-read, the compensation vectors will not be created.



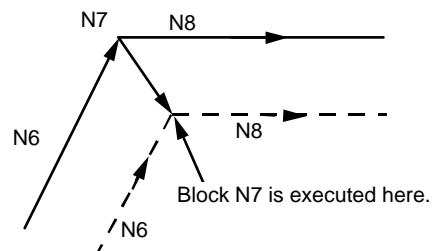
11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(b) When assigned in the compensation mode

When 4 or more blocks without movement do not follow in succession in the compensation mode and there is no M command prohibiting pre-read, the intersection vectors will be created as usual.

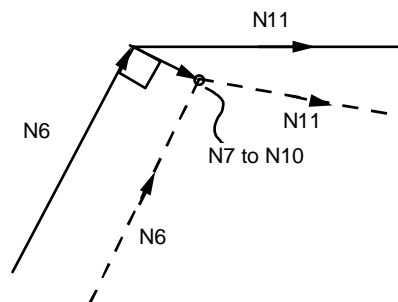
```
N6 G91X100. Y200.;
N7 G04X P1000; ----- Block without
N8 X200. ;                movement
```



When 4 or more blocks without movement follow in succession and when there is an M command prohibiting pre-read in the compensation mode, the compensation vectors are created perpendicularly at the end point of the previous block.

```
N6 X100. Y200.;
N7 G4 X1000;
N8 F100 ;
N9 S500 ;
N10 M4 ;
N11 X100. ;
```

} Block without movement

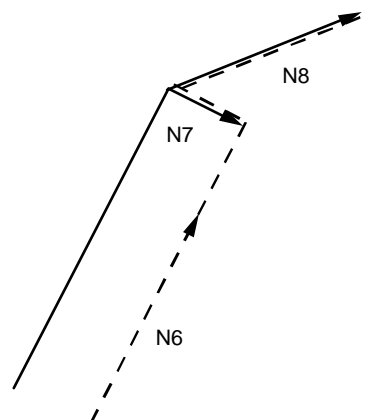


In this case, cutting results.

(c) When commanded together with compensation cancel

Only the compensation vectors are canceled when a block without movement is commanded together with the G40 command.

```
N6 X100. Y200. ;
N7 G40 G04P1000 ;
N8 X100. Y50. ;
```

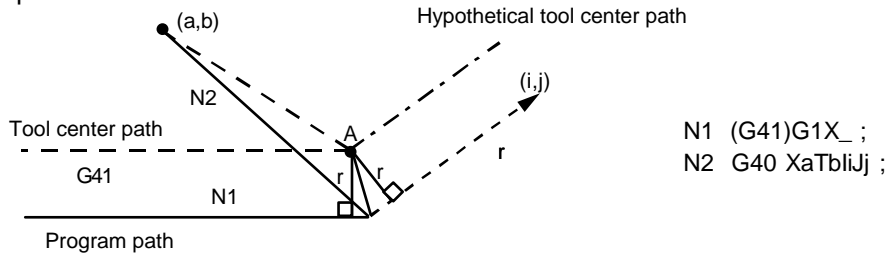


11. TOOL COMPENSATION

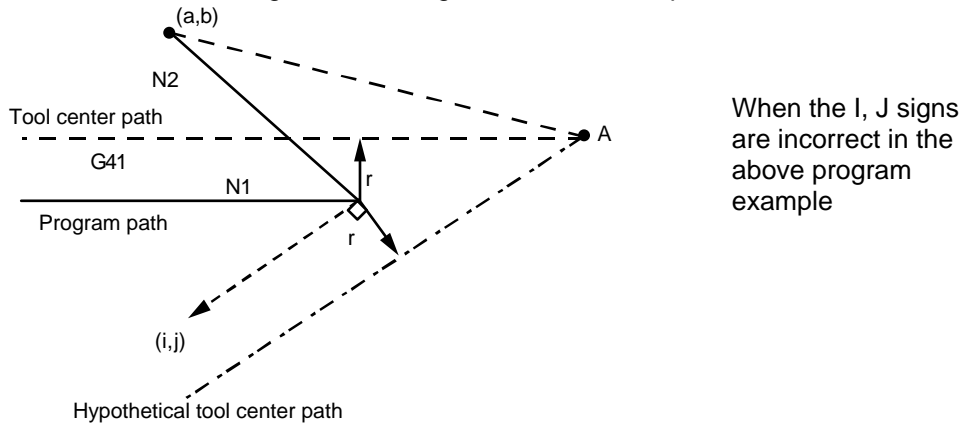
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(6) When I, J, K are commanded in G40

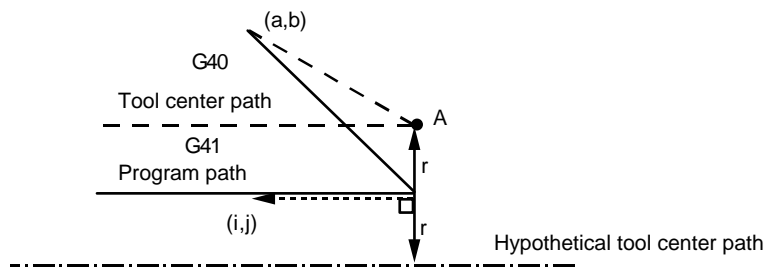
- (a) When the last movement command block among the 4 blocks before the G40 block is in the G41 or G42 mode, it is considered to be commanded in the vector I, J and K direction from the end point of the last movement command, and compensation is canceled after interpolation up to the intersection with the corresponding hypothetical tool center path. There is no change in the compensation direction.



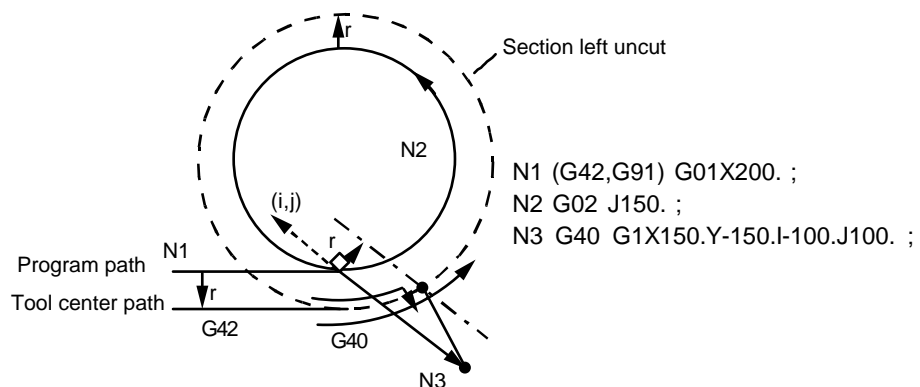
In this case, care is required since the intersection must be sought even when the command vector is incorrect, as in the figure below, regardless of the compensation direction.



When the compensation vectors are extremely large as a result of the intersection calculation, perpendicular vectors are created in the block before G40.



- (b) Care should be taken with the G40 command after an arc command since a section may be left uncut when the arc exceeds 360° due to the contents of I, J and K.



11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

11.2.5 G41/G42 commands and I, J, K designation

The compensation direction can be intentionally changed by issuing the G41/G42 command and I, J, K in the same block.

(1) Command format

G17 (XY plane) G41/G42 X__ Y__ I__ J__ ;
G18 (ZX plane) G41/G42 X__ Z__ I__ K__ ;
G19 (YZ plane) G41/G42 Y__ Z__ J__ K__ ;

Assign a linear command (G00, G01) in a movement mode.

(2) I, J type vectors (G17 XY plane selection)

The new I, J type vector (G17 plane) created by this command is now described. Similar descriptions apply to vector K, I for the G18 plane and to J, K for the G19 plane.

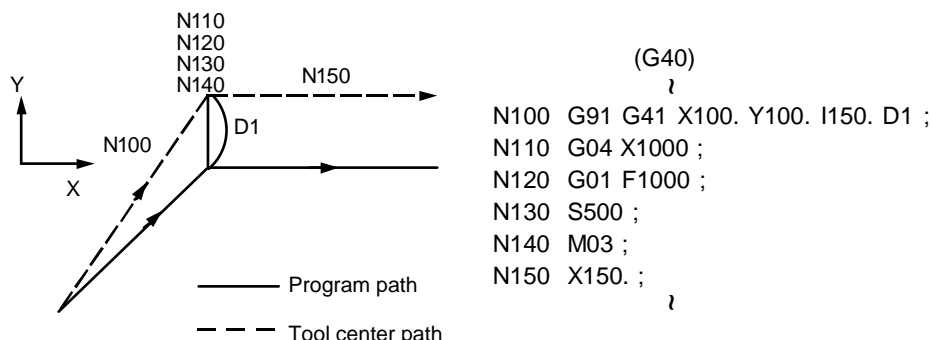
As shown in the figures below, the vectors with a size equivalent to the compensation amount are made to serve as the I, J type compensation vector perpendicularly to the direction designated by I, J without the intersection of the programmed path being calculated.

The I, J vector can be commanded during the compensation mode (when G41/42 are commanded in the same block) even at the compensation start.

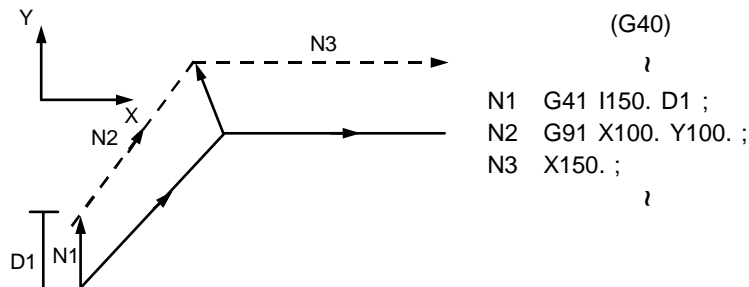
Even when the compensation mode is entered, if there is no G41/G42 command, the I, J vector command will be ignored.

(a) When I, J is commanded at compensation start

1) When there are movement commands



2) When there are no movement commands

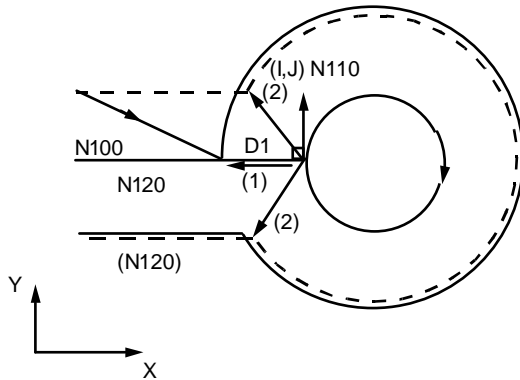


11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(b) When I, J has been assigned in G41/G42 mode

1) When there are movement commands



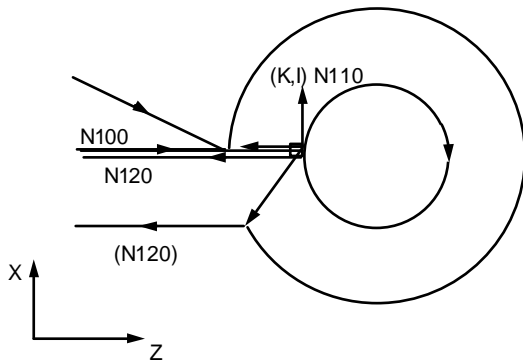
```

}
(G17 G41 G91)
N100 G41 G00X150. J50. ;
N110 G02 I150. ;
N120 G00 X-150. ;
}

```

————— Program path
 - - - - - Tool center path
 - - - - - Path with intersection calculation

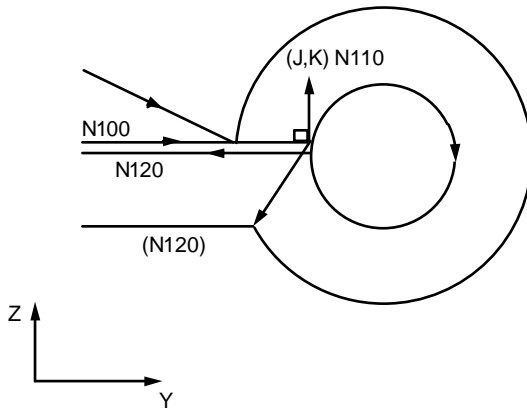
(1) I,J type vector
 (2) Intersection calculation type vector



```

}
(G18 G41 G91)
N100 G41 G00 Z150. I50. ;
N110 G02 K50. ;
N120 G00 Z-150. ;
}

```

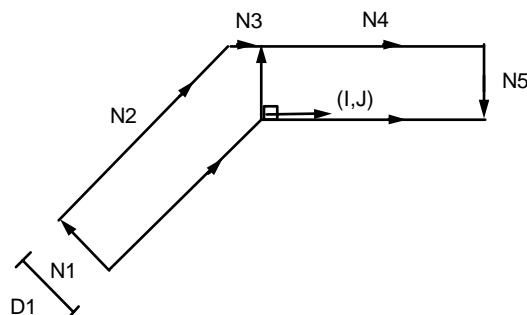


```

}
(G19 G41 G91)
N100 G41 G00 Y150. K50. ;
N110 G02 J50. ;
N120 G00 Y-150. ;
}

```

2) When there are no movement commands



```

N1 G41 D1 G01 F1000 ;
N2 G91 X100. Y100. ;
N3 G41 I50. ;
N4 X150. ;
N5 G40 ;

```

11. TOOL COMPENSATION

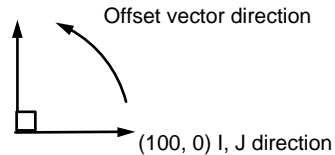
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(3) Direction of offset vectors

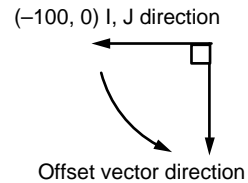
(a) In G41 mode

Direction produced by rotating the direction commanded by I, J through 90° to the left from the forward direction of the Z axis (3rd axis) as seen to the zero point

(Example 1) With I100.



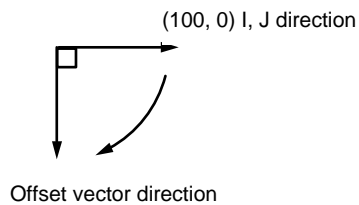
(Example 2) With I -100.



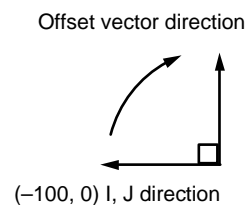
(b) In G42 mode

Direction produced by rotating the direction commanded by I, J through 90° to the right from the forward direction of the Z axis (3rd axis) as seen to the zero point

(Example 1) With I100.

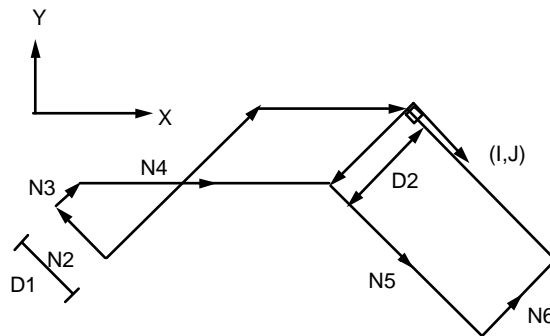


(Example 2) With I -100.



(4) Selection of offset modal

The G41 or G42 modal can be selected at any time.



```

N1 G28 X0Y0 ;
N2 G41 D1 F1000 ;
N3 G01 G91 X100. Y100. ;
N4 G42 X100. I100. J-100. D2;
N5 X100. Y-100. ;
N6 G40 ;
N7 M02 ;
%
```

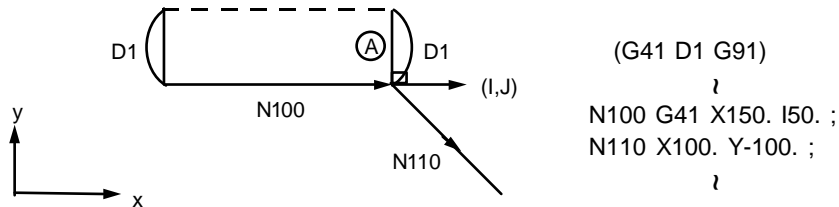
11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(5) Compensation amount for offset vectors

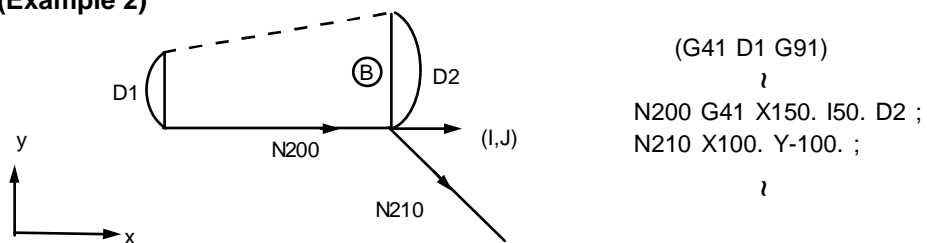
The compensation amounts are determined by the compensation number (modal) in the block with the I, J designation.

(Example 1)



Vector (A) is the compensation amount entered in compensation number modal D1 in the N100 block.

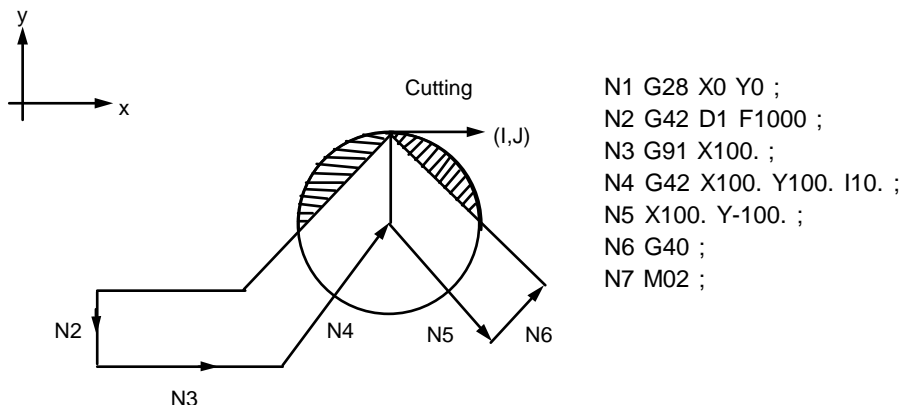
(Example 2)



Vector (B) is the compensation amount entered in compensation number modal D2 in the N200 block.

(6) Notes

- (a) Issue the I, J type vector in a linear mode (G00, G01). If it is issued in an arc mode at the start of compensation, program error will result.
 An I, J designation in an arc mode functions as an arc center designation in the circular mode.
- (b) When the I, J type vector has been designated, it is not deleted (avoidance of interference) even if there is interference. Consequently, overcutting may arise in such a case.



11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

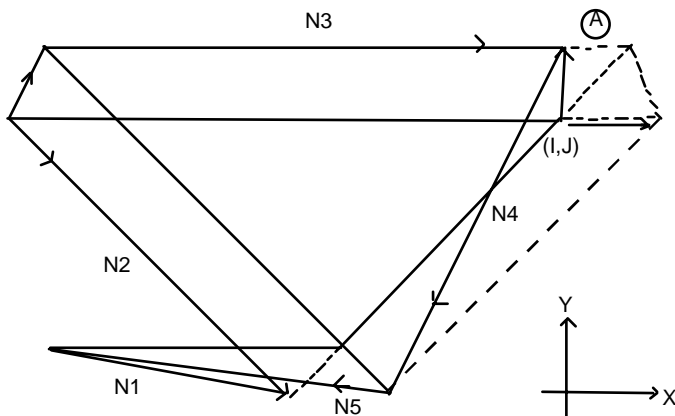
(7) Supplementary notes

(a) The vectors differ for the G38 I_J_ (K_) command and G41/G42 I_J_ (K_) command.

G38	G41/G42
: (G41) : G38 G91 X100. I50. J50. ; :	: (G41) : G41 G91 X100. I50. J50. ; :
Vector with size of compensation amount in I, J direction	Vector with size of compensation amount perpendicularly to the I, J direction

(b) Refer to the following table for the compensation direction which can be applied depending on whether or not the G41/G42 and I, J (K) commands are used in combination.

G41/G42	I, J, (K)	Compensation method
No	No	Intersection calculation type vector
No	Yes	Intersection calculation type vector
Yes	No	Intersection calculation type vector
Yes	Yes	I, J type vector (No insertion block)



```

N1 G91 G01 G41 X200
  D1 F1000 ;
N2 X-150. Y150. ;
N3 G41 X300. I50. ;
N4 X-150. Y-150. ;
N5 G40 X-200. ;
  
```

The (A) insertion block is not used with I, J type vector compensation

11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

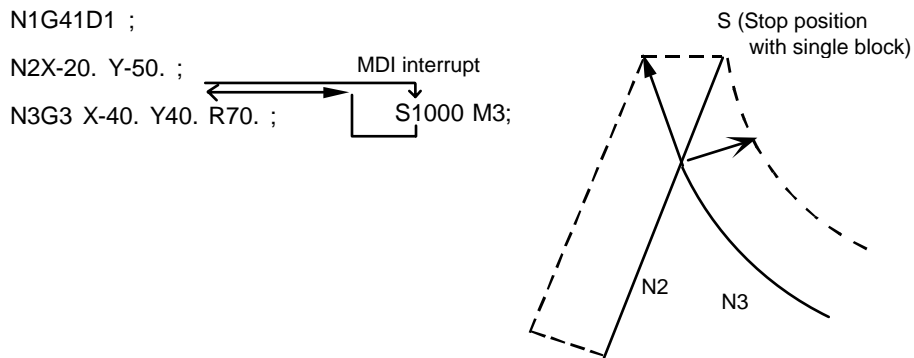
11.2.6 Interrupts during tool radius compensation

(1) MDI interrupt

Tool radius compensation is valid in any automatic operation mode - whether tape, memory or MDI operation.

An interrupt based on MDI will give the result as in the figure below after block stop during tape or memory operation.

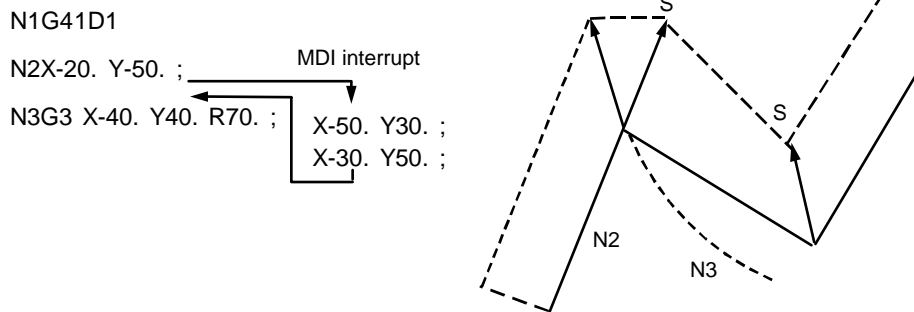
(a) Interrupt without movement (tool path does not change)



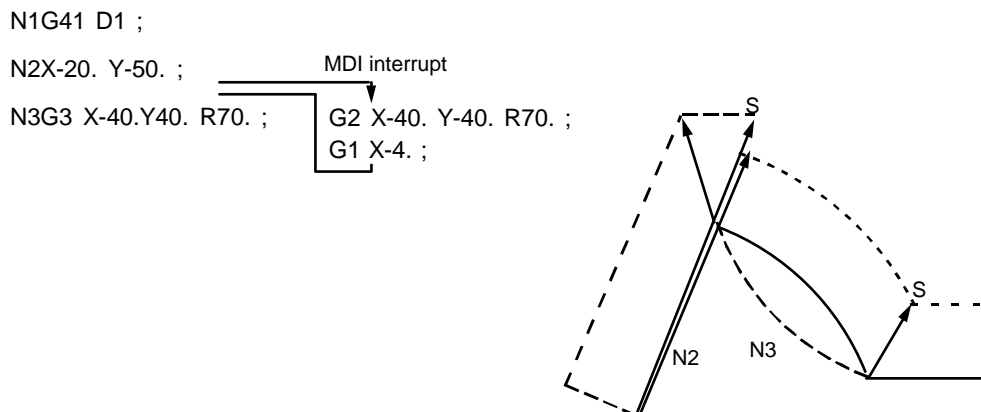
(b) Interrupt with movement

The compensation vectors are automatically re-calculated at the movement block after interrupt.

With linear interrupt



With arc interrupt

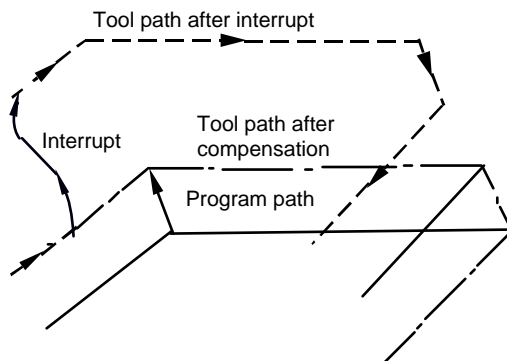


11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(2) Manual interrupt

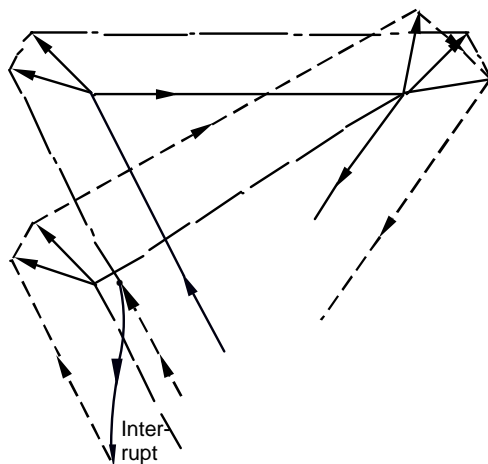
(a) Interrupt with manual absolute OFF.

The tool path is shifted by an amount equivalent to the interrupt amount.

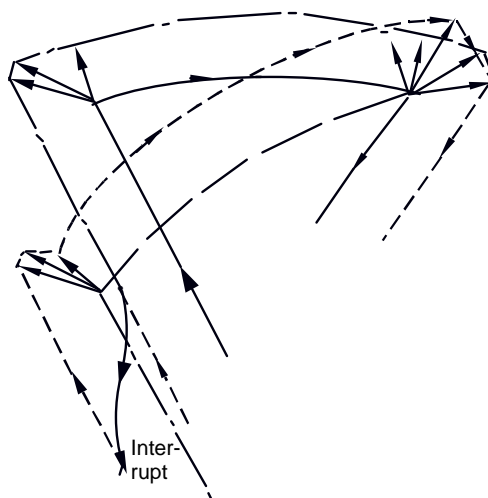


(b) Interrupt with manual absolute ON.

In the incremental value mode, the same operation results as with manual absolute OFF.



In the absolute value mode, however, the tool returns to its original path at the end point of the block following the interrupted block, as shown in the figure.



11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

11.2.7 General cautions for tool radius compensation

(1) Assigning the compensation amounts

The compensation amount is assigned by designating the compensation amount number using a D code. Once a D code has been designated, it will remain valid until a separate D code is subsequently commanded. Program error occurs when designation is made by an H code. Besides being used to designate the compensation amounts for tool radius compensation, the D codes are also used to assign the compensation amounts for tool position offset.

(2) Changing the compensation amounts

(a) Compensation amounts are normally changed when a different tool has been selected in the radius compensation cancel mode. However, when an amount is changed in a compensation mode, the vectors at the end point of the block are calculated using the compensation amount designated in that block. Refer to "11.2.8 Changing the compensation number in the compensation mode" for details.

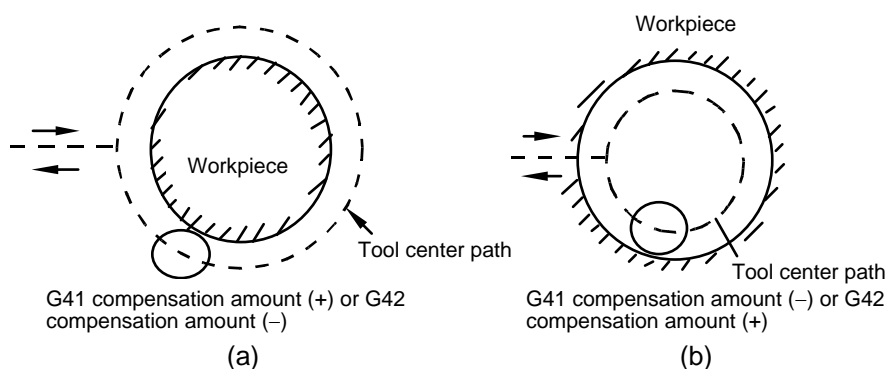
(b) To set the tool compensation amount (tool length and tool radius) during automatic operation, set during the single block stop.

The following will occur when the compensation amount is set during automatic operation.

State during automatic operation	When tool length offset amount is set	When tool radius compensation amount is set
During single stop	Whether to validate the setting from the next block or from the next H No. command can be set with parameter (110051 Op Tolcomp ofs valid).	Valid from the next block.
Other than the above	The set compensation amount is validated from the tool compensation command block several blocks ahead of the block being executed.	

(3) Compensation amount sign and tool center path

When the compensation amount is given a negative (-) sign, the resulting shape is the same as when the G41 and G42 commands have all been replaced. Thus, the tool that was rotating around the outer periphery of the workpiece rotates around the inner periphery, and that rotating around the inner periphery rotates around the outer periphery.



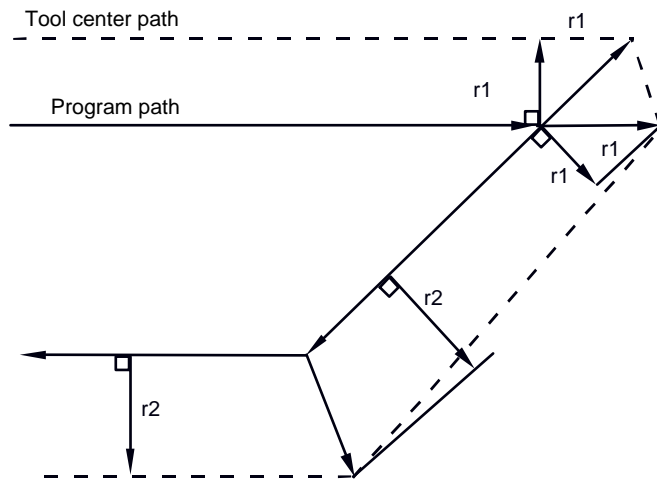
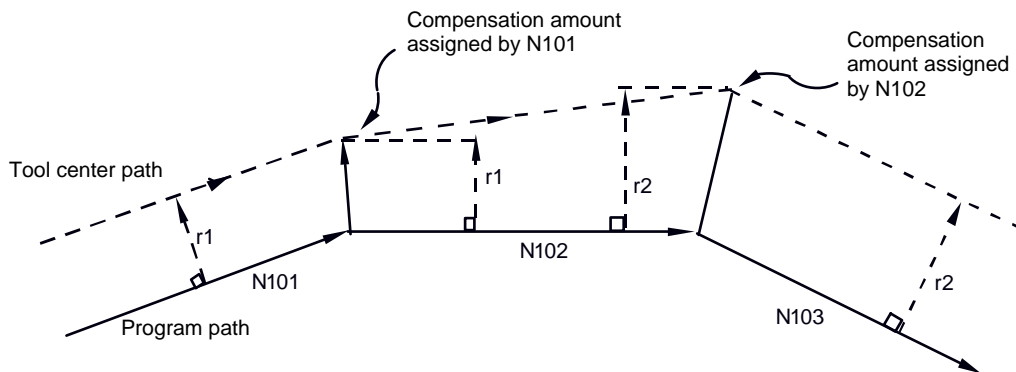
11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

11.2.8 Changing the compensation number in the compensation mode

In principle, the compensation numbers should not be changed in a compensation mode. If they are changed, operation will be as follows.

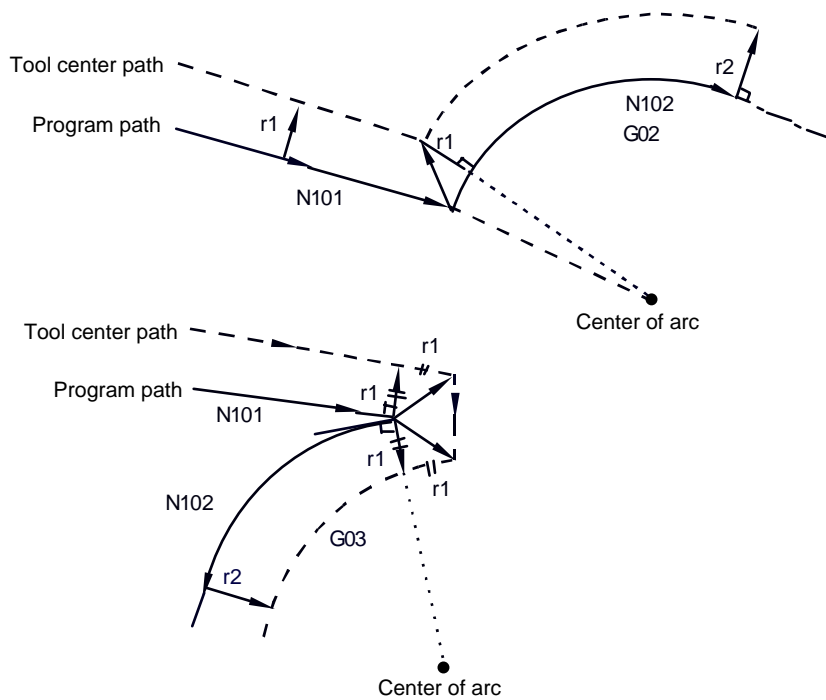
(a) Linear→Linear



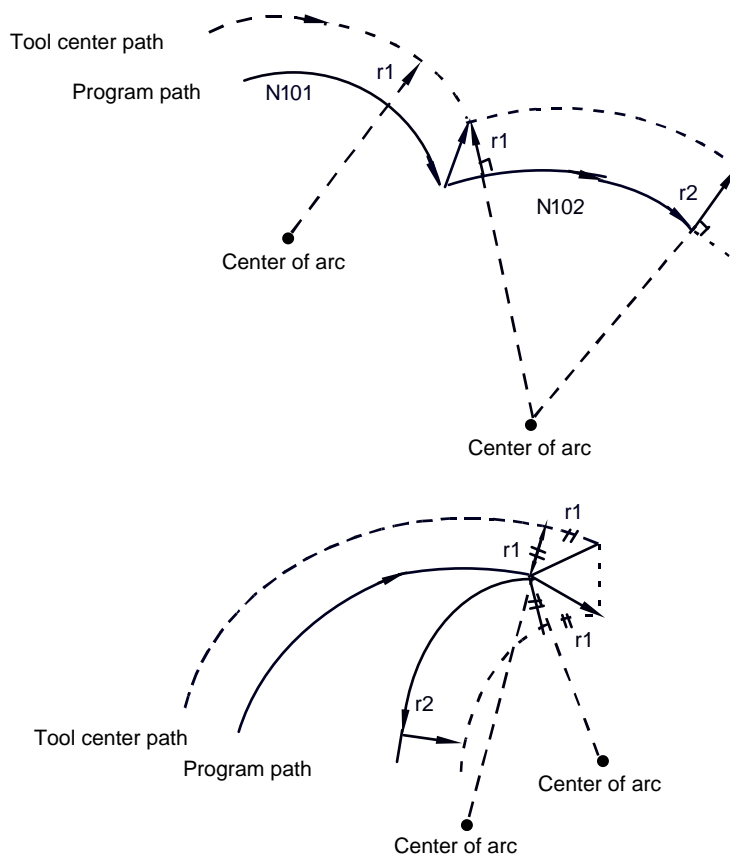
11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(b) Linear→Arc



(c) Arc→Arc



11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

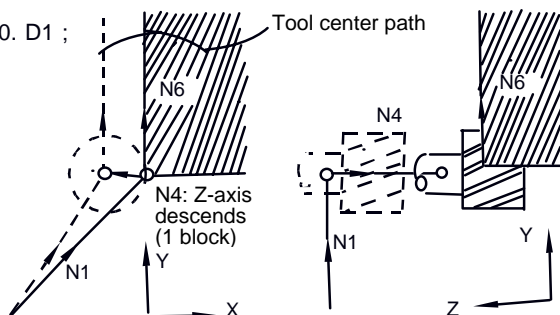
11.2.9 Tool radius compensation start and Z-axis cutting

The usual method of Z-axis cutting consists in first applying radius compensation (normally on the XY plane) beforehand at a position distanced from the workpiece when cutting starts. Bear in mind the following points for programming if the Z-axis operation is to be divided into two stages: rapid traverse and then cutting feed after the work is approached.

When the following has been programmed.

```

N1 G91 G00 G41 X 500. Y 500. D1 ;
N2 S1000 ;
N3 M3 ;
N4 G01 Z-300. F1 ;
N6   Y 100. F2 ;
    .
    .
    .
    .
    
```

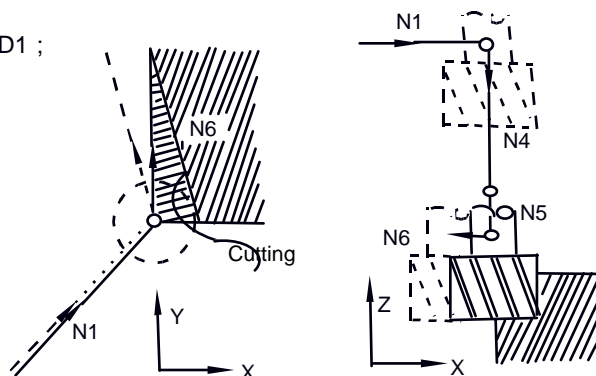


If programmed in this way, it is possible to read up to the N6 block at N1 compensation start, determine the relationship between N1 and N6 and proceed with compensation properly as shown in the above figure.

Next, when the N4 block in the above program has been divided into two:

```

N1 G91 G00 G41 X 500. Y 500. D1 ;
N2 S1000 ;
N3 M3 ;
N4   Z-250. ;
N5 G01 Z-50. F1 ;
N6   Y 100. F2 ;
    
```



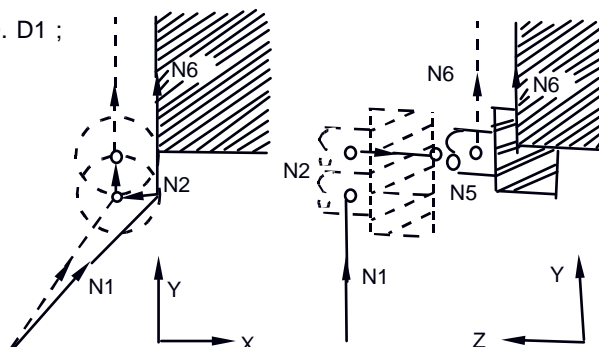
It is not possible here to read as far as the N6 block at N1 compensation start since there are four blocks (N2 to N5) without a command on the XY plane.

Compensation is therefore based on the information in N1 block only, and the NC system cannot create compensation vectors at compensation start. Over-cutting therefore results, as shown in the above figure.

In cases like this, by taking the calculations performed inside the NC system into prior consideration, it is possible to safeguard against over-cutting provided that, after the Z axis has descended, the command in the same direction at the advance direction is already assigned immediately before the Z axis cutting.

```

N1 G91 G00 G41 X 500. Y 400. D1 ;
N2   Y100. S1000 ;
N3 M3 ;
N4   Z-250. ;
N5 G01 Z-50. F1 ;
N6   Y 100. F2 ;
    
```



The same direction as the N6 advance direction is assigned in N2 so that compensation is performed properly.

11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

11.2.10 Interference check

(1) Outline

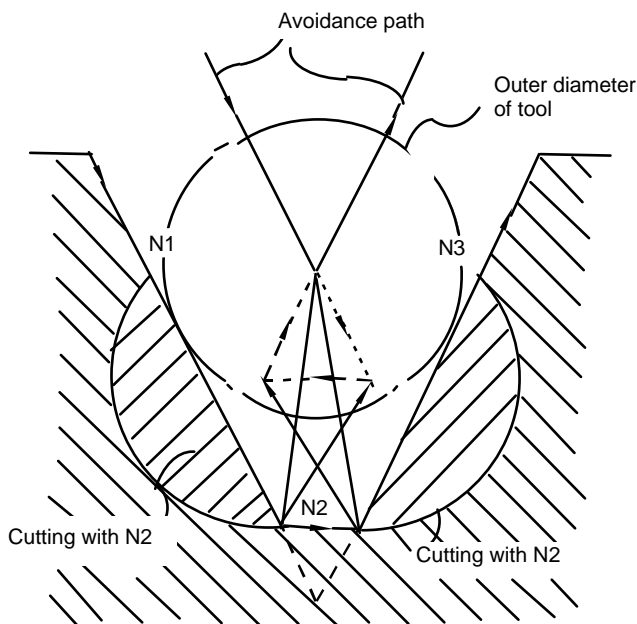
A tool, whose radius has been compensated under the tool radius compensation function by the usual 2-block pre-read, may sometimes cut into the workpiece. This is known as interference, and interference check is the function which prevents this from occurring.

There are three types of interference check, as indicated below, and each can be selected for use by parameter.

Function	Parameter	Operation
Interference check alarm function	Rad compen intrf byp OFF Tcomp interf chk OFF	Program error results before block causing cut is executed, and operation stops.
Interference check avoidance function	Rad compen intrf byp ON Tcomp interf chk OFF	Tool path is changed so that workpiece is not cut into.
Interference check invalid function	Tcomp interf chk ON	Cutting proceeds unchanged even when it occurs. Use this for microscopic segment programs.

(Example)

```
(G41)
N1 G90 G1 X50. Y-100. ;
N2 X70. Y-100. ;
N3 X120. Y0 ;
```



(a) With alarm function

The alarm occurs before N1 is executed and so, using the buffer correction function, N1 block can be changed as below and machining can be continued:

```
N1 G90 G1 X20. Y-40. ;
```

(b) With avoidance function

The intersection of N1 and N3 is calculated and the interference avoidance vectors are created.

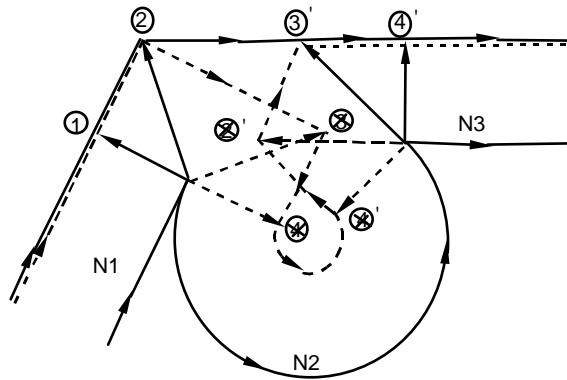
(c) With interference check invalid function

The tool passes while cutting the N1 and N3 linear.

11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(Example of interference check)



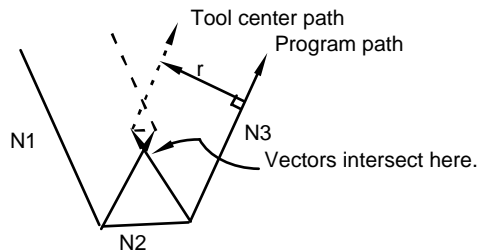
Vector ① ④' check → No interference
 ↓
 Vector ② ③' check → No interference
 ↓
 Vector ③ ②' check → Interference → Vectors ③ ②' deleted
 ↓
 Vectors ④ ①' deleted

As a result of the above processing, vectors ①, ②, ③' and ④' remain as the valid vectors and operation is executed with the path connecting vectors ①, ②, ③' and ④' as the interference avoidance path.

(2) Detailed description

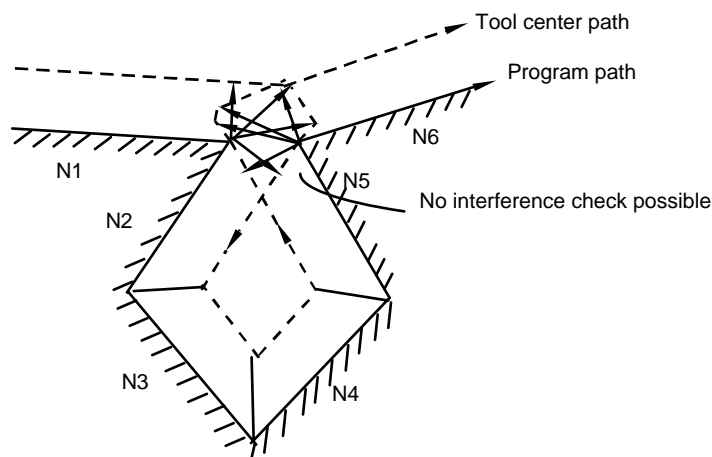
(a) Conditions regarded as interference

Interference is regarded as occurring when the compensation calculated vectors, which have been created at the point of contact with movement commands, intersect in a case where there are movement commands in 3 blocks among the 5 pre-read blocks.



(b) When interference check is not possible

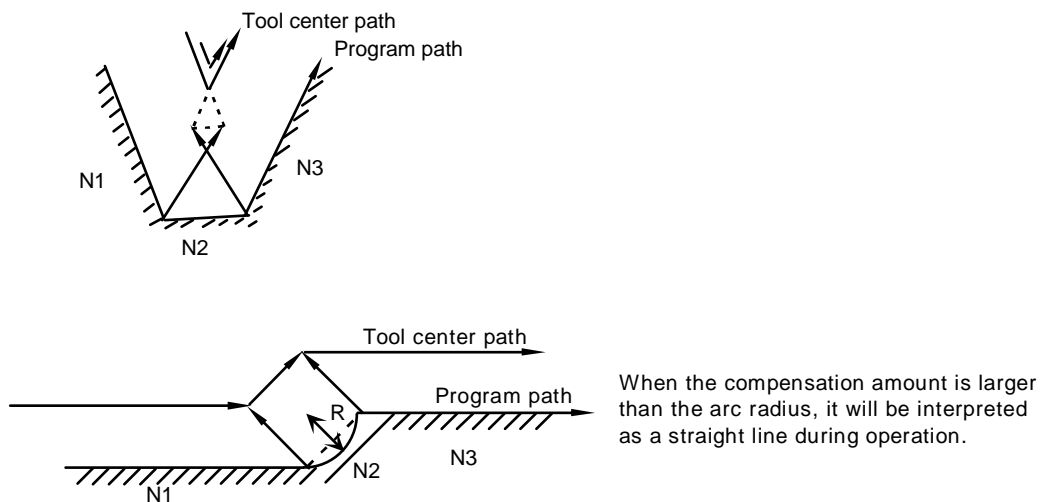
- 1) When it is not possible to pre-read movement commands for 3 blocks
 (When there are 3 or more blocks without movement among the 5 pre-read blocks)
- 2) When interference occurs at the fourth or subsequent movement command block



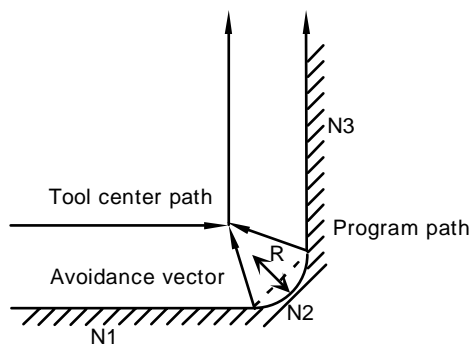
11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(c) Operation during interference avoidance

The following operation is undertaken when the interference avoidance function is available.



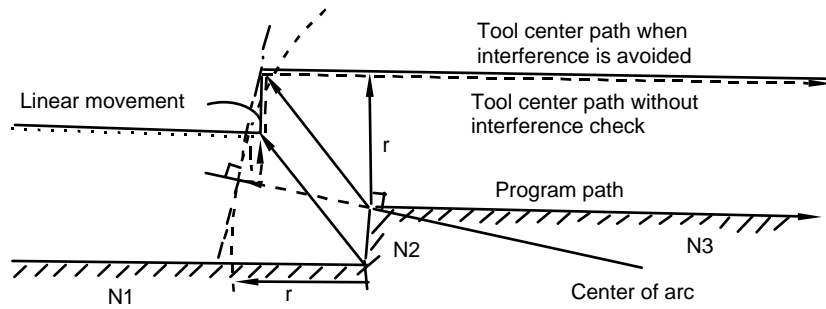
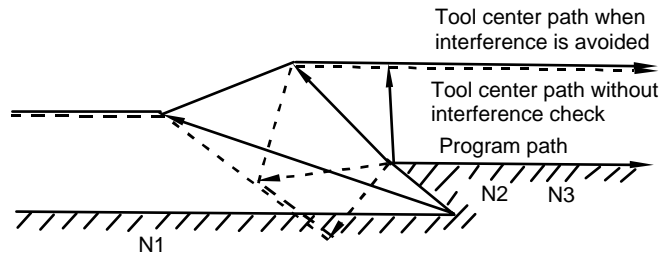
Note that if the arc block is interpreted as a straight line, when the compensation direction with the previous and next blocks is on the inner side, it will not be interpreted as a straight line, and interference avoidance will be carried out.



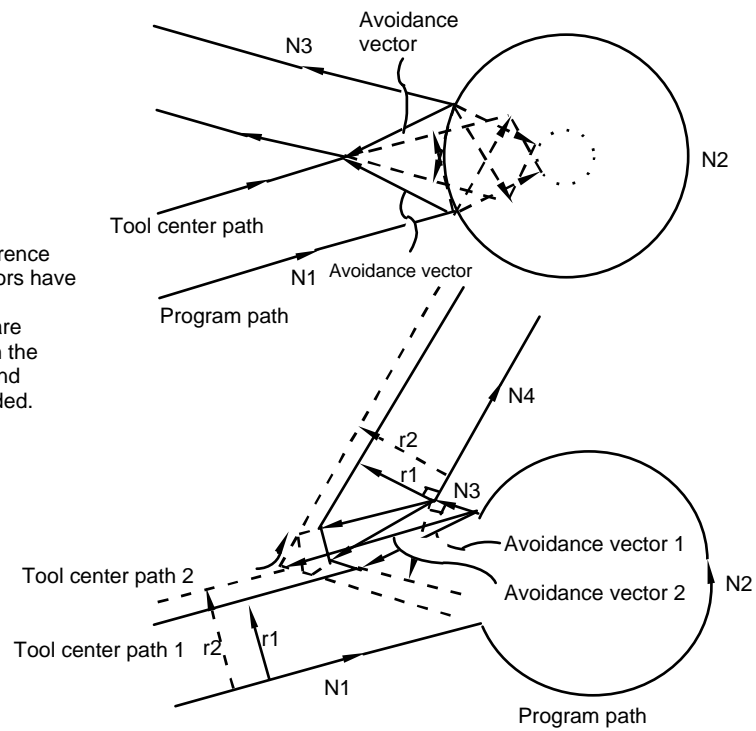
11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

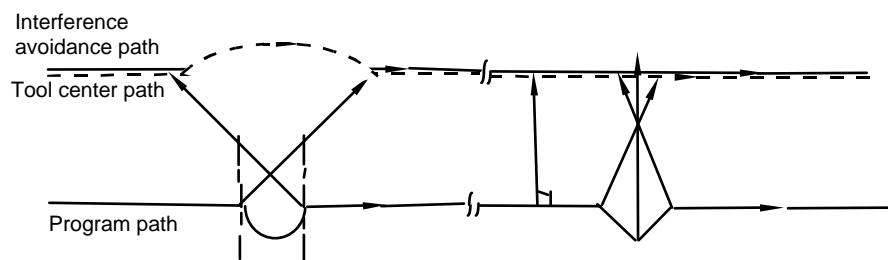
Solid line vector : Valid
Broken line vector : Invalid



When all the interference avoidance line vectors have been deleted, new avoidance vectors are created as shown in the figure on the right and interference is avoided.



In the figure below, the groove is left uncut.



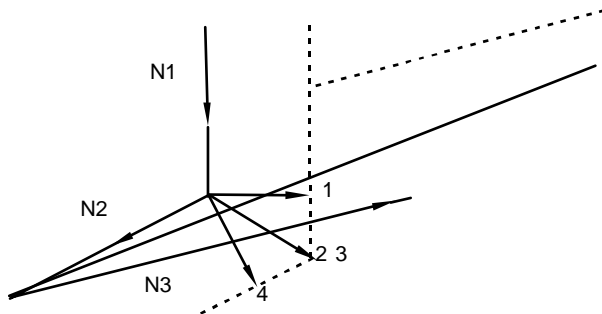
11. TOOL COMPENSATION
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

(3) Interference check alarm

The interference check alarm occurs under the following conditions.

(a) When the interference check alarm function has been selected

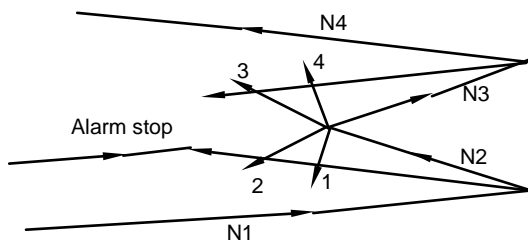
When all the vectors at the end point of its own block have been deleted
 When, as shown in the figure, vectors 1 to 4 at the end point of the N1 block have all been deleted, program error results prior to N1 execution.



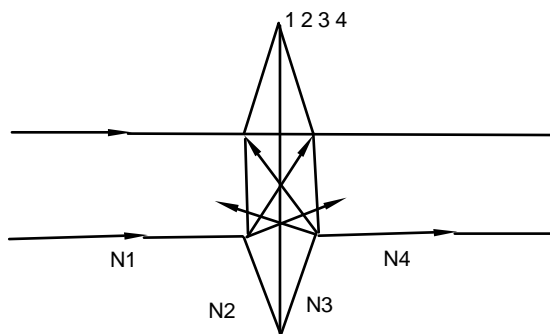
(b) When the interference check avoidance function has been selected

1) When there are valid vectors at the end point of the following block even when all the vectors at the end point of its own block have been deleted

a. When, as shown in the figure, the N2 in interference check is conducted, the N2 end point vectors are all deleted but the N3 end point vectors are regarded as valid. Program error now occurs at the N1 end point.



b. In a case such as that shown in the figure, the tool will move in the reverse direction at N2. Program error results prior to N1 execution.



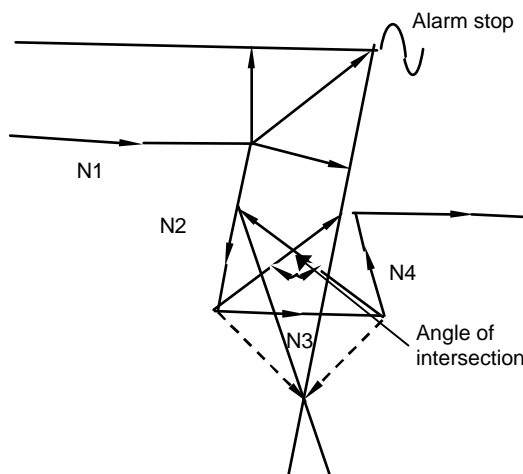
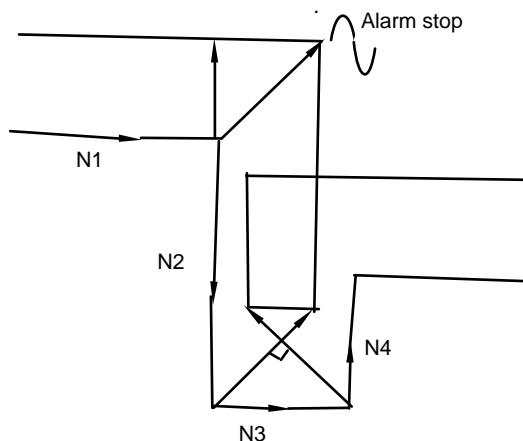
11. TOOL COMPENSATION

11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

2) When the avoidance vectors cannot be created

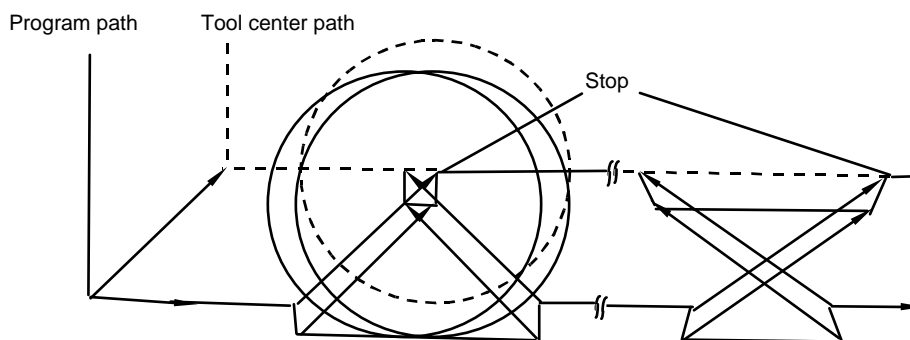
Even when, as shown in the figure, the conditions for creating the avoidance vectors are met, it may still be impossible to create these vectors or the interference vectors may interfere with N3.

As a result, program error will occur at the N1 end point when the vector intersecting angle is more than 90°.



3) When the program advance direction and the advance direction after compensation are reversed

When a groove that is parallel or has a widening bottom that is narrower than the tool diameter is programmed, even if interference does not actually occur, it will be interpreted as occurring.



11. TOOL COMPENSATION

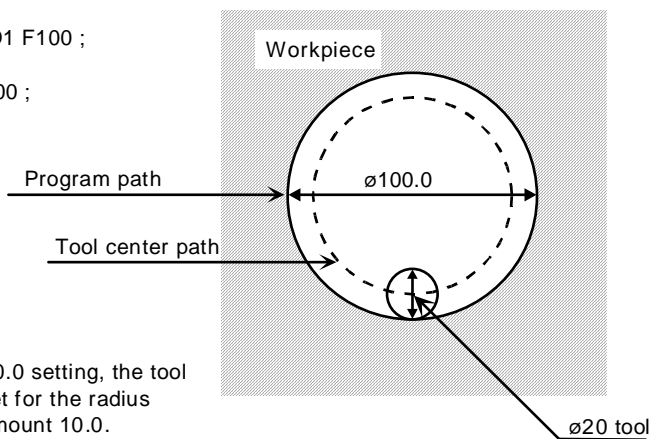
11.2 Tool Radius Compensation; G40/G41/G42/G38/G39

11.2.11 Compensation amount diameter designation

By setting the control parameter "#320039 OFS Diameter DESIGN" ON, the compensation amount set in the command tool number can be recognized as the diameter compensation amount, and converted into a radius compensation amount during compensation.

(Example) For $\phi 100.0$ circle machining

```
:  
G01 G41 X50. D1 F100 ;  
G03 I-50. ;  
G01 G40 X0 F500 ;  
:
```

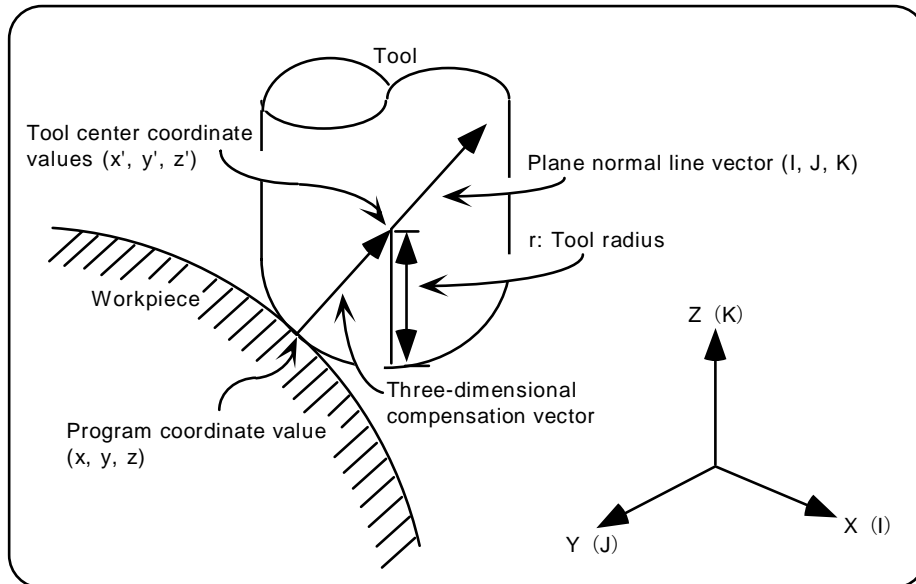


With the D1 = 20.0 setting, the tool center path is set for the radius compensation amount 10.0.

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

(1) Outline

The three-dimensional tool radius compensation offsets the tool in a three-dimensional space following the commanded three-dimensional vectors.



As shown above, the tool is moved with the tool center coordinate values (x', y', z') offset by the tool radius r in respect to the program coordinate values (x, y, z) following the plane normal line vector (I, J, K).

While two-dimensional tool radius compensation creates the vectors at a right angle in the (I, J, K) direction, three-dimensional tool radius compensation creates the vector in the (I, J, K) direction. (The vector is created at the end point of the block.)

The three-dimensional compensation vector (offset) axis elements are as follows.

$$Hx = \frac{I}{\sqrt{(I^2 + J^2 + K^2)}} \times r$$

$$Hy = \frac{J}{\sqrt{(I^2 + J^2 + K^2)}} \times r$$

$$Hz = \frac{K}{\sqrt{(I^2 + J^2 + K^2)}} \times r$$

Thus, the tool center coordinate values (x', y', z') are each expressed as follows. Note that (x, y, z) are the program coordinate values.

$$\begin{aligned} x' &= x + Hx \\ y' &= y + Hy \\ z' &= z + Hz \end{aligned}$$

(Note 1) Three-dimensional compensation vector (Hx, Hy, Hz) refers to the plane normal line vector such as follows ;

- The direction is same as the plane normal line vector (I, J, K).
- The size equals to the tool radius (r).

(Note 2) When the machining parameter "330013 3-dimensional compen" is set to a value other than "0", "330013 3-dimensional compen" will be used as the $\sqrt{(I^2 + J^2 + K^2)}$ value. (Refer to the Parameter Manual for details.)

(Note 3) This function is an additional specifications. If commanded when the function is not provided, "PR0154 No 3D-compensation spec." will occur.

11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

(2) Command format

The compensation number D and plane normal line vector (I, J, K) are commanded in the same block as the three-dimensional tool radius compensation command G41 (G42).

Setting all three axes of the plane normal line vector (I, J, K) are required. If any axis is not commanded, the mode will be handled as the normal tool radius compensation mode. (When setting "0" to the axes, this command is valid.)

<u>G41 (G42)</u>	<u>Xx Yy Zz</u>	<u>li Jj Kk</u>	<u>Dd</u> ;	Three-dimensional tool radius compensation start → Refer to (Example 1) and (Example 2).
Three-dimensional tool radius compensation command *1	Movement axis command compensation space *2	Plane normal line vector	Compensation No.	
↓	<u>Xx Yy Zz</u>	<u>li Jj Kk</u> ;		In three-dimensional tool radius compensation command Each axis movement command and the new plane normal line vector are commanded. → Refer to (Example 3) to (Example 7).
<u>G40</u> ; (or <u>D00</u> ;)) <u>G40 Xx Yy Zz</u> ; (or <u>Xx Yy Zz D00</u> ;))				Three-dimensional tool radius compensation cancel The three-dimensional tool radius compensation cancel command G40 (or D00) is commanded independently or in the same block with a movement axis command. → Refer to (Example 8) and (Example 9).

Compensation G code	Compensation amount: +	Compensation amount: -	D00
G40	Cancel	Cancel	Cancel
G41	I, J, K direction	I, J, K reverse direction	Cancel
G42	I, J, K reverse direction	I, J, K direction	Cancel

*2 The compensation space is determined by the three-dimensional tool radius compensation start block's axis address command (X, Y, Z, U, V, W). Here, U, V and W are each the additional axes for the X, Y and Z axis.

If the X axis and U axis (Y and V, Z and W) are commanded simultaneously in the three-dimensional tool radius compensation start block, the currently commanded plane selection axis will have the priority.

If the axis address is not commanded, it will be interpreted that the X, Y and Z axes are commanded for the coordinate axes.

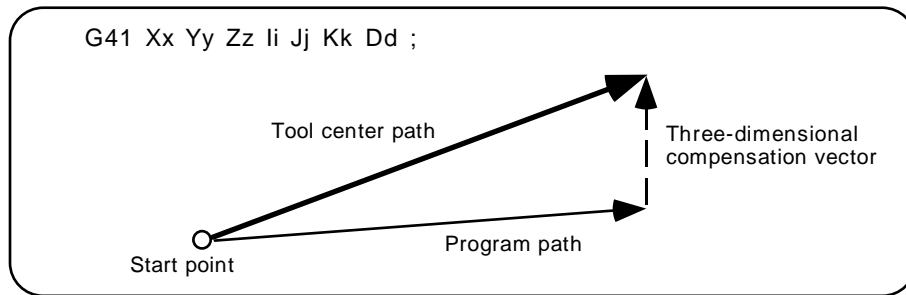
(Example) G17

G41 Xx Yy Zz li Jj Kk ;	XYZ space
G17 ; G41 Yy li Jj Kk ;	XYZ space
G17 V ; G41 Xx Vv Zz li Jj Kk ;	XVZ space
G17 W ; G41 Ww li Jj Kk ;	XYW space
G17 ; G41 Xx Yy Zz Ww li Jj Kk ;	XYZ space
G17 W ; G41 Xx Yy Zz Ww li Jj Kk ;	XYW space
G17 ; G41 li Jj Kk ;	XYZ space
G17 U ; G41 li Jj Kk ;	UYZ space

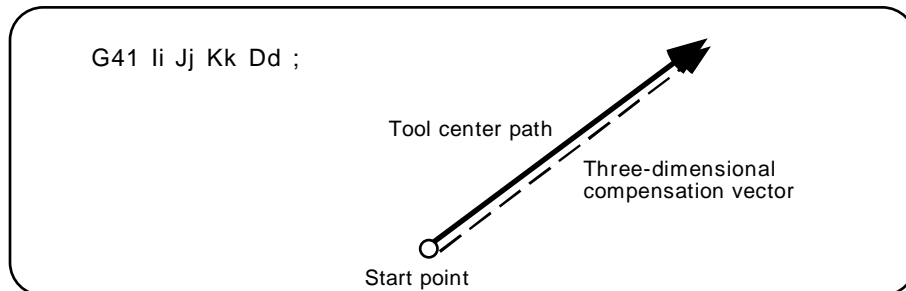
11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

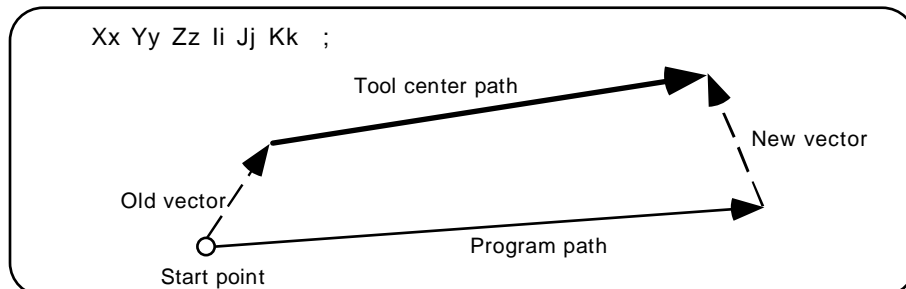
(Example 1) Radius compensation start: When there is a movement command



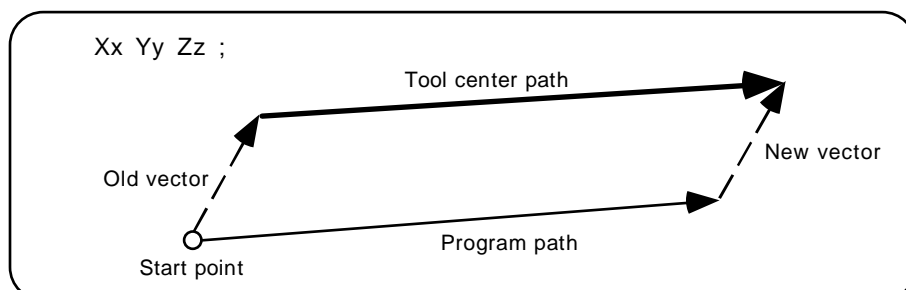
(Example 2) Radius compensation start: When there is no movement command



(Example 3) Movement during radius compensation: When there is a movement command and a plane normal line vector command



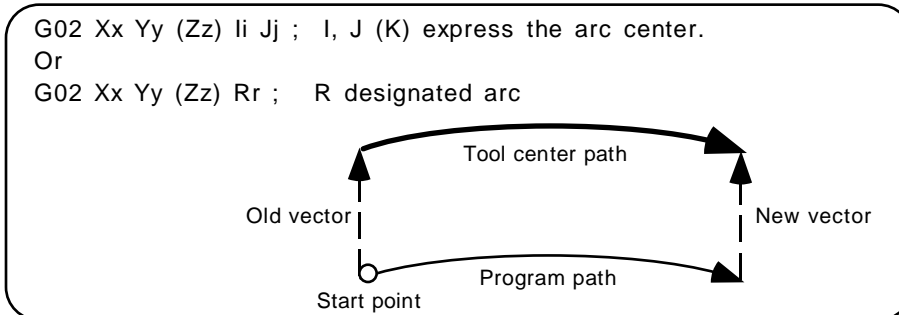
(Example 4) Movement during radius compensation: When there is no plane normal line vector command



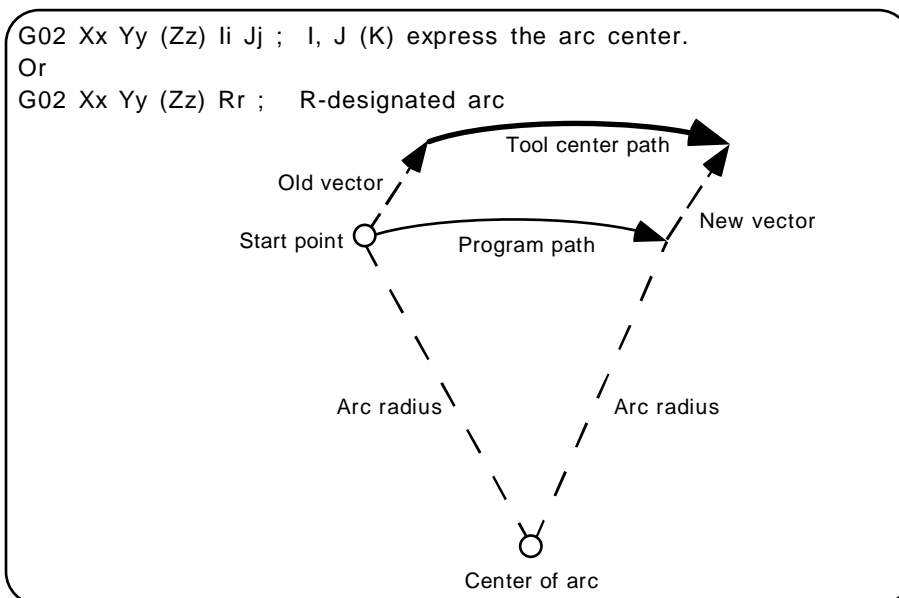
11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

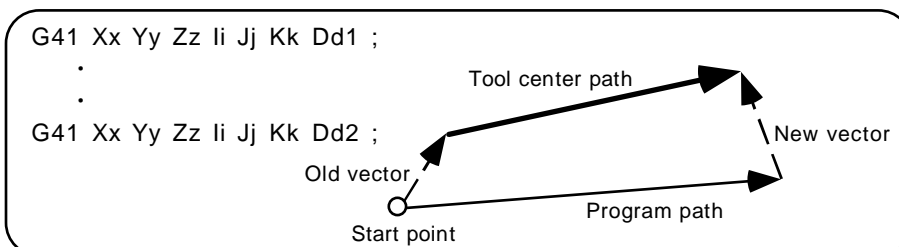
(Example 5) Movement during radius compensation: For arc or helical cutting
 The I, J, K command for an arc or helical cutting is the arc center command.
 The new vector is equivalent to the old vector.
 For the R command, even if I, J, K is commanded, they will be ignored, and the new
 vector will be equivalent to the old vector.



(Note) The center coordinate value will not shift for an arc or helical cutting.
 Thus, when the arc I, J, K is commanded with the following type of vector, the
 error "PR0070 Arc end point deviation large" will occur.



(Example 6) Movement during radius compensation: When offset amount is changed

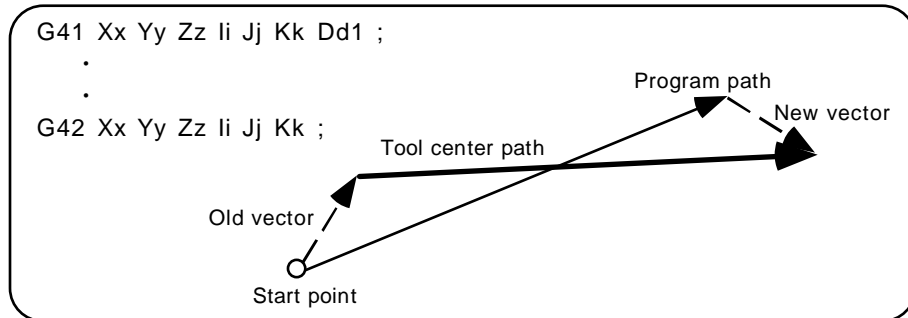


(Note 1) If I, J, K is not commanded in a block where the offset amount is changed, the
 vector will be equivalent to the old vector. In this case, the modal will change,
 but the offset amount will change when I, J, K is commanded.

11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

(Example 7) Movement during radius compensation: When offset direction is changed

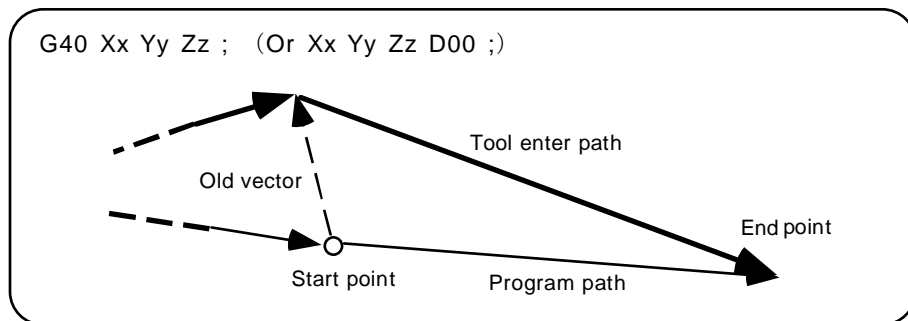


(Note 1) If I, J and K are not commanded in a block where the offset direction is to be changed, the vector will be equivalent to the old vector and the offset direction will not be changed

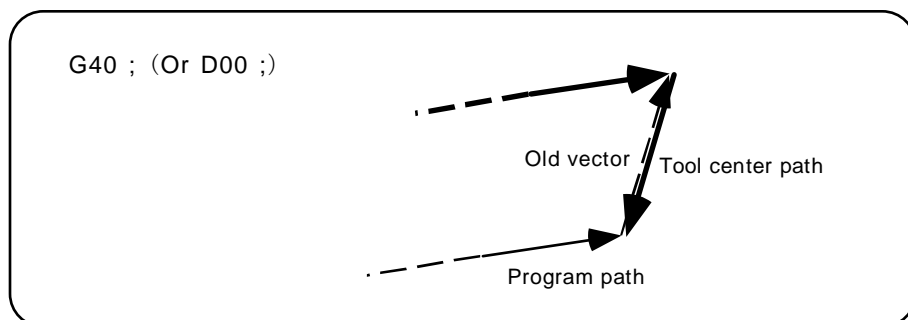
In this case, the modal will change, but the offset direction will change when I, J and K are commanded.

(Note 2) If the offset direction is changed in an arc (G2/G3) block, I, J will be the center of arc, so the offset direction will not change. For the R command, even if I, J and K are commanded, they will be ignored, and the offset direction cannot be changed.

(Example 8) Radius compensation cancel: When there is a movement command



(Example 9) Radius compensation cancel: When there is no movement command



11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

(3) Related items with other functions

- (a) Tool radius compensation

If the plane normal line vector (I, J, K) is not commanded for all three axes in the three-dimensional tool radius compensation start block, the mode will be handled as the normal tool radius compensation mode.

If G41 (G42) is commanded without commanding the plane normal line vector during three-dimensional tool radius compensation, the modal will change but the old vector will be used.

If G41 (G42) with the plane normal line vector is commanded during tool radius compensation, this command will be ignored and the normal tool radius compensation will take place.
- (b) Tool length offset

The tool length offset is applied on the coordinate values after three-dimensional tool radius compensation.
- (c) Tool position offset

The tool position offset is applied on the coordinate values after three-dimensional tool radius compensation.
- (d) Fixed cycle

A program error "PR0155 Fix cyc executed during compen" will occur.
- (e) Scaling

Scaling is applied on the coordinate values before three-dimensional tool radius compensation. Scaling is not applied on the plane normal line vector (I, J, K).

D1=10.

```
G90 ;
G51 X0 Y0 P0.5 ;
N1 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
N2 X-30. Y-30. Z-20. ;
N3 X-50. Y-20. Z-10. ;
N4 Y0. ;
```

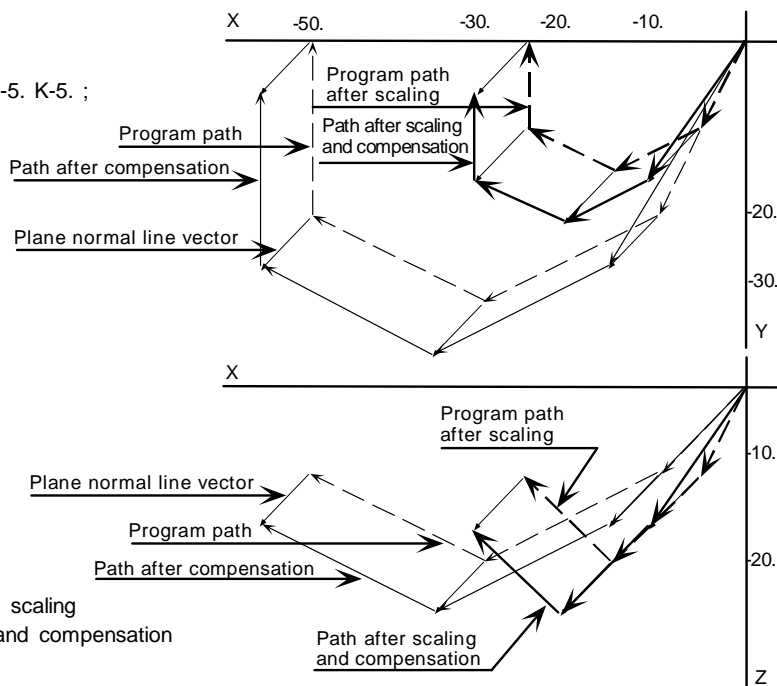
```
N1( -5.000, -10.000, -10.000 )
N1( -10.773, -15.773, -15.773 )
```

```
N2( -15.000, -15.000, -20.000 )
N2( -20.773, -20.773, -25.773 )
```

```
N3( -25.000, -10.000, -10.000 )
N3( -30.773, -15.773, -15.773 )
```

```
N4( -25.000, 0.000, -10.000 )
N4( -30.773, -5.773, -15.773 )
```

* Upper line: Program value after scaling
Lower line: Value after scaling and compensation



11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

(f) Program coordinate rotation

Program coordinate rotation is executed in respect to the coordinate values before three-dimensional tool radius compensation.

The plane normal line vector (I, J, K) dose not rotate.

D1=10.

```
G90 ;
G68 X0 Y0 R45. ;
N1 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
N2 X-30. Y-30. Z-20. ;
N3 X-50. Y-20. Z-10. ;
N4 Y0. ;
```

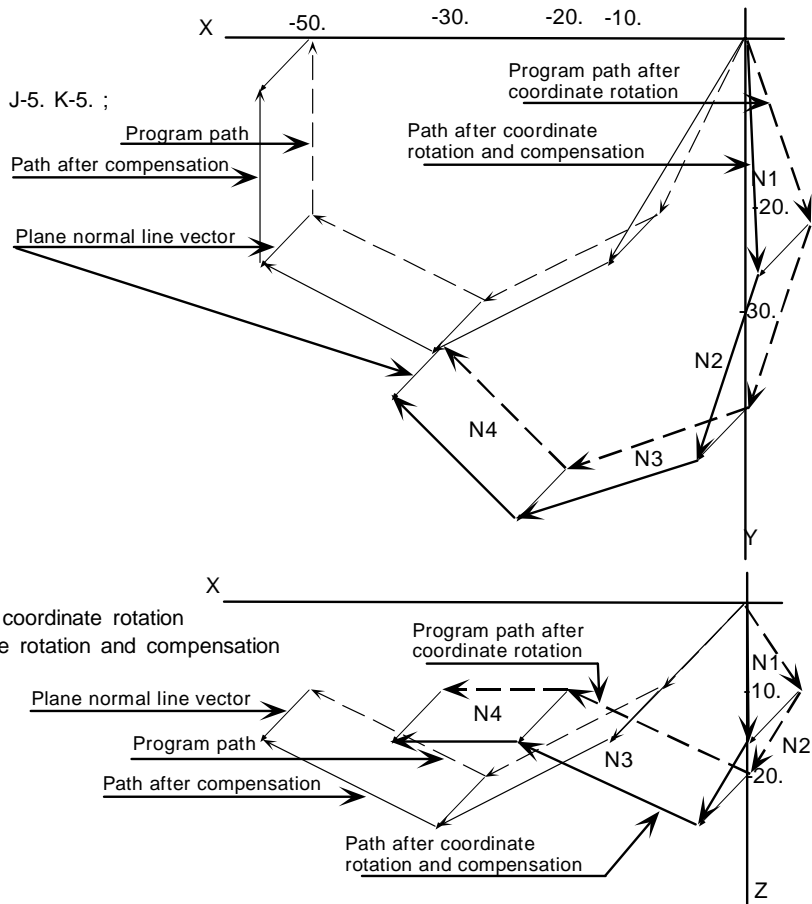
```
N1( 7.071, -21.213, -10.000 )
N1( 1.298, -26.986, -15.773 )
```

```
N2( 0.000, -42.426, -20.000 )
N2( -5.773, -48.199, -25.773 )
```

```
N3( -21.213, -49.497, -10.000 )
N3( -26.986, -55.270, -15.773 )
```

```
N4( -35.355, -35.355, -10.000 )
N4( -41.128, -41.128, -15.773 )
```

Upper line: Program value after coordinate rotation
Lower line: Value after coordinate rotation and compensation



(g) Parameter coordinate rotation

Parameter coordinate rotation is executed on the coordinate value after three-dimensional tool radius compensation.

The plane normal line vector (I, J, K) rotates.

(h) Mirror image

Mirror image is applied on the coordinate values after three-dimensional tool radius compensation.

Mirror image is applied on the plane normal line vector (I, J, K).

(i) Skip

The program error "PR0608 Skip during radius compen" will occur.

(j) Zero point compare

The compensation amount will not be canceled. Thus, if this is commanded during three-dimensional tool radius compensation, the path will be deviated by the offset amount, so the "PR0434 Verification error axis exists" will occur.

(k) Automatic corner override

Automatic corner override is invalid during three-dimensional tool radius compensation.

11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

(l) Machine coordinate system selection

1) For an absolute value command, all axes will be temporarily canceled at the commanded coordinate position.

D1=10.

G90 ;
 N1 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
 N2 X-30. Y-30. Z-20. ;
 N3 X-50. Y-20. Z-10. ;
 N4 G53 Y0 ;

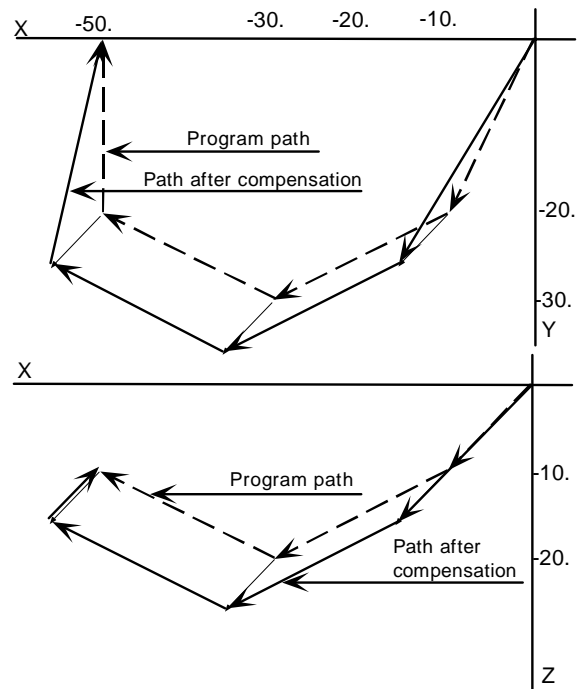
N1(-10.000, -20.000, -10.000)
 N1(-15.773, -25.773, -15.773)

N2(-30.000, -30.000, -20.000)
 N2(-35.773, -35.773, -25.773)

N3(-50.000, -20.000, -10.000)
 N3(-55.773, -25.773, -15.773)

N4(-50.000, 0.000, -10.000)
 N4(-50.000, 0.000, -10.000)

※ Upper line: Program value
 Lower line: Value after compensation



2) For an incremental value command, the axis will move by the amount obtained by subtracting each axis vector from the movement increment value.
 (The offset amount is temporarily canceled.)

D1=10.

G91 ;
 N1 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
 N2 X-20. Y-10. Z-10. ;
 N3 X-20. Y10. Z10. ;
 N4 G53 Y20. ;

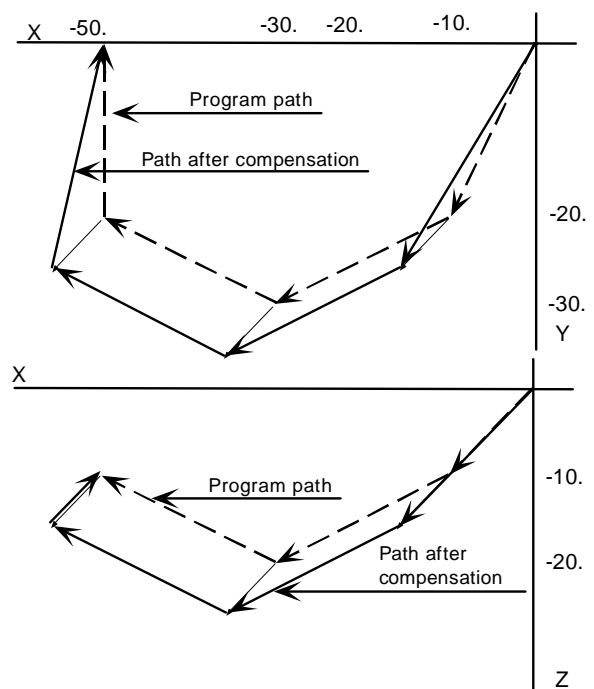
N1(-10.000, -20.000, -10.000)
 N1(-15.773, -25.773, -15.773)

N2(-30.000, -30.000, -20.000)
 N2(-35.773, -35.773, -25.773)

N3(-50.000, -20.000, -10.000)
 N3(-55.773, -25.773, -15.773)

N4(-50.000, 0.000, -10.000)
 N4(-50.000, 0.000, -10.000)

※ Upper line: Program value
 Lower line: Value after compensation



11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

(m) Coordinate system setting

When commanded in the same block as the coordinate system setting, the coordinate system will be set, and operation will start up independently with the plane normal line vector (I, J, K).

D1=10.

G91 ;
 N1 G92 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
 N2 X-20. Y-10. Z-10. ;
 N3 X-30. Y-10. Z10. ;
 N4 Y20. ;

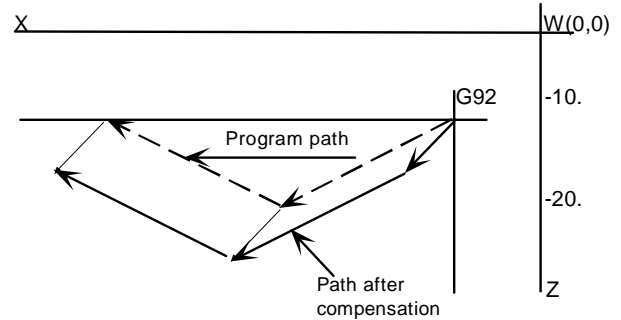
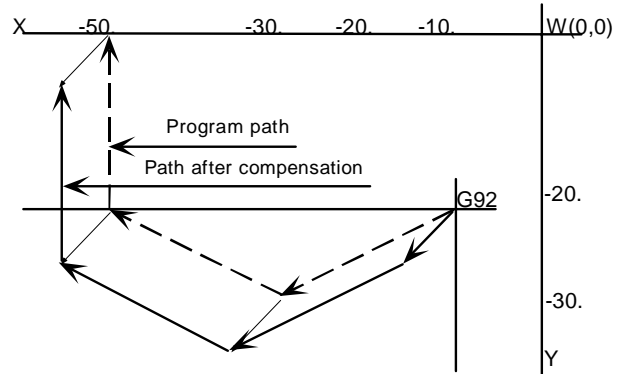
N1(-10.000, -20.000, -10.000)
 N1(-15.773, -25.773, -15.773)

N2(-30.000, -30.000, -20.000)
 N2(-35.773, -35.773, -25.773)

N3(-50.000, -20.000, -10.000)
 N3(-55.773, -25.773, -15.773)

N4(-50.000, 0.000, -10.000)
 N4(-55.773, -5.773, -15.773)

※ Upper line: Program value
 Lower line: Value after compensation



(n) Reference point return

All axes will be temporarily canceled at the intermediate point.

D1=10.

G91 ;
 N1 G41 D1 X-10. Y-20. Z-10. I-5. J-5. K-5. ;
 N2 X-20. Y-10. Z-10. ;
 N3 X-20. Y10. Z10. ;
 N4 G28 X0 Y0 Z0 ;

N1(-10.000, -20.000, -10.000)
 N1(-15.773, -25.773, -15.773)

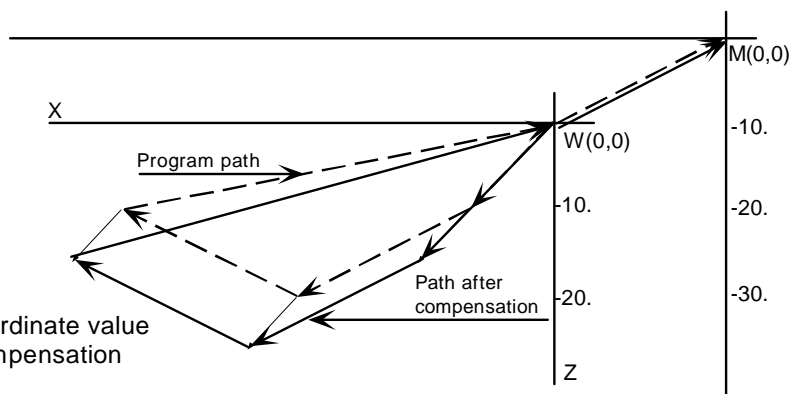
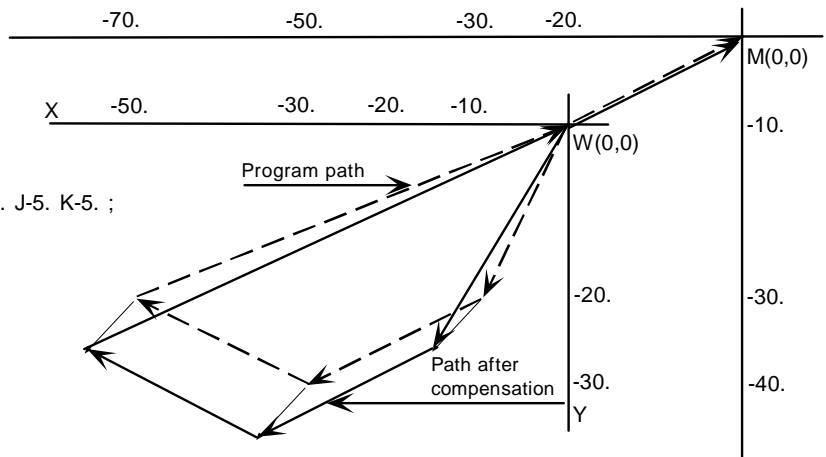
N2(-30.000, -30.000, -20.000)
 N2(-35.773, -35.773, -25.773)

N3(-50.000, -20.000, -10.000)
 N3(-55.773, -25.773, -15.773)

N4(0.000, 0.000, 0.000)
 N4(0.000, 0.000, 0.000)

N4(20.000, 10.000, 10.000)
 N4(20.000, 10.000, 10.000)

※ Upper line: Workpiece coordinate value
 Lower line: Value after compensation



11. TOOL COMPENSATION

11.3 Three-dimensional Tool Radius Compensation; G40/G41/G42

- (o) NC reset
The three-dimensional tool radius compensation will be canceled if NC reset is executed during three-dimensional tool radius compensation.
- (p) Emergency stop
The three-dimensional tool radius compensation will be canceled by the emergency stop or emergency stop cancel during three-dimensional tool radius compensation.

(4) Restrictions

- (a) The compensation number is selected with the D command, but the D command is valid only when G41 and G42 are commanded. If D is not commanded, the number of the previous D command will be valid.
- (b) Change to the compensation mode in the G00 or G01 mode. When changed during the arc mode, the program error "PR0150 Radius compen in arc modal" will occur.
The compensation direction and offset amount after the changes are valid from the block where I, J and K are commanded in the G00 or G01 mode.
If the three-dimensional tool radius compensation command is used in a block not containing the plane normal line vector (I, J, K) during the arc mode, only the modal information will change. The plane normal line vector will be validated from the block where I, J and K are commanded next.
- (c) During the three-dimensional tool radius compensation mode in a certain space, the space cannot be changed to another space and three-dimensional tool radius compensation cannot be carried out. To change the compensation space, always carry out cancel (G40 or D00).

(Example)

G41 Xx Yy Zz Ii Jj Kk; Compensation starts in X, Y, Z space.

·
·

G41 Uu Yy Zz Ii Jj Kk; Compensation is carried out in X, Y, Z space, and U axis moves by commanded value.

- (d) If the compensation number D exceeds 1 to 40 with the standard specifications or 1 to 800 (max.) with the additional specifications, the program error "PR0170 No compensation number" will occur.
- (e) Only the G40 and D00 commands can be used to cancel three-dimensional tool radius compensation.
- (f) If the size ($I^2+J^2+K^2$) of the vector commanded with I, J and K overflows, the program error "PR0035 Address range over" will occur.

11. TOOL COMPENSATION

11.4 Shape Compensation Amount, Wear Compensation Amount

11.4 Shape Compensation Amount, Wear Compensation Amount

11.4.1 Outline

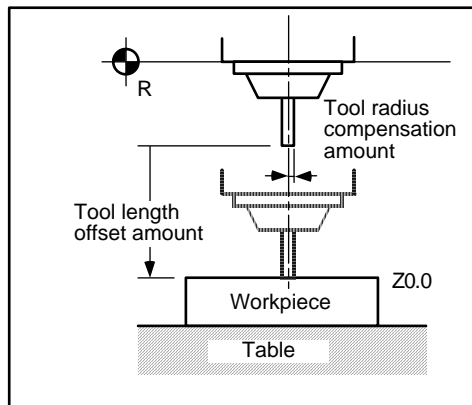
Of the tool positions in a direction parallel to the control axis, the tool shape offset amount and wear compensation amount are registered. The compensation may extend over two or more axes.

11.4.2 Detailed description of shape compensation amount

- (a) Of the tool position offsets in a direction parallel to the control axis, the rotation tool's longitudinal direction offset amount is registered as the tool length offset amount. The tool length offset amount includes type I and type II.

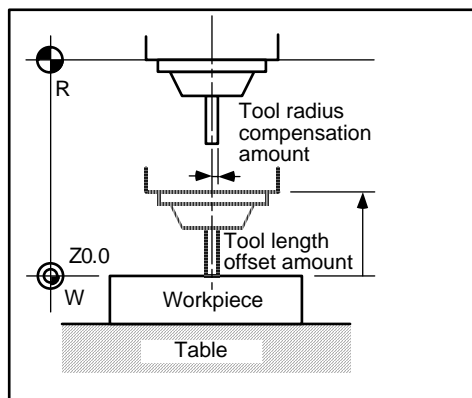
1) Type I

The distance from the tip of the tool to the workpiece upper end when the tool position is at the reference point is registered as the shape compensation amount. The tool length offset amount is set with a negative value.



2) Type II

The distance from the reference point to the tip of the tool when the tool position is at the reference point is measured and registered as the shape compensation amount. The tool length offset amount is set with a positive value.



- (b) Of the tool position compensates in a direction parallel to the control axis, the rotation tool's radius direction compensation amount is registered as the tool radius compensation amount. The tool radius compensation amount is set with a positive value.

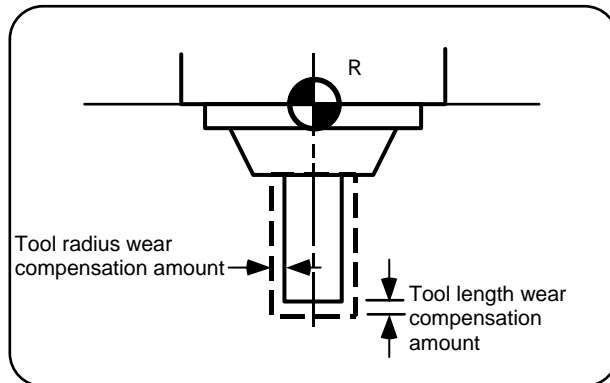
11. TOOL COMPENSATION

11.4 Shape Compensation Amount, Wear Compensation Amount

11.4.3 Detailed description of wear compensation amount

The tool length wear compensation amount and tool radius wear compensation amount can be set independently.

If a negative value is set for the tool length compensation amount, a negative value must be set for the tool length wear compensation amount. Set a positive value for the tool radius wear compensation amount.



11.5 Number of Compensation Sets

(1) Outline

The number of tool compensation sets can be selected with parameters.

(2) Detailed description

The number of tool compensation sets can be selected as follows.

(a) Number of 1-axis compensation amount sets

40 sets	Standard specifications
100 sets	Additional specifications
200 sets	Additional specifications
400 sets	Additional specifications
800 sets	Additional specifications
990 sets	Additional specifications

(b) Number of 2-axis compensation amount sets

20 sets	Standard specifications
40 sets	Additional specifications
80 sets	Additional specifications

11.6 Tool Position Offset; G45/G46/G47/G48

(1) Outline

By using this command, movement is controlled so that the end point position of the movement command is changed to a position extended or reduced by the tool offset amount.

With this function, the difference in the distance between program coordinate position and actual machine tool installation position caused by the tool position can be offset, allowing machining and operation efficiency to be improved.

However, this function has been prepared for compatibility with conventional machining programs not provided with the tool radius compensation function. Thus, do not use this function when creating a new machining program.

(2) Command format

<u>G45</u>	<u>G00</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Dd1</u>	<u>(Hh1)</u>	;
Offset		Movement axis	Movement axis	Offset number		

G45 command	G46 command	G47 command	G48 command
Extend by offset amount	Reduce by offset amount	Extend by double the offset amount	Reduce by double the offset amount
<div style="display: flex; align-items: center;"> <div style="writing-mode: vertical-rl; transform: rotate(180deg); font-size: small; margin-right: 5px;">Program command</div> </div>	<div style="display: flex; align-items: center;"> <div style="writing-mode: vertical-rl; transform: rotate(180deg); font-size: small; margin-right: 5px;">Program command</div> </div>	<div style="display: flex; align-items: center;"> <div style="writing-mode: vertical-rl; transform: rotate(180deg); font-size: small; margin-right: 5px;">Program command</div> </div>	<div style="display: flex; align-items: center;"> <div style="writing-mode: vertical-rl; transform: rotate(180deg); font-size: small; margin-right: 5px;">Program command</div> </div>

(3) Detailed description

(a) When the start point and end point are on the axis, the radius can be extended or reduced only in respect to a 1/4 or 3/4 circle.

- a. For 1/4 circle, 3/4 circle
 - To offset to the outer side of the circle: G45
 - To offset to the inner side of the circle: G46
- b. For 1/2 circle
 - To offset to the outer side of the circle: G47
 - To offset to the inner side of the circle: G48

Note that in this case, the arc start point must already be offset in the required direction.

(If offset is commanded independently to the arc, the arc start point radius and arc end point radius will be deviated by the offset amount.)

(b) When an absolute value is commanded, the movement from the previous block's end point to the position commanded in the block containing G45 (to G48) will be extended or reduced in each axis direction. Thus, when an absolute value is commanded, the offset will be applied to the movement amount in that block. If the movement amount is zero, the movement will stop immediately, and the offset amount will not be applied.

11. TOOL COMPENSATION
11.6 Tool Position Offset; G45/G46/G47/G48

(c) The correspondence for the incremental value is shown below.

Command	Equivalent command movement amount (Designated offset amount = I)	Actual example (When X = 1000)
G45Xx Dd (Hh)	$X(x + I)$	I = 10 X = 1010 I = -10 X = 990
G45X - x Dd (Hh)	$X - (x + I)$	I = 10 X = -1010 I = -10 X = -990
G46Xx Dd (Hh)	$X(x - I)$	I = 10 X = 990 I = -10 X = 1010
G46X - x Dd (Hh)	$X - (x - I)$	I = 10 X = -990 I = -10 X = -1010
G47Xx Dd (Hh)	$X(x + 2 * I)$	I = 10 X = 1020 I = -10 X = 980
G47X - x Dd (Hh)	$X - (x + 2 * I)$	I = 10 X = -1020 I = -10 X = -980
G48Xx Dd (Hh)	$X(x - 2 * I)$	I = 10 X = 980 I = -10 X = 1020
G48X - x Dd (Hh)	$X - (x - 2 * I)$	I = 10 X = -980 I = -10 X = -1020

(d) If n axes are commanded simultaneously, the same amount of offset will be applied on all axes commanded within the range of the simultaneous control axes.
The tool position offset is also valid for the additional axis.

(e) Do not use this command in the fixed cycle mode. If commanded in the fixed cycle mode, it will be ignored.

(f) If the command direction is reversed because of the internal operation by the extension and retraction, the axis will move in the reverse direction.

(g) When G45 to G48 is commanded, the offset amount per time is the offset amount designated with the offset number. The difference with the previous offset will not be moved as with the tool length offset (G43).

(h) The compensation amount programmed in the number specified in the D or H command is taken out from tool offset memory.

① Tool offset memory (type I)

A distinction between length compensation and radius compensation is not made. Thus, the same offset is taken out regardless of whether it is commanded with H or D.

② Tool offset memory (type II)

A distinction between length compensation and radius compensation is made. Offset corresponding to the specified number is taken out from length compensation offset when H command is given; it is taken out from radius compensation offset when D command is given.

(4) Notes

(a) Even if the offset number is not commanded in the same block as the G45 to G48 commands, the offset will be applied with the previously registered tool position offset number. An alarm will occur if the designated offset number exceeds the specifications.

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.1 Automatic Coordinate System Setting

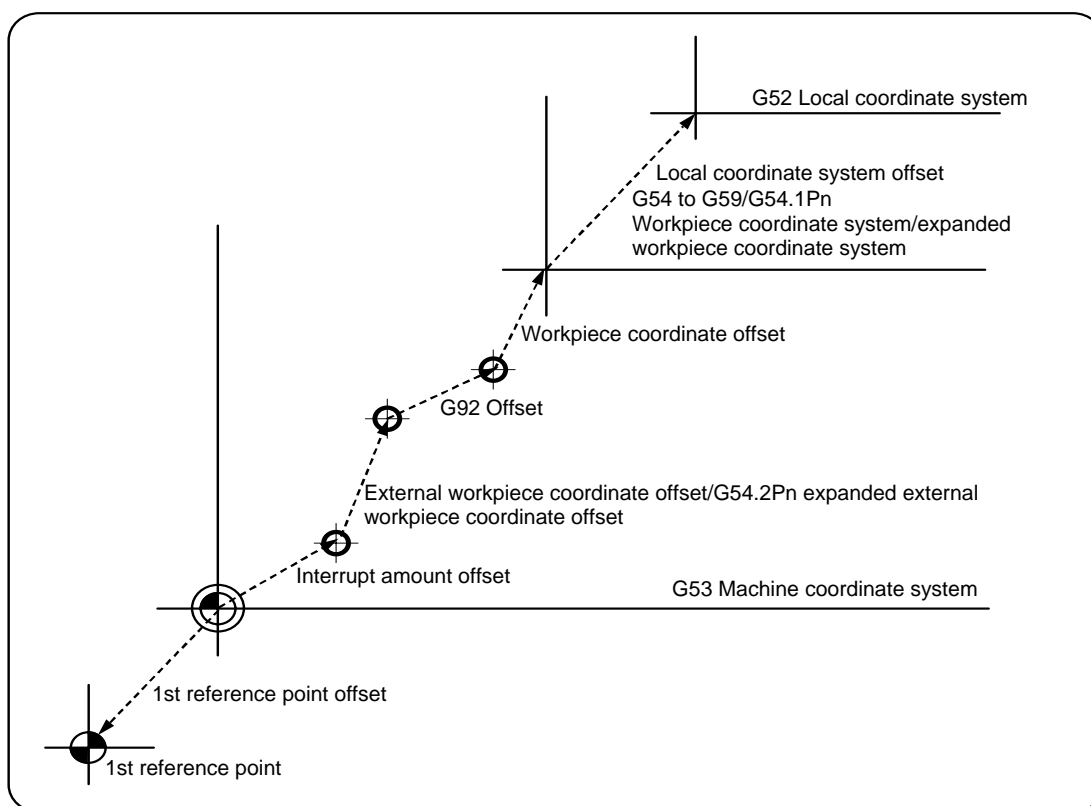
(1) Outline

The coordinate systems are created with parameter settings in the following cases:

- When the reference point is reached with the first manual or automatic dog-type reference point return after the NC power is turned ON.
- Immediately after the power is turned ON when using the absolute position specifications.

The actual machining program is programmed on the coordinate system set above.

The coordinate system handled by the NC is shown below. The points that can be specified with the movement commands are points on the workpiece coordinate system (local coordinate system when local coordinate offset is designated) or on the machine coordinate system.



(2) Detailed description

(a) The following coordinate systems are automatically established by this function.

- 1) Basic machine coordinate system corresponding to G53
- 2) External workpiece coordinate system
- 3) G54 to G59, G54.1 (additional workpiece coordinate system) workpiece coordinate system

The offset amount of the local coordinate system (G52) is canceled.

(b) All parameter set in the NC are assigned as the distance from the basic machine coordinate system's zero point. Thus, decide which position on the basic machine coordinate system to place the 1st reference point, and then set the zero point position of each workpiece coordinate system.

(c) The local coordinate system offset is independent for each of G54 to G59.

(d) During the G54.1Pn (expanded workpiece coordinate system selection) modal, the local coordinate system offset is zero, and the G52 command cannot be used.

(e) G54.1Pn and G54.2Pn cannot both be used. Use only one of these.

12.2 Machine Coordinate System

(1) Outline

The machine coordinate system is a coordinate system that indicates positions (tool change position, stroke end position, etc.) characteristic to the machine. These coordinates are automatically set when the first dog-type reference point return is completed after the power is turned ON, or immediately after the power is turned ON when using the absolute position specifications.

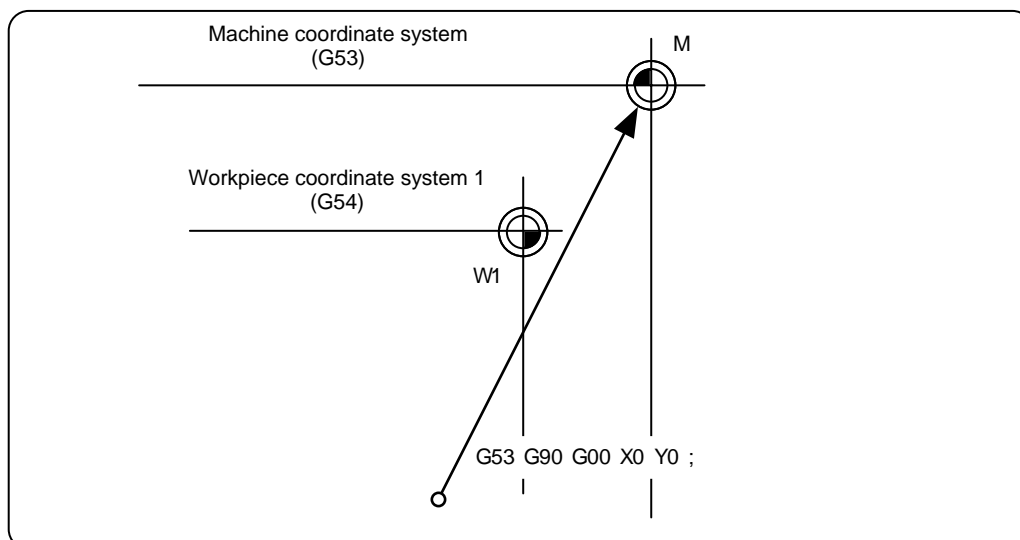
(2) Command format

The program format for movement commands to points on the machine coordinate system is as follows.

<u>G53</u>	<u>G90</u>	<u>G00</u>	<u>Xx1 Yy1 Zz1</u> ;
Coordinate system selection	Absolute/incremental command	Movement mode	End point coordinate on machine coordinate system

(3) Detailed description

- (a) The machine coordinate system is automatically set when the first dog-type reference point return is completed after the power is turned ON, or immediately after the power is turned ON when using the absolute position specifications.
- (b) The machine coordinate system does not change with the coordinate system setting (G92) command.
- (c) The G53 (movement on machine coordinate system) command is an unmodal command that is valid only in the commanded block.
- (d) The G53 command moves over the selected coordinate system with an incremental value during the incremental value command mode (G91).
- (e) Even if G53 is commanded, the compensation amount of the commanded axis' tool radius will not be canceled.
- (f) The 1st reference point coordinate value indicates the distance from the machine coordinate system's zero point to the reference point (zero point) return position.
- (g) Whether to use a fixed rapid traverse or a commanded modal for the movement mode can be changed with parameters.
- (h) If the absolute/incremental command is omitted, the modal at that time will be followed.
- (i) The selected workpiece coordinate system will not change with the G53 command.
- (j) If the G53 command and zero point return (G28) command are commanded in the same block the latter command will be valid.



12. COORDINATE SYSTEM SETTING FUNCTIONS
12.3 Workpiece Coordinate System; G54 to G59

12.3 Workpiece Coordinate System; G54 to G59

(1) Outline

(a) Workpiece coordinate system (Six sets)

This function allows the same shape to be machined easily when machining multiple workpieces with the same shape. This is carried out by executing one machining program for the coordinate system of each workpiece.

Up to six workpiece coordinate systems can be selected.

The G54 workpiece coordinate system is selected when the power is turned ON and then the reset signal that cancels the modal information is input.

(b) Expanded workpiece coordinate system selection (+48 sets or 96 sets) (Option)

In addition to the six workpiece coordinate system sets in G54 to G59, 48 sets (or 96 sets) of workpiece coordinate systems can be used with the G54.1Pn command.

(2) Command format

(a) Workpiece coordinate system (Six sets)

The format for workpiece coordinate system selection and the movement command to the workpiece coordinates is as follows.

<u>(G90)</u>	<u>G54</u>	<u>G00</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u> ;
(Absolute value)	Coordinate sys-tem designation	Movement mode	End point coordinate value

G code	Function
G54	Workpiece coordinate system 1 selection (W1)
G55	Workpiece coordinate system 2 selection (W2)
G56	Workpiece coordinate system 3 selection (W3)
G57	Workpiece coordinate system 4 selection (W4)
G58	Workpiece coordinate system 5 selection (W5)
G59	Workpiece coordinate system 6 selection (W6)

(b) Expanded workpiece coordinate system selection (+48 sets or 96 sets) (Option)

The format for the workpiece coordinate system selection and movement command to the workpiece coordinates using the G54.1Pn command is as follows.

<u>(G90)</u>	<u>G54.1Pn</u>	<u>G00</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u> ;
(Absolute value)	Coordinate sys-tem designation	Movement mode	End point coordinate value

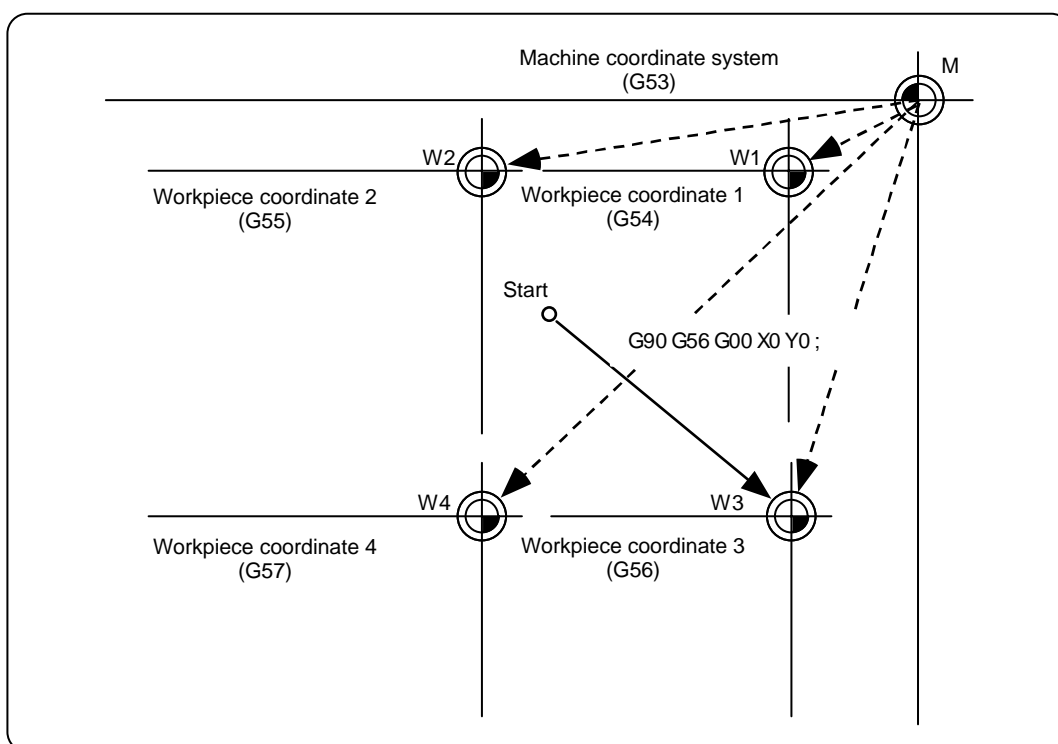
The n value for P following G54.1 indicates each workpiece coordinate system. Designate 1 to 48 or 1 to 96.

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.3 Workpiece Coordinate System; G54 to G59

(3) Detailed description

- (a) With any of the G54 to G59 commands, the tool radius compensation amounts for the assigned axis will not be canceled even if workpiece coordinate system selection is assigned.
- (b) The G54 workpiece coordinate system is selected when the power is turned ON.
- (c) Commands G54 to G59 are modal.
- (d) Each workpiece coordinate zero point is assigned as the distance from the machine coordinate system's zero point.
- (e) The workpiece coordinate system's offset setting amount can be changed as often as necessary.
- (f) The following three setting methods can be used.
 - 1) Setting with setting and display unit
 - 2) Setting with commands from machining program (G10)
 - 3) Setting from user PLC



- (g) The coordinate system will move with G92 in a workpiece coordinate system.
- (h) A new workpiece coordinate system 1 is set by setting the coordinate system (G92) command in the G54 (workpiece coordinate system 1) mode. At the same time, the other workpiece coordinate systems 2 to 6 (G55 to G59) will move in parallel and new workpiece coordinate systems 2 to 6 will be set.
- (i) A hypothetical machine coordinate system is formed at the position which deviates from the new workpiece reference point by an amount equivalent to the workpiece coordinate system offset amount.
- (j) By setting the hypothetical machine coordinate system, the new workpiece coordinate system will be set at a position which deviates from that hypothetical machine coordinate system zero point by an amount equivalent to the workpiece coordinate system offset amount.
- (k) When the first automatic reference point return (G28) or manual reference (zero) point return is completed after the power has been turned ON, the basic machine coordinate system and workpiece coordinate systems are set automatically in accordance with the parameter settings.
- (l) If no P code exists in the same block as G54.1, the input is treated as P1.

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.3 Workpiece Coordinate System; G54 to G59

(4) Notes

- (a) The program error "PR0062" will occur when G54X_Y_ ; is assigned subsequent to reference point return (automatic or manual) after the power is turned ON. (A speed command is required since the speed is controlled by G01.)
- (b) If the numeric value of P that follows G54.1 is other than 1 to 96 (including P0), the program error "PR0035" will occur.
- (c) Do not use any G command for which a P code is to be used in the same block as G54.1.
- (d) If the G54.1 command is executed when there are no additional specification for the number of sets of the workpiece offset, the program error "PR0437" will occur.
- (e) The local coordinate system cannot be used when the G54.1 is modal. If the G52 command is executed while G54.1 is modal, the program error "PR0438" will occur.

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.4 Local Coordinate System; G52

12.4 Local Coordinate System; G52

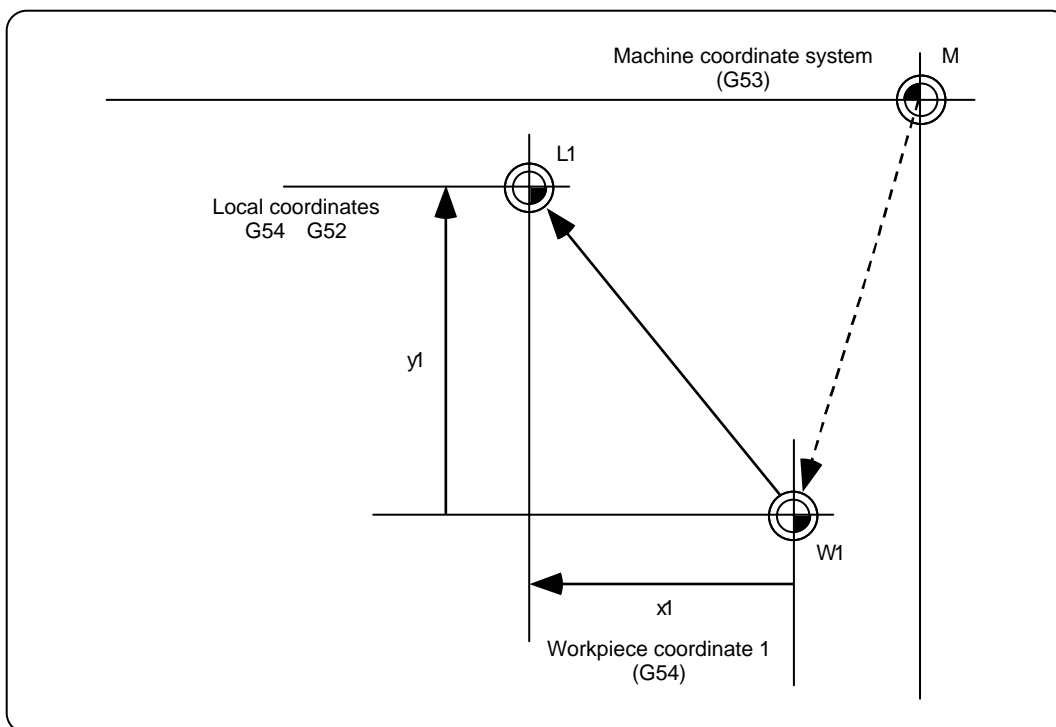
(1) Outline

This function is used to designate a further coordinate system within the currently selected workpiece coordinate system. Using this, the workpiece coordinate system can be changed temporarily.

A local coordinate system in which the commanded position is the program zero point can be set on the workpiece coordinate system.

The local coordinate system can be designated independently for each of the G54 to G59 workpiece coordinate systems.

This can also be used to instead of changing the deviation of the machining program zero point and machining workpiece zero point with the coordinate system setting (G92) command.



(2) Command format

The format for designating the local coordinate system is as follows.

<u>(G54)</u>	<u>G52</u>	<u>Xx1 Yy1 Zz1</u> ;
Workpiece coordinate selection	Local coordinate setting	Local coordinate offset

G code	Function
G54 G52	Local coordinate system setting on workpiece coordinate system 1
G55 G52	Local coordinate system setting on workpiece coordinate system 2
G56 G52	Local coordinate system setting on workpiece coordinate system 3
G57 G52	Local coordinate system setting on workpiece coordinate system 4
G58 G52	Local coordinate system setting on workpiece coordinate system 5
G59 G52	Local coordinate system setting on workpiece coordinate system 6
G52X0Y0Z0α0 ;	Local coordinate system cancel

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.4 Local Coordinate System; G52

(3) Detailed description

- (a) The G52 command is an unmodal command, but it is valid until a new G52 command is issued. The G52 command does not change the zero point position of the workpiece coordinate system (G54 to G59), and is handy for using another coordinate system.
- (b) The local coordinate system is canceled when the reset signal is input, or when manual or automatic dog-type reference point return is carried out.
- (c) The zero point position of the local coordinate system is commanded as the distance (local coordinate offset) from the zero point of the designated workpiece coordinate system.
- (d) If the workpiece coordinate system is not designated, the local coordinate system will be created on the currently selected workpiece coordinate system.
- (e) The local coordinate system is canceled with the following program.

`(G54 to G59) G52X0Y0Z0α0 ;`

- (f) The coordinate commands in the absolute value mode (G90) will cause movement to the position in the local coordinate system.

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.5 External Workpiece Coordinate Offset

12.5 External Workpiece Coordinate Offset

(1) Outline

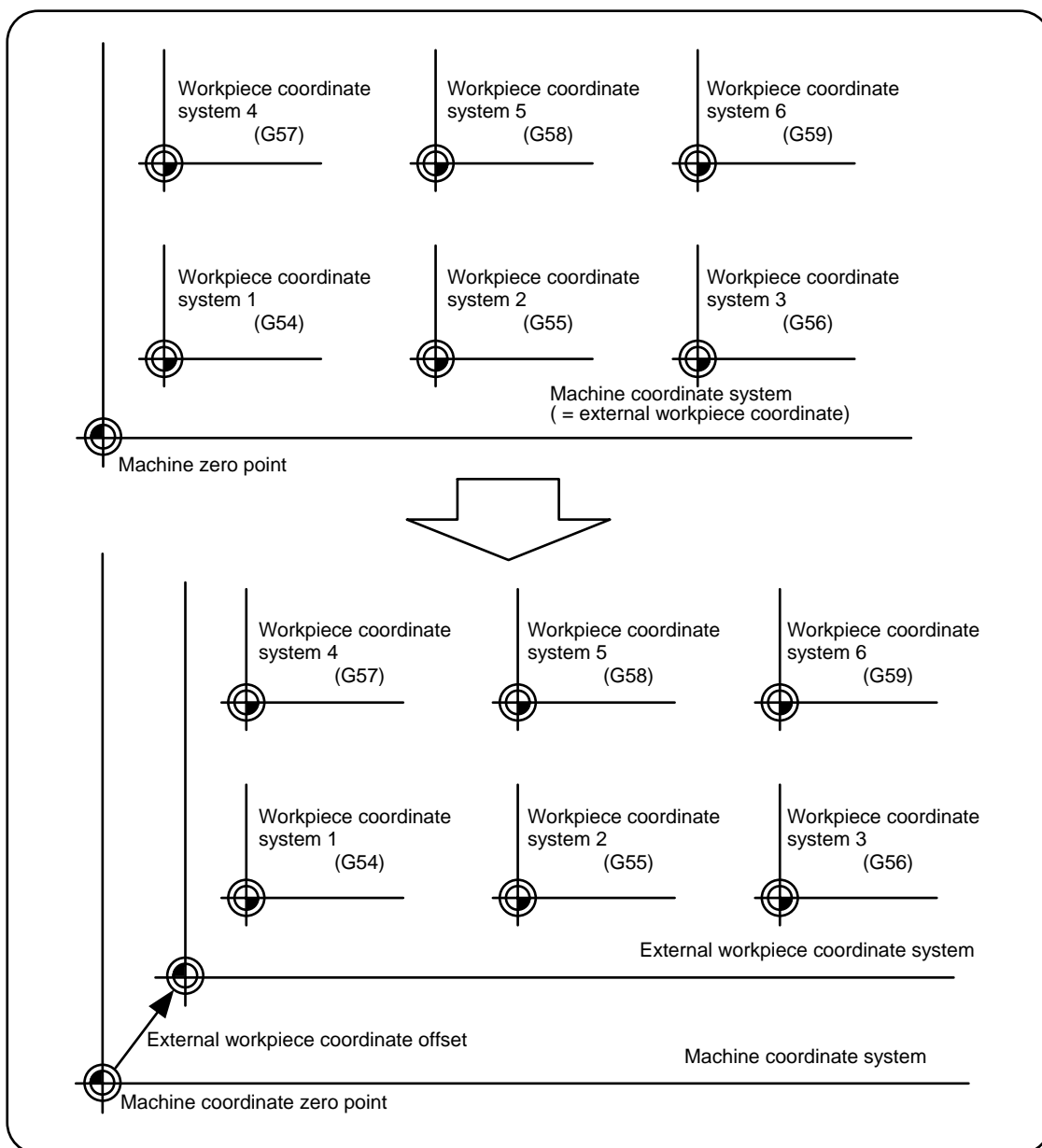
An external workpiece coordinate offset used as the reference for all workpiece coordinate systems is provided externally from the workpiece coordinates.

By setting the external workpiece coordinate offset, these can be shifted, and all workpiece coordinate systems can be shifted simultaneously by that offset amount.

(2) Detailed description

(a) If the external workpiece coordinate offset is zero, the external workpiece coordinate system will match the machine coordinate system.

(b) A movement command cannot be executed when the external workpiece coordinates are selected.



(c) In addition to the normal external workpiece coordinate system offset, 48 sets (or 96 sets) of expanded external workpiece coordinate system offsets can be set and selected as an option.

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.6 Coordinate System Setting; G92

12.6 Coordinate System Setting; G92

(1) Outline

The absolute value coordinate system in the program and the currently displayed position value can be preset to newly commanded values without moving the machine or using the workpiece coordinate system.

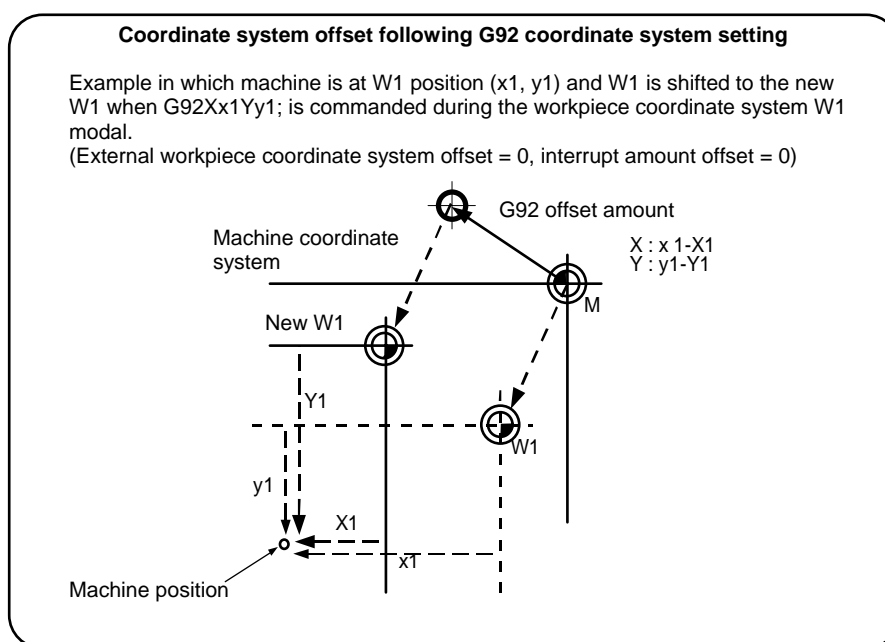
(2) Command format

<u>G92</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u> ;
Coordinate system setting	Setting coordinate values

(3) Detailed description

- (a) If the coordinate system setting by G92 is commanded, the G92 offset amount will be inserted and the workpiece coordinate system will be shifted so that the machine position in the current workpiece coordinate system becomes the coordinate value commanded with G92 as shown below.

The machine does not move, and all workpiece coordinate systems from G54 to G59 based on the machine coordinate system (or the external workpiece coordinate system when the external workpiece coordinate offset is set) will be shifted.



- (b) The shifted coordinate system will return to the original system when dog-type reference point return or the program is executed.
- (c) If the manual absolute switch is OFF and manual axis movement is carried out, etc., causing the workpiece coordinate system to deviate, the correct workpiece coordinate system can be recovered with the following procedure.
- 1) Carry out reference point return while the coordinate system is deviated.
 - 2) Then command G92G53X0Y0Z0;. With this command, the workpiece coordinate value and current value display will be preset to the workpiece coordinate system offset value.
- (d) If S is commanded in the same block as G92 during the constant surface speed (G96) modal, the S command will be handled as the spindle clamp speed for the constant surface speed control (G96), and the normal S command will not be output.
- (e) Whether to always move with absolute values or to follow the incremental value/absolute value modal can be selected with the machine parameters.
- (f) Whether to preset the current position counter can be selected with the parameters.

12. COORDINATE SYSTEM SETTING FUNCTIONS
12.7 Rotation Axis Coordinate System

12.7 Rotation Axis Coordinate System

(1) Outline

The axis designated as the rotation axis with the parameters is controlled with the rotation axis coordinate system. The rotation axis coordinate system is from 0 to $\pm 360^\circ$, but the counter display is 0 to 359.999.

(2) Detailed description

- (a) When using incremental value commands, the axis will move by the commanded movement amount. However, when using absolute value commands, the movement amount will be within one rotation (within $\pm 360^\circ$). In either case, the rotation direction is determined by the sign. If commanded with a (-), the axis will move in the (-) direction, and if commanded with a (+), the axis will move in the (+) direction.
- (b) In respect to the absolute value command, the axis can be moved with a short cut to the end point by setting the parameters, or can be moved linearly like a normal linear axis as shown below.

These parameters can be set for each axis.

	Rotation type		Linear type
	Shortcut valid	Shortcut invalid	
ABS command	Moves with a shortcut to the end point.	Moves according to the sign for the remainder obtained by dividing the incremental movement amount by 360° .	Moves according to the sign for the movement amount obtained by subtracting the current value from the end point as with the normal linear axis. (The value is not rounded to 360° .)
INC command	Moves in the direction of the commanded sign by the incremental amount commanded using the current position as the start point.		

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.8 Plane Selection; G17/G18/G19

12.8 Plane Selection; G17/G18/G19

(1) Outline

These are the G command codes used to specify the commanded plane for the arc command, tool radius compensation and coordinate rotation command, etc.

(2) Command format

G17 Xp, Yp plane designation
G18 Zp, Xp plane designation
G19 Yp, Zp plane designation

(3) Detailed description

- (a) Which of the X, Y or Z axes to keep the additional axis parallel to can be selected with parameters. By using this, the plane can be selected by two random axes that are not parallel. If the rotation axis is registered as a parallel axis, a plane containing the rotation axis can be selected.
- (b) Which of the G17, G18 or G19 states to set the initial state to (state when power is turned ON or reset) can be selected with parameters.
- (c) The Y-Z cylindrical plane (hypothetical plane that develops cylinder) can be selected with G16 only in the milling mode.
- (d) The movement command and plane selection can be commanded in the same block.

<p>G19 X100. ;</p> <p>G17 X100. R50. ;</p>	<p>In this program command, X100. is an axis that does not exist in the G19 (Yp, Zp) plane. However, YpZp is selected by G19, and the X axis moves 100.mm separately from G19 plane.</p> <p>In this program command, the Xp, Yp plane is selected with the G17, so the arc command is controlled over this plane.</p>
--	---

- (e) Plane selection can be used to select the following planes.
 - Plane for circular interpolation (including helical cutting)
 - Plane for tool radius compensation
 - Plane for pattern rotation
 - Plane for program coordinate rotation
 - Plane for fixed cycle positioning

(4) Parallel axis

Table 1 Examples of plane selection parameter registration

	Basic axis	Parallel axis 1	Parallel axis 2
I	X	U	W
J	Y		
K	Z	V	B

- (a) As shown in the Table 1, the basic axis and its parallel axis can be registered up to the 2nd parallel axis.
- (b) The basic axis can be an axis other than X, Y and Z.
- (c) An axis that is not registered is irrelevant for the plane selection.

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.8 Plane Selection; G17/G18/G19

(5) Plane selection method

(a) Table 1 indicates the following.

I is the G17 plane horizontal axis or G18 plane vertical axis.
J is the G17 plane vertical axis or G19 plane horizontal axis.
K is the G18 plane horizontal axis or G19 plane vertical axis.

Thus,

G17 IJ plane
G18 KI plane
G19 JK plane

(b) The plane selected by the basic axis and one of the parallel axis depends on the axis address commanded in the same block as the plane selection (G17, G18, G19).

For Table 1 parameter registration example

G17X_Y_ ; XY plane
G17W_Y_ ; WY plane
G18X_V_ ; VX plane
G18U_V_ ; VU plane
G19Y_Z_ ; YZ plane
G19Y_V_ ; YV plane

(c) The plane will not change in a block where the plane selection G code (G17, G18, G19) is not commanded.

G17X_Y_ ; XY plane
Y_Z_ ; XY plane (Plane does not change)

(d) If the axis address is omitted in the block where the plane selection G code (G17, G18, G19) is commanded, it will be interpreted that the basic three axes addresses have been omitted.

For Table 1 parameter registration example

G17 ; XY plane
G17U_ ; UY plane
G18U_ ; ZU plane
G18V_ ; VX plane
G19Y_ ; YZ plane
G19V_ ; YV plane

(e) The axis command that does not exist in the plane determined by the plane selection G code (G17, G18, G19) is irrelevant to the plane selection.

For Table 1 parameter registration example

If G17 U_Z_ is commanded, the UV plane is selected, and Z moves without relevance to the plane.

(f) If the basic axis or parallel axis is commanded in duplicate in the same block as the plane selection G code (G17, G18, G19), the plane will be determined using the basic axis, No. 1 parallel axis and No. 2 parallel axis as the order of priority.

For Table 1 parameter registration example

If G17 U_Y_W_ is commanded, the UY plane is selected, and W moves without relevance to the selected plane.

12. COORDINATE SYSTEM SETTING FUNCTIONS
12.9 Automatic 1st Reference Point Return; G28/G29

12.9 Automatic 1st Reference Point Return; G28/G29

(1) Outline

By commanding G28 during automatic operation, the axis can be returned to the 1st reference point. If the middle point is commanded, the axis will be positioned to that point at rapid traverse, and then will independently return to the 1st reference point.

By commanding G29, after positioning each axis independently to the G28 or G30 middle point, the axis can be positioned to the commanded position.

G code	Function
G28	Automatic 1st reference point return
G29	Start position return (Return from 1st reference point to middle point position for 1st reference point return start, and then position to position designated by program.)

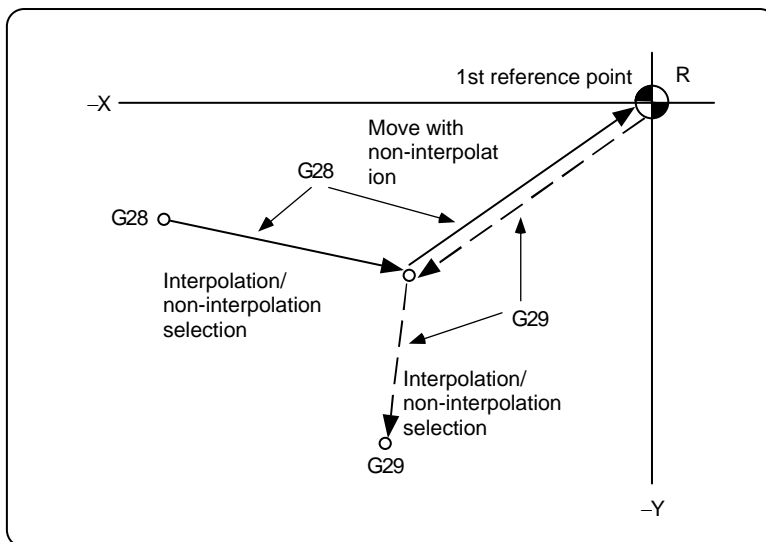
(2) Command format

The G28 program format is as follows.

<u>G28</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u> ;	After positioning the designated axis at rapid traverse to the commanded position (middle point), each axis is independently returned to the 1st reference point.
Return command	Return control axis (middle point)	

The G29 program format is as follows.

<u>G29</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u> ;	After positioning each axis independently at rapid traverse via the middle point with G28 and G30, the axis is positioned at rapid traverse to the position designated with the program.
Return command	Return control axis (Commanded position)	



12. COORDINATE SYSTEM SETTING FUNCTIONS

12.9 Automatic 1st Reference Point Return; G28/G29

(3) Detailed description

- (a) If an incremental detection method position detector is being used, the first return to 1st reference point after the NC power is turned ON uses the dog-type. However, the second and following returns can be selected from dog-type or high-speed type using the parameters. When using the absolute position detection method, the high-speed type will always be used.
- (b) The return pattern for the automatic 1st reference point return is the same as for the manual 1st reference point return.
- (c) The number of axes that can be simultaneously returned to the 1st reference point depends on the number of simultaneous control axes.
- (d) If tool radius compensation or tool nose radius compensation is not canceled during the 1st reference point return, it will be temporarily canceled during the movement to the middle point. The compensation will be reapplied at the next movement after returning.
- (e) If the tool length offset is not canceled during the 1st reference point return, the offset will be canceled during the movement from the middle point to the 1st reference point, and the offset amount will also be cleared. However, the tool length offset can be temporarily canceled with the parameters. In this case, the offset will be reapplied at the next movement command.
- (f) The movement to the G28 middle point and the movement from the G29 middle point to the commanded point can be selected as interpolation or non-interpolation with the parameters. The movement from the G28 middle point to the 1st reference point and the movement to the G29 middle point will be non-interpolation.
- (g) Even if single block is selected, the program will not stop at the middle point.
- (h) The zero point reached signal is output when the 1st reference point return is completed.

12. COORDINATE SYSTEM SETTING FUNCTIONS
12.10 2nd, 3rd and 4th Reference Point Return; G30

12.10 2nd, 3rd and 4th Reference Point Return; G30

(1) Outline

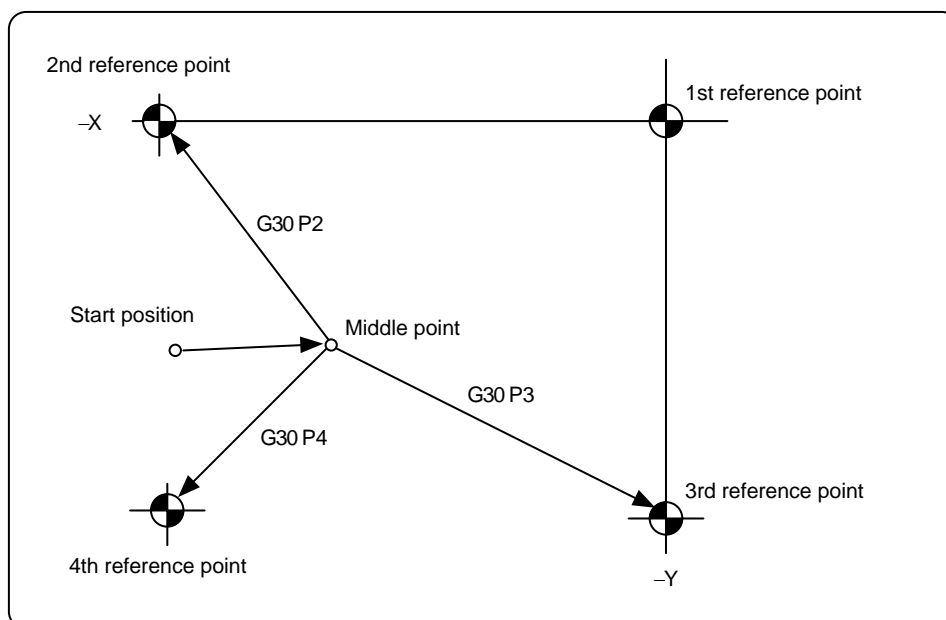
By commanding G30Pn during automatic operation, the axis can be returned to the machine's characteristic set point (2nd, 3rd, 4th reference point) in the same manner as automatic reference point return. The 2nd, 3rd, 4th reference point positions are set with the parameters.

G code	Function
G30 P2	2nd reference point return
G30 P3	3rd reference point return
G30 P4	4th reference point return

(2) Command format

The G30 program format is as follows.

<u>G30</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u>	<u>Pp1</u> ;	After positioning the designated axis to the commanded middle point with rapid traverse, each axis is independently returned to the reference point.
Return command	Return control axis (middle point)	Return point number	



12. COORDINATE SYSTEM SETTING FUNCTIONS

12.10 2nd, 3rd and 4th Reference Point Return; G30

(3) Detailed description

- (a) If the P command is omitted, P0, P1, P5 and higher will be 2nd reference point return.
- (b) The number of axes that can be simultaneously returned to the reference point depends on the number of simultaneous control axes.
- (c) If tool radius compensation is not canceled during reference point return, it will be temporarily canceled during the movement to the middle point. The compensation will be reapplied at the next movement after returning.
- (d) If the tool length compensation is not canceled during reference point return, the tool length compensation will be canceled and the offset amount will be cleared after reference point return is completed. The tool length compensation can be temporarily canceled with the parameters. In this case, the tool offset will be reapplied at the next movement command.
- (e) The movement to the middle point can be selected as interpolation or non-interpolation with the parameters. The movement from the middle point to the reference point will be non-interpolation.
- (f) Even if single block is selected, the program will not stop at the middle point.

12. COORDINATE SYSTEM SETTING FUNCTIONS

12.11 Reference Point Check; G27

12.11 Reference Point Check; G27

(1) Outline

By commanding G27, after positioning to the position issued by the program, if the positioning point is the designated reference point, the reference point reached signal will be output to the machine side in the same manner as G28. Thus, it can be checked whether the machining program created to start from the reference point and return to the reference point has correctly returned to the reference point.

(2) Command format

The G27 program format is as follows.

<u>G27</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Zz1</u>	<u>Pp1</u> ;	After the designated axis is positioned with rapid traverse to the commanded position, if the returned point is the reference point, the reference point reached signal is output. If the P command is omitted, it will be handled as 1st reference point check.
Compare command	Return control axis		Compare number		
P1	:	1st reference point check			
P2	:	2nd reference point check			
P3	:	3rd reference point check			
P4	:	4th reference point check			

(3) Detailed description

- If the P command is omitted, it will be handled as 1st reference point check.
- The number of axes that can be simultaneously reference point checked depends on the number of simultaneous control axes.
- The alarm "434" occurs if the position is not the reference point when the command is completed.
- Whether to use interpolation or non-interpolation movement can be changed with the parameters.
- The M independent command cannot be issued in the G27 command block. If simultaneously commanded, the alarm "435" will occur.

13. PROGRAMMING SUPPORT FUNCTIONS

13.1 High-speed Machining Mode; G05

(1) Outline

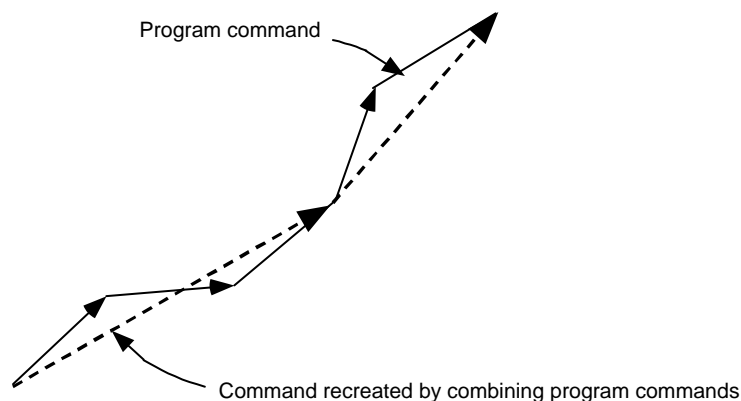
This function allows a fine block with extremely short block length in respect to the speed to be executed at a high speed for die machining, etc. The high-speed machining mode includes mode I and II according to the differences in performance (functions).

(2) Command format

G05 P1 ;	:	High-speed machining mode I ON
G05 P2 ;	:	High-speed machining mode II ON
G05 P0 ; (or G05) ;	:	High-speed machining mode OFF

(3) Detailed description

- (a) In the high-speed machining mode, the blocks are pre-read. If the length of one block is sufficiently short in respect to the commanded speed, and the block can be completed with one interpolation calculation, multiple blocks are combined and recreated into a block length that can be completed with one interpolation.



- (b) The high-speed machining mode I is commanded with G05P1, and the high-speed machining mode II is commanded with G05P2. The high-speed machining mode is canceled with G05P0. The incremental value/absolute value commands in the high-speed machining mode follow the modals for when G05 is commanded.

13. PROGRAMMING SUPPORT FUNCTIONS

13.1 High-speed Machining Mode; G05

(c)

Mode	Command	M6
Standard mode	G05 (P0)	8.4m/min (Regardless of tool radius compensation ON/OFF)
High-speed mode I	G05P1	16.8m/min (Regardless of tool radius compensation ON/OFF)
High-speed mode II	G05P2	67m/min (When tool radius compensation is OFF.)

The numbers in the above table indicate the maximum speed (m/min) when the G1 block for 1mm segment is executed.

The performance conditions are common and are as follow:

- 4-axis system (including spindle) or less
- 1 system
- Three axes or less simultaneously commanded with G01
- Block containing only axis name and movement amount (macros and variable commands are not contained)
- Only stored stroke limit I is valid for stroke check

For high-speed mode II, the following condition applies in addition to the above.

- Tool radius compensation OFF

There are no particular limits to the program for the high-speed machining mode I. However, if the above conditions are not satisfied, the designated speed may not be ensured.

- (d) Override, maximum cutting speed clamp, single block operation, dry run, manual pulse generator interrupt, graphic trace and high-accuracy control mode are all valid even in the high-speed machining mode.

(4) Restrictions

- (a) Before commanding G05P2, cancel the radius compensation, mirror image, scaling, coordinate rotation, hypothetical axis interpolation and three-dimensional radius compensation.
- (b) The automatic operation process has the priority during the high-speed machining mode, so the display response may be delayed.
- (c) During the G05P2, G05P0 and G05P1 command blocks, the speed will decelerate once, so command these at a position where the tool is separated from the workpiece.
- (d) When using the high-speed machining mode with communication or tape operation, the machining speed may be suppressed to a low speed due to restrictions in the program transfer speed.
- (e) The cutting feed clamp speed during the high-speed machining mode is the minimum speed of the movement axes' cutting feed clamp speeds. The cutting feed clamp speed when not in the high-speed machining mode is set so that the feedrate of each movement axis does not exceed the respective cutting feed clamp speed. Thus, the cutting feed clamp speed during the high-speed machining mode may decrease compared to when not in the high-speed machining mode.

(5) Notes

- (a) The only address P commands that are valid are P0, P1 and P2. A program error will occur if other P commands are designated, or if there is no P command.
- (b) Do not command addresses other than N or P in the G05 block. A program error will occur if an address other than N or P is commanded.
- (c) Decimal points are invalid for address P. Even if a decimal point is used, it will be ignored.

13. PROGRAMMING SUPPORT FUNCTIONS
13.2 Subprogram Control; G22/G23

13.2 Subprogram Control; G22/G23

(1) Outline

When the same pattern is used repeatedly during machining, this function registers that machining pattern as a subprogram. This allows the same machining to be realized easily by calling out the subprogram from the main program when required. The program can be effectively used with this function.

The call is designated with the program number and sequence number. The main program is returned to from the subprogram by commanding G23. Other subprograms can be called from the subprogram, and a depth of up to eight levels can be programmed.

(2) Command format

(a) Calling the subprogram

<u>G22</u>	<u>Ll1</u>	<u>Hh1</u>	<u>Pp1</u>	<u>.Dd1 ;</u>	Program number address L
(G22	Aa1	Hh1	Pp1	.Dd1 ;	Program number address O)
Call command	Subprogram number (Branch to subprogram)	Sequence number	Number of repetitions	Subprogram device No. (Note)	
	Ll1 (Subprogram)				
	(Oo1) :				
	Nh1				
	:				
	G23 ; (Return to main program)				

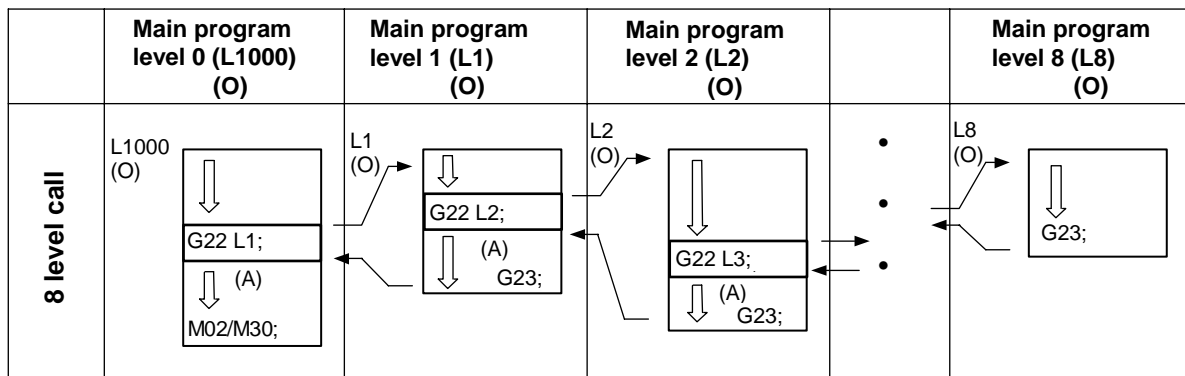
(Note) Set the device No. in which a subprogram exists. The call sequence designated in the machining parameters is followed if this setting is omitted.
 The d1 device No. is set in the machining parameters.

(b) Returning from the subprogram

<u>G23</u>	<u>Hh2 ;</u>
Return command	Return destination sequence number (When omitted, the program will return to the block following the call block.)

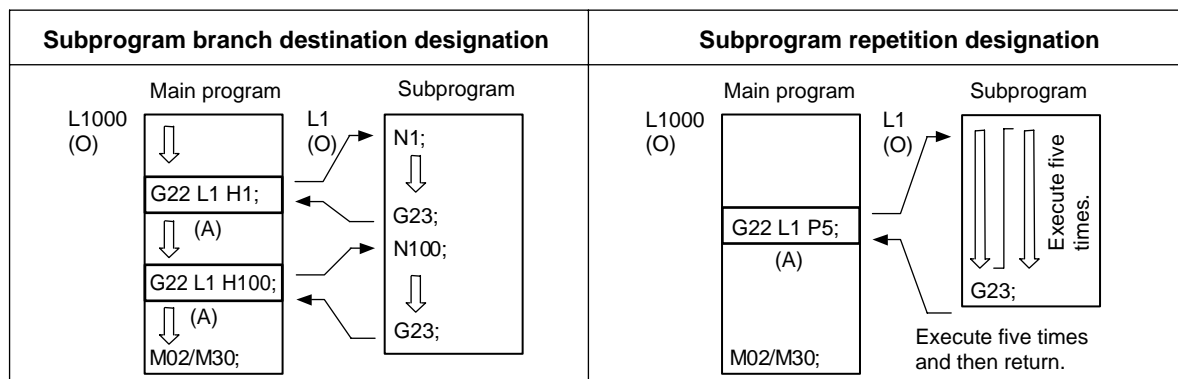
(3) Detailed description

- (a) The subprogram uses the same format as the machining program for normal memory operation. The only point that differs is that the subprogram end command G23 (H_) is inserted as an independent block at the last block.
- (b) The subprogram numbers 1 to 99999999 can be used.
- (c) The subprogram call level can be used up to eight levels. The program error "PR0230" will occur if this is exceeded.



13. PROGRAMMING SUPPORT FUNCTIONS
13.2 Subprogram Control; G22/G23

(d) The subprogram branch destination and the subprogram repetition can also be designated.



(e) In addition to subprogram stored in the NC memory, subprograms stored in the personal computer's floppy disk or hard disk, etc., can also be run.

The relation of the main program storage place and subprogram storage place is as follows.

		Subprogram storage place	NC side		Personal computer side			
			Memory	IC card	Floppy disk	Hard disk/ data server	Tape RS-232C	
Main program storage place	NC side	Memory	○	○	○	○	○	×
	IC card	○	○	○	○	○	○	×
	MDI	○	○	○	○	○	○	×
Personal computer side	Floppy disk	○	○	○	○	○	○	×
	Hard disk/ data server	○	○	○	○	○	○	×
	Tape RS-232C	○	○	○	○	○	○	×

The ○ mark indicates a combination that can be called.

(4) Notes

- (a) The program error "232" will occur if the designated L(A) (program number) is not found.
- (b) The G22L(A); G23; block will branch to "L(A)100" after "X100." is executed if the program is "X100.G22L(A)100;".
- (c) If G23; is commanded in the main program, the program will return to the head. (This also applies for MDI.)
 Note that a program error will occur if the parameter (Main M99 alarm stop) is ON.
- (d) The subprogram can be branched to with G22L(A); using tape operation, but a sequence number cannot be designated as the return destination with G23H_;. (H_ will be ignored.)
- (e) If the sequence number is designated with G23H_;, it may take longer to search for the designated position depending on the program.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

13.3 Fixed Cycle; G80 to G89/G73/G74/G76

13.3.1 Outline

Using this function, hole drilling cycles such as drill, tapping, etc., can be commanded with a program containing one block. By designating the operation mode, various fixed cycle can be called out. By editing the standard fixed cycle subprogram, the fixed cycle sequence can be changed by the user. The user can also register and edit an original fixed cycle program.

List of machining center system hole drilling fixed cycle functions

G code	Hole drilling start (-Z direction)	Operation at hole bottom		Return operation (+Z direction)	Application
		Dwell	Spindle		
G80	—	—	—	—	Cancel
G81	Cutting feed	—	—	Rapid traverse	Drilling, spot drilling cycle
G82	Cutting feed	Valid	—	Rapid traverse	Drilling, counter boring cycle
G83	Intermittent feed	—	—	Rapid traverse	Deep hole drilling cycle
G84	Cutting feed	Valid	Reverse run	Cutting feed	Tapping cycle
G85	Cutting feed	—	—	Cutting feed	Boring cycle
G86	Cutting feed	Valid	Stop	Rapid traverse	Boring cycle
G87	Rapid traverse	—	Forward run	Cutting feed	Back boring cycle
G88	Cutting feed	Valid	Stop	Rapid traverse	Boring cycle
G89	Cutting feed	Valid	—	Cutting feed	Boring cycle
G73	Intermittent feed	Valid	—	Rapid traverse	Step cycle
G74	Cutting feed	Valid	Forward run	Cutting feed	Reverse tapping cycle
G76	Cutting feed	—	Oriented spindle stop	Rapid traverse	Fine boring cycle

(Note) G87/G74 can be used as the manual synchronous tapping function. Refer to the MELDAS 600M Series Instruction Manual (BNP-B2237) "CHAPTER 2 MACHINE OPERATIONS" for details on the operation.

13.3.2 Command format

The basic program format of the fixed cycle command is as follows.

<u>G98</u>	<u>G81</u>	<u>Xx1</u> <u>Yy1</u>	<u>Zz1</u>	<u>Rr1</u>	<u>Qq1</u>	<u>Pp1</u>	<u>Ll1</u>	<u>Ff1</u> ;
							(<u>Ee1</u>) (<u>Hh1</u>)	
Return point	Operation mode	Hole position data	Hole machining data			Feedrate		

(a) Return point

The following two hole drilling axis return levels can be used after the fixed cycle machining operation is completed.

G code	Function
G98	Initial point level return
G99	R point level return

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(b) Operation mode

G code	Application
G80	Cancel
G81	Drilling, spot drilling cycle
G82	Drilling, counter boring cycle
G83	Deep hole drilling cycle
G84	Tapping cycle
G85	Boring cycle
G86	Boring cycle
G87	Back boring cycle
G88	Boring cycle
G89	Boring cycle
G73	Step cycle
G74	Reverse tapping cycle
G76	Fine boring cycle

(c) Hole position data

X, Y X axis, Y axis hole drilling position command (rapid traverse) (Incremental/absolute)

(Note) When there is no drilling position command, the drilling will not be carried out though the drilling data is saved.

- There are x and y commands : drilling will be carried out.
- There are only z and r commands : data will be saved only (No drilling)

(d) Hole machining data

Address		Details
M2 label O = 0	M2 label O = 1	
Z	Z	Hole bottom position designation (Incremental/absolute)
R	R	Hole R position designation (Incremental/absolute)
Q	Q	Cut-in amount per rotation at G73, G83 (Incremental) Shift amount at G76, G87
P(E)	E	Dwell time at hole bottom
L(H)	P	Number of fixed cycle repetitions designation
F	F	Cutting feedrate

(Note 1) The number of repetitions can be designated as follows according to whether label L or label O is used.

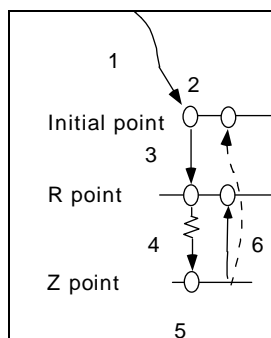
	Address L	Address H	Address P	Address K
Label L	○	○	×	○
Label O	×	×	○	○

(Note 2) Parameter (110134 valid ADR_K FIX) must be set to validate the address K command.

(Note 3) When using label L, if addresses L, H and K are designated in the same block, the order of priority will be address H, L and then K.

(Note 4) When using label O, if addresses P and K are designated in the same block, address P will have the priority.

13.3.3 Description of operation

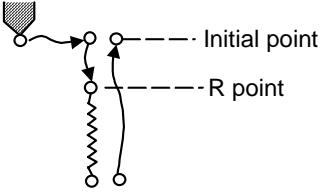
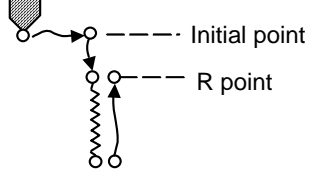
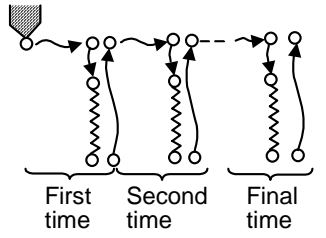
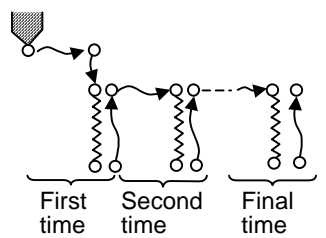


- Operation 1: This denotes the X and Y axis positioning, and positioning is done by G00. By setting the parameters, the axis can be moved according to the current O1 group's G code.
- Operation 2: This action takes place at the positioned (initial) point. When the G87 command is assigned, the M19 command is output from the NC unit to the machine. This M command is executed and when the finish (FIN) signal arrives at the NC unit, the next action starts. If the single block stop switch is in the ON position, the block stop mode is established upon completion of the positioning.
- Operation 3: The tool is positioned as far as the R point by rapid traverse.
- Operation 4: Hole machining is conducted by cutting feed.
- Operation 5: This action takes place at the hole position. It differs according to the fixed cycle mode. Possible actions include spindle stop (M05), spindle reverse rotation (M04), spindle forward rotation (M03), dwell and tool shift.
- Operation 6: The tool is retracted as far as the R point. Depending on the fixed cycle mode, it will be retracted at the cutting feed or rapid traverse rate.
- Operation 7: The tool is returned as far as the initial point at the rapid traverse rate.

Whether operation 6 or 7 is to be regarded as the completion of the fixed cycle can be selected by the G commands (G98/G99).

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

The relation of the G98/G99 mode and No. of repetition designation is as shown below.

No. of hole drilling	Program example	G98 At power ON, at cancel with M02, M30, and reset button	G99
Only one execution	G81X100. Y100. Z-50. R25. F1000;	 <p>Initial level return is executed.</p>	 <p>R point level return is executed.</p>
Second and following executions	G81X100. Y100. Z-50. R25. L5F1000;	 <p>Initial level return is executed for all times.</p>	

- (a) With the label L, when the number of repetition is two or more times, the axis will return to the point determined by G98/G99 modal at the last time.
- (b) With the label O, when the number of repetition is two or more times, the axis will return to the initial point regardless of G98/G99 mode at the last time.

13.3.4 Detailed description

(1) Positioning planes and hole drilling axis

The basic control elements of the positioning plane and hole drilling axis are contained in the fixed cycles. The positioning plane is determined by the G17, G18 or G19 plane selection command, and the hole drilling axis is the axis that is perpendicular to the above selected plane.

Plane selection	Positioning plane	Hole drilling axis
G17 (X-Y)	Xp-Yp	Zp
G18 (Z-X)	Zp-Xp	Yp
G19 (Y-Z)	Yp-Zp	Xp

(Note) Xp, Yp and Zp denote the basic axes X, Y and Z or the parallel axes of these basic axes.

- (c) Any axis except the hole drilling axis can be assigned for the positioning.
- (d) The hole drilling axis is determined by the axis address of the hole drilling axis commanded in the same block as the fixed cycle command (G81 to G89, G73, G74, G76). Hole drilling cannot be carried out if there is no axis address command.

(Example) When G17 (X-Y plane) has been selected and the W axis has been set as the axis parallel to the Z axis

```
G81 ..... Z_ ;    Z axis is set as the hole drilling axis.
G81 ..... W_ ;    W axis is set as the hole drilling axis.
G81 ..... ;       Z axis is set as the hole drilling axis.
```

(Note 1) The Z axis can be fixed as the hole drilling axis by parameter setting.

(Note 2) The hole drilling axis should be selected in the fixed cycle cancel mode.

In this explanation, the command value is incremental, the positioning plane is XY, and the hole drilling axis is the Z axis.

(2) Fixed cycles and uni-directional positioning (G60)

Using fixed cycles and the uni-directional positioning (G60) command in combination enables a tool to be positioned with a high accuracy at the hole drilling point without any backlash error.

(3) Workpiece coordinate settings in fixed cycle mode

The axis-designated axis moves with the set workpiece coordinate system.

The Z axis is valid from the R point positioning or Z axis movement after positioning is completed.

(Note) For addresses Z and R, reprogram when changing the workpiece coordinates even if the values are the same.

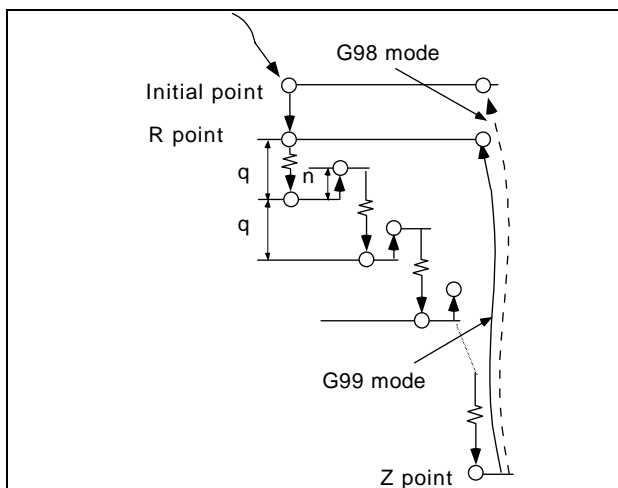
(Example)

```
G54 Xx1 Yy1 Zz1 ;
G81 Xx2 Yy2 Zz2 Rr2 ;
      :
      :
G55 Xx3 Yy3 Zz2 Rr2 ; ... Command again even if the Z and R values are the
      :                               same as the previous command.
      Xx4 Yy4 ;
      Xx5 Yy5 ;
      :
      :
```

13.3.5 Description of each command

(1) G73 (Step cycle)

G73 Xx Yy Zz Qq Rr Ff Pp (Ee);



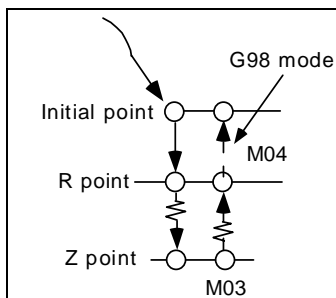
- 1) G0 Xx Yy;
- 2) G0 Zr;
- 3) G1 Zq Ff;
- 4) G4 Pp;
- 5) G0 Z (z-n);
- 6) G1 Z (q+n) Ff;
- ⋮
- ⋮
- (n) G98 mode G0 Z - (z+r);
G99 mode G0 Z - z;

- Using the G73 command, the tool can be returned by rapid traverse by an amount equivalent to "n" mm and then switched to cutting feed for the second and subsequent passes.
- The return amount "n" is set with the parameters.
- The stop position during single block operation applies upon completion of the 1), 2) and (n) commands.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(2) G74 (Reverse tapping cycle)

G74 Xx Yy Zz Qq Rr Ff Pp (Ee) ;



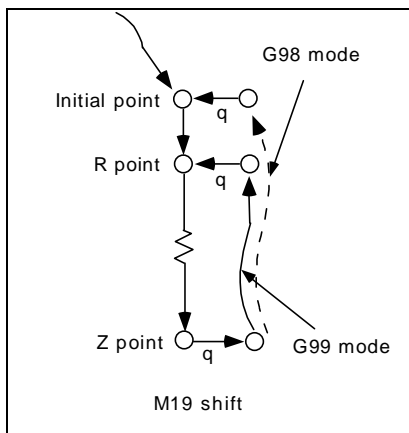
- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) G4 Pp ;
- 5) M03 ; (Spindle forward rotation)
- 6) G1 Z - z Ff ;
- 7) G4 Pp ;
- 8) M04 ; (Spindle reverse rotation)
- 9) G98 mode G0 Z - r ;
G99 mode No movement

- Override is canceled while G74 is being executed and the override is automatically set to 100%.
- Dry run is also ignored.
- When the feed hold button is pressed while G74 is being executed, the tool will not stop immediately with sequences 3) to 6) but will stop upon completion of 6). It will stop immediately with sequence 1), 2) and 9) rapid traverse.
- The stop position during single block operation applies upon completion of the 1), 2) and 9) commands.
- The "in-tapping" NC output signal is output while G74 is modal.
- The M03, M04, M05 and S codes are not output while the G74 synchronous tapping is modal.
- If F/E are not commanded in the G84, G74 command block, the program error "PR0183 No pitch/thread number" will occur. Note that the previous F/E values will be used if the parameter (#320024 Synchronized tapping) is turned ON.
In this case, the F/E values will be initialized when the NC is reset.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(3) G76 (Fine boring)

G76 Xx Yy Zz Rr Iq1 Jq2 Ff ;



- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) M19 ; (Spindle orientation)
- 5) G1 Xq1 Yq2 Ff ; (Shift)
- 6) G98 mode G0 Z - (z + r) ;
G99 mode G0 Z - z ;
- 7) G0 X - q1 Y - q2 ;
- 8) M03 ; (Spindle forward rotation)

- The stop position during single block operation applies upon completion of the 1), 2) and 7) commands.
- This command can be used to perform high-precision boring without leaving scratches or marks on the machining surface.
- This is escaped (retracted) after cutting in the shifted state in the reverse direction to the tool nose.
- The shift amount is assigned as follows by addresses I, J and K.
 - I and J with G17
 - K and I with G18
 - J and K with G19

(Note) Even if address K for designating the number of fixed cycle repetitions is valid, address K will designate the shift amount rather than the number of repetitions.

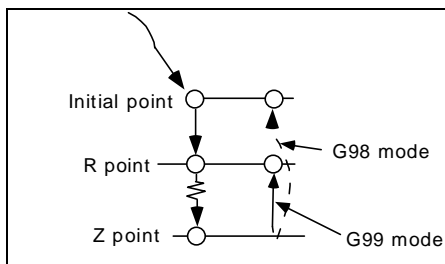
- The shift amount is executed by linear interpolation and the feedrate accords with the F command.
- I, J and K should be assigned with incremental values in the same block as the hole position data.
- I, J and K are treated as modals during fixed cycles.

(Note) When the parameter fixing the hole drilling axis as the Z axis has been set, the shift amount can also be assigned by address Q and not by I and J. In a case like this, the shift amount is set by parameter to +X, -X, +Y or -Y. The sign of the Q value is ignored and handled as plus. Care should be taken because the value of Q is modal in a fixed cycle and considered as the cutting amount of subsequent G83, G87 and G73 commands.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(4) G81 (Drilling, spot drilling)

G81 Xx Yy Zz Rr Ff ;

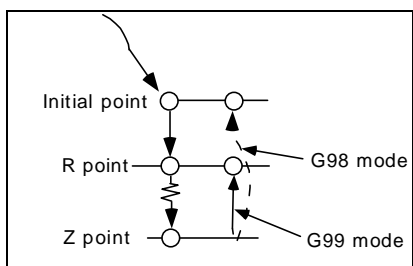


- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) G98 mode G0 Z - (z + r) ;
 G99 mode G0 Z - z ;

- The stop position during single block operation applies upon completion of the 1), 2) and 4) commands.

(5) G82 (Drilling, counterboring)

G82 Xx Yy Zz Rr Ff Pp (Ee) ;



- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) G4 Pp ; (Dwell)
- 5) G98 mode G0 Z - (z + r) ;
 G99 mode G0 Z - z ;

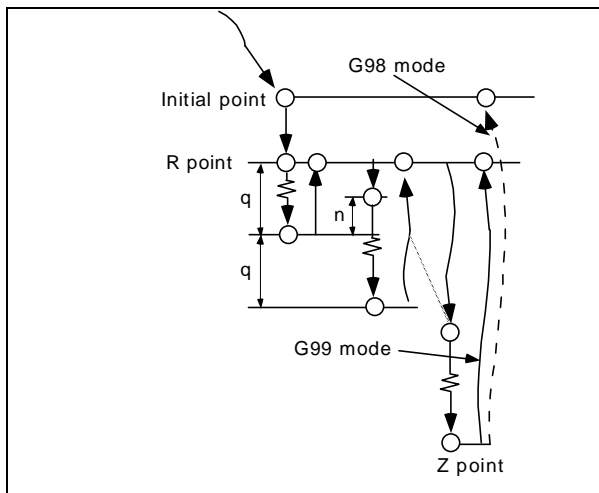
- The stop position during single block operation applies upon completion of the 1), 2) and 5) commands.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(6) G83 (Deep hole drilling cycle)

G83 Xx Yy Zz Rr Qq Ff ;

Q: Where Q is the cut amount for each pass, which is always assigned as an incremental value.



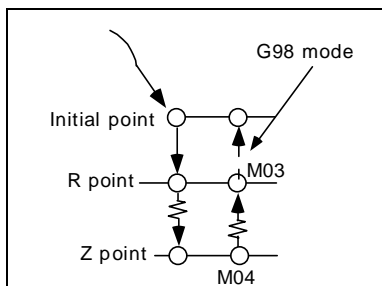
- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zq Ff ;
- 4) G0 Z - q ;
- 5) G0 Z (q - n) ;
- 6) G1 Z (q + n) Ff ;
- 7) G0 Z - 2 * q ;
- 8) G0 Z (2 * q - n) ;
- 9) G1 Z (q + n) Ff ;
- 10) G0 Z - 3 * q ;
- ...
- ...
- (n) G98 mode G0 Z - (z + r) ;
- G99 mode G0 Z - z ;

- Using the G83 command, the tool can be switched from rapid traverse to cutting feed in front of "n" mm from the position machined immediately before for the second and subsequent passes.
- "n" is set with the parameters.
- Program so that $q > n$.
- The stop position during single block operation applies upon completion of the 1), 2) and (n) commands.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(7) G84 (Tapping cycle)

G84 Xx Yy Zz Rr Ff Pp (Ee) ;



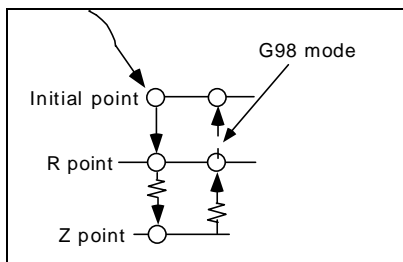
- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) G4 Pp ;
- 5) M04 ; (Spindle reverse rotation)
- 6) G1 Z - z Ff ;
- 7) G4 Pp ;
- 8) M03 ; (Spindle forward rotation)
- 9) G98 mode G0 Z - r ;
 G99 mode No movement

- Whether to use the synchronous tapping mode or asynchronous tapping mode follows the parameter settings.
- Override is canceled while G84 is being executed and the override is automatically set to 100%.
- Dry run is also ignored.
- When the feed hold button is pressed while G84 is being executed, the tool will not stop with sequences 3) to 6) but will stop immediately upon completion of 6). It will stop immediately with sequence 1), 2) and 9) rapid traverse.
- The stop position during single block operation applies upon completion of the 1), 2) and 9) commands.
- The "in-tapping" NC output signal is output while G84 is modal.
- The M03, M04, M05 and S codes are not output in the G84 synchronous tapping modal.
- If F/E are not commanded in the G84, G74 command block, the program error "PR0183 No pitch/thread number" will occur. Note that the previous F/E values will be used if the parameter (#320024 Synchronized tapping) is turned ON.
 In this case, the F/E values will be initialized when the NC is reset.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(8) G85 (Boring)

G85 Xx Yy Zz Rr Ff ;

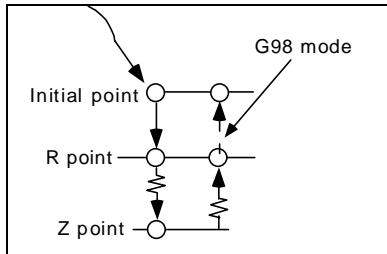


- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) G1 Z-z Ff ;
- 5) G98 mode G0 Z-r ;
G99 mode No movement

- The stop position during single block operation applies upon completion of the 1), 2), 4) or 5) commands.

(9) G86 (Boring)

G86 Xx Yy Zz Rr Ff Pp (Ee) ;



- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) G4 Pp
- 5) M05 ; (Spindle stop)
- 6) G98 mode G0 Z-(z+r) ;
G99 mode G0 Z-z ; S
- 7) M03 ; (Spindle forward rotation)

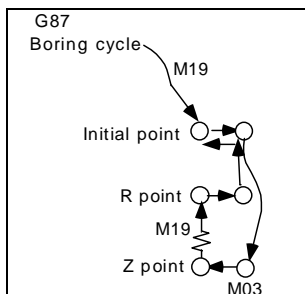
- The stop position during single block operation applies upon completion of the 1), 2) and 7) commands.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(10) G87 (Back boring)

G87 Xx Yy Zz Rr Iq1 Iq2 Ff ;

(Note) Take care with the z and r assignment. (The z and r signs are reversed.)
 There is no R point return.



- 1) G0 Xx Yy ;
- 2) M19 ; (Spindle orientation)
- 3) G0 Xq1 Yq2 ; (Shift)
- 4) VG0 Zr ;
- 5) G1 X - q1 Y - q2 Ff ; (Shift)
- 6) M03 ; (Spindle forward rotation)
- 7) G1 Zz Ff ;
- 8) M19 ; (Spindle orientation)
- 9) G0 Xq1 Yq2 ; (Shift)
- 10) G98 mode G0 Z - (z + r) ;
 G99 mode G0 Z - z ;
- 11) G0 X - q1 Y - q2 ; (Shift)
- 12) M03 ; (Spindle forward rotation)

- The stop position during single block operation applies upon completion of the 1), 4), 6) and 11) commands.
- This command can be used to perform high-precision boring without leaving scratches or marks on the machining surface.
- This is escaped (retracted) after cutting in the shifted state in the reverse direction to the tool nose.
- The shift amount is assigned as follows by addresses I, J and K.
 - I and J with G17
 - K and I with G18
 - J and K with G19

(Note) Even if address K for designating the number of fixed cycle repetitions is valid, address K will designate the shift amount rather than the number of repetitions.

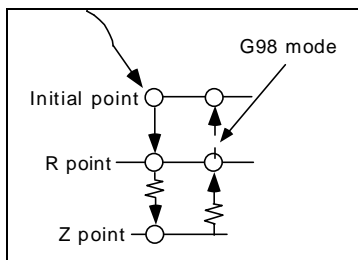
- The shift amount is executed by linear interpolation and the feedrate accords with the F command.
- I, J and K should be assigned with incremental values in the same block as the hole position data.
- I, J and K are treated as modals during fixed cycles.

(Note) When the parameter fixing the hole drilling axis as the Z axis has been set, the shift amount can also be assigned by address Q and not by I and J. In a case like this, the shift amount is set by parameter to +X, -X, +Y or -Y.
 The sign of the Q value is ignored and handled as plus.
 Care should be taken because the value of Q is modal in a fixed cycle and considered as the cutting amount of subsequent G83, G87 and G73 commands.

13. PROGRAMMING SUPPORT FUNCTIONS
13.3 Fixed Cycle; G80 to G89/G73/G74/G76

(11) G88 (Boring)

G88 Xx Yy Zz Rr Ff Pp (Ee);

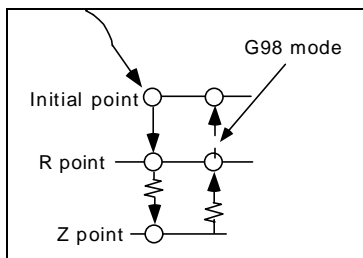


- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) G4 Pp
- 5) M05 ; (Spindle stop)
- 6) Stop with single block stop switch ON
- 7) G98 mode G0 Z – (z + r) ;
G99 mode G0 Z – z ;
- 8) M03 ; (Spindle forward rotation)

- The stop position during single block operation applies upon completion of the 1), 2) 6) and 8) commands.

(12) G89 (Boring)

G89 Xx Yy Zz Rr Ff Pp (Ee);



- 1) G0 Xx Yy ;
- 2) G0 Zr ;
- 3) G1 Zz Ff ;
- 4) G4 Pp
- 5) G1 Z – z ;
- 6) G98 mode G0 Z – r ;
G99 mode No movement

- The stop position during single block operation applies upon completion of the 1), 2), 5) or 6) commands.

13.4 Corner Chamfering, Corner R; ,C/,R

13.4.1 Outline

Chamfering or corner rounding is automatically inserted between blocks by adding ",C " or ",R " to the linear-arc program.

13.4.2 Corner chamfering

(1) Function

The corner is chamfered in such a way that the points, which are distanced by the lengths assigned by ",C_" from the hypothetical start and final corners which would apply were to be performed, are connected.

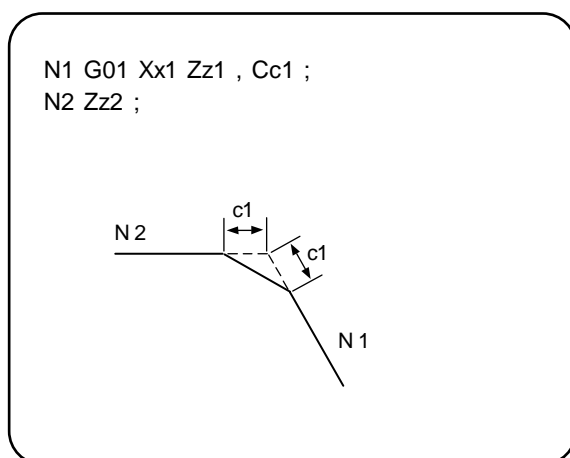
(2) Command format

N100 G01 X_ Z_ ,C_;	Length up to chamfering starting point or end point from hypothetical corner
N200 G01 X_ Z_;	Next block's end point coordinates

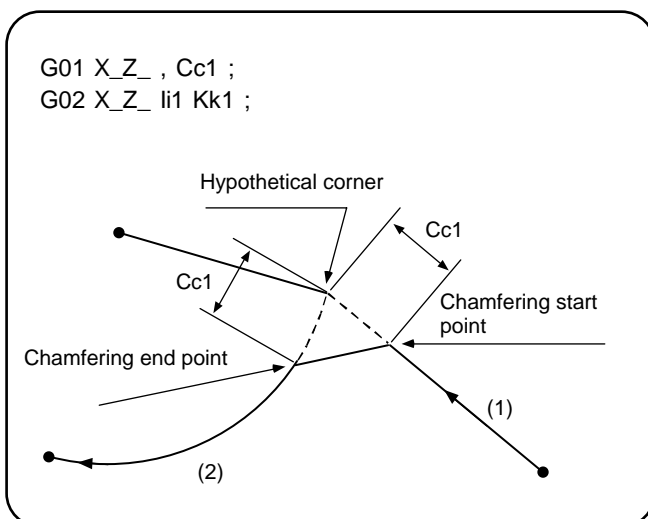
Chamfering is performed at the point where N100 and N200 intersection.

(3) Description of operation

(a) Linear – Linear

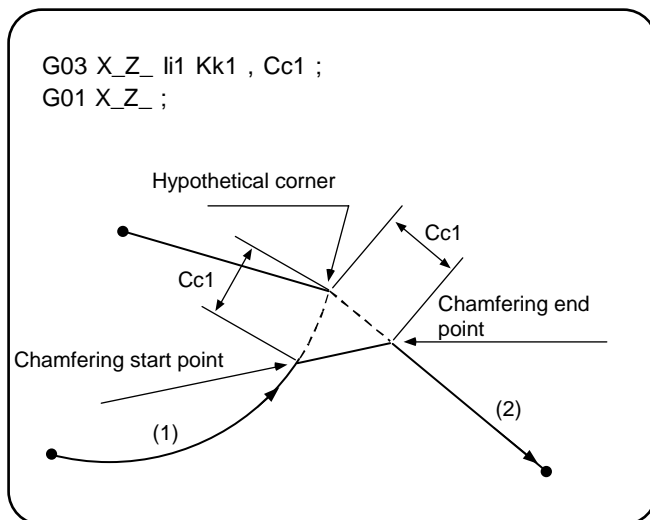


(b) Linear – Arc

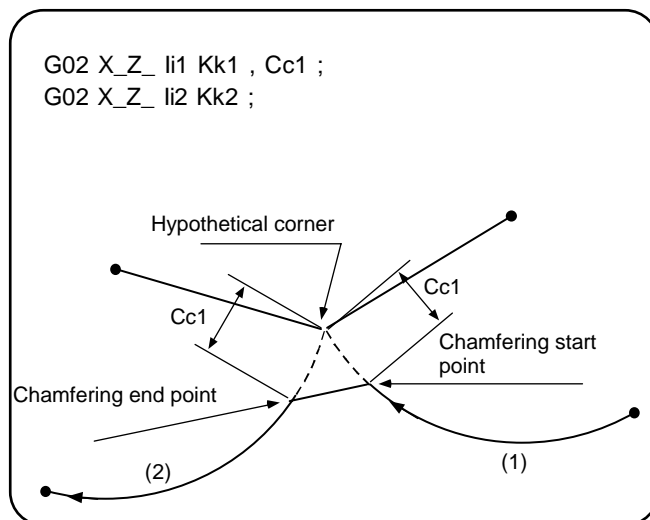


13. PROGRAMMING SUPPORT FUNCTIONS
13.4 Corner Chamfering, Corner R; ,C/,R

(c) Arc – Linear



(d) Arc – Arc



(4) Detailed description

- (a) The start point of the block following the corner chamfering serves as the hypothetical corner.
- (b) When the comma in ,C is not present, the command is handled as a C command.
- (c) When both corner R command ",R" are assigned in the same block the latter command is valid.
- (d) Tool offset is calculated for the shape which has already been subjected to corner chamfering.

(5) Notes

- (a) A program error will occur if the block commanding corner chamfering or the block following it is a positioning command or thread cutting command.
- (b) A program error will occur if the block following that commanding corner chamfering is not a group 01 G command.
- (c) A program error will occur when the movement amount in the block assigning corner chamfering is less than the chamfering amount.
- (d) A program error will occur when the movement amount in the block following the block commanded corner chamfering is less than the chamfering amount.

13. PROGRAMMING SUPPORT FUNCTIONS
13.4 Corner Chamfering, Corner R; ,C/,R

13.4.3 Corner R

(1) Function

The hypothetical corner that assumes that there is no rounding, is rounded with the radius arc designated with ",R".

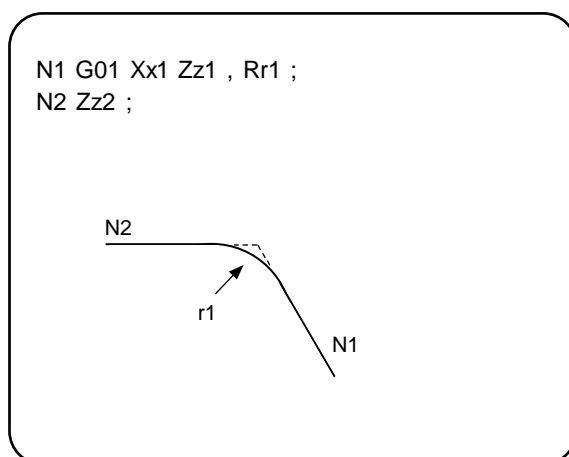
(2) Command format

<pre>N100 G01 X_ Z_ ,R_;</pre>
\uparrow Arc radius for corner rounding
<pre>N200 G01 X_ Z_;</pre>

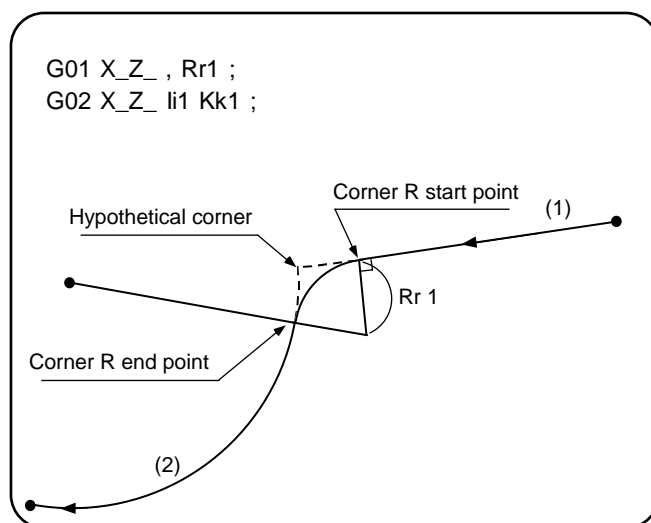
Corner R is performed at the point where N100 and N200 hypothetical corner.

(3) Description of operation

(a) Linear – Linear

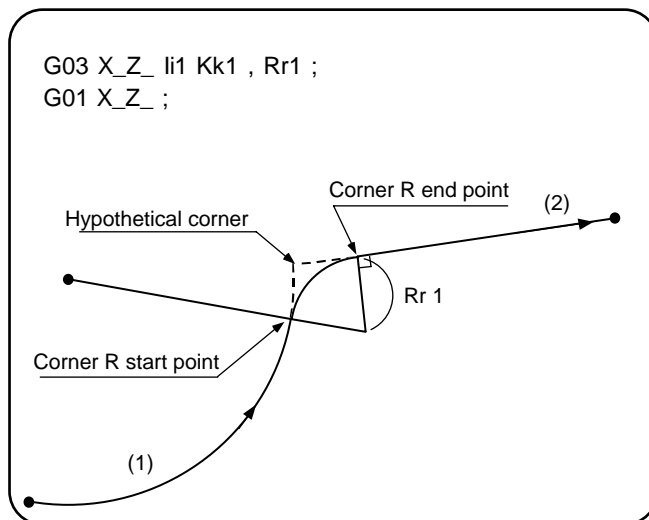


(b) Linear – Arc

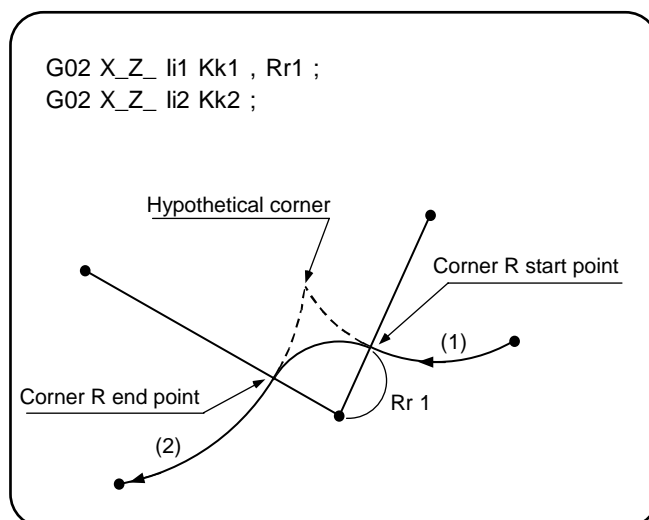


13. PROGRAMMING SUPPORT FUNCTIONS
13.4 Corner Chamfering, Corner R; ,C/R

(c) Arc – Linear



(d) Arc – Arc



(4) Detailed description

- (a) The start point of the block following the corner R serves as the hypothetical corner.
- (b) When the comma in ,R is not present, the command is treated as an R command.
- (c) If the corner chamfering command ",C" is issued in the same block, the latter command will be valid.
- (d) Tool offset is calculated for the shape which has already been subjected to corner R.

(5) Notes

- (a) A program error will occur if the block commanding corner R or the block following it is a positioning command or thread cutting command.
- (b) A program error will occur if the block following that commanding corner R is not a group 01 G command.
- (c) A program error will occur when the movement amount in the block commanded corner R is less than the R value.
- (d) A program error will occur when the movement amount in the block following the block corner R is less than the R value.

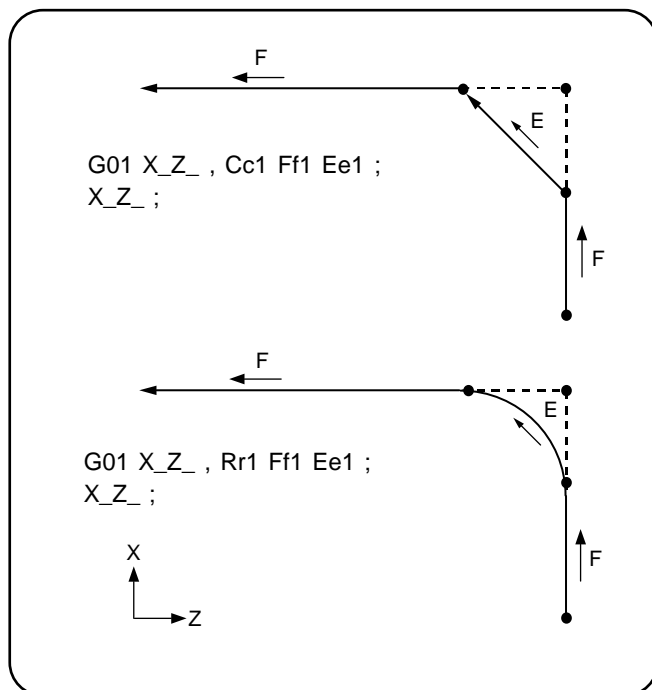
13. PROGRAMMING SUPPORT FUNCTIONS

13.4 Corner Chamfering, Corner R; ,C/,R

13.4.4 Corner chamfering/corner R speed E designation

The feedrate for the corner chamfering and corner R areas can be commanded with the E command. This allows the correct shape to be cut at the corner area.

(Example)



- The E command is a modal. It is valid even for the next corner chamfer and corner R areas.
- The E command modal has independent asynchronous feedrate modals and synchronous feedrate modals. The valid speed follows the asynchronous/synchronous mode (G94/G95).
- If the E command is 0 or if E has not been designated yet, the feedrate at the corner chamfering and corner R area will be the F command feedrate.
- The E command modal is not cleared even when the reset button is pressed. It is cleared when the power is turned OFF. (This is the same as the F command.)

13.5 Mirror Image; G50.1/G51.1/G62

13.5.1 Outline

When cutting a left/right symmetrical shape, the other shape can be machined with just the left or the right program. The programming time can be shortened by using this mirror image function.

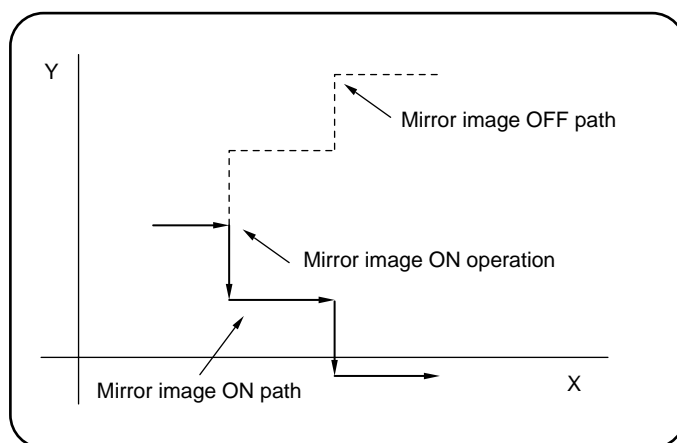
The mirror image command methods include the following:

- External switch mirror image input from PLC
- Parameter set mirror image
- G command mirror image

13.5.2 External switch mirror image and parameter set mirror image

(1) Detailed description

By reversing the signs of the command values for program operation, symmetrical cutting can be carried out.



- (a) Mirror image can be applied on all control axes.
- (b) The mirror image is changed after block stop.
- (c) The mirror image ON/OFF control can be changed with the following operations.
 - 1) External input signal ON/OFF changeover using the switch on the machine operation panel via the PLC.
 - 2) Setting and display unit . . . ON/OFF changeover using parameter operations.
- (d) The mirror center becomes the position where the mirror image was turned ON.

(2) Notes

- (a) When turning the mirror image parameter OFF during execution of a machining program, make sure to turn it OFF at the same coordinate position as where it was turned ON. If the changeover position changes, always execute reference point return and G92G53X_Y_Z_α_;

13. PROGRAMMING SUPPORT FUNCTIONS
13.5 Mirror Image; G50.1/G51.1/G62

13.5.3 G command mirror image

(1) Outline

The mirror image can be directly designated with the G command by creating a machining program.

G code	Function
G50.1	G command mirror image cancel
G51.1	G command mirror image ON

or

G code	Function
G62	G command mirror image ON/cancel

(2) Command format

The G command mirror image program format is as follows.

<u>G51.1</u> Mirror image ON	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u> ; Command axis and command position	The mirror image is applied on the (x1, y1, z1) as if a mirror is placed.
------------------------------------	--	---

or

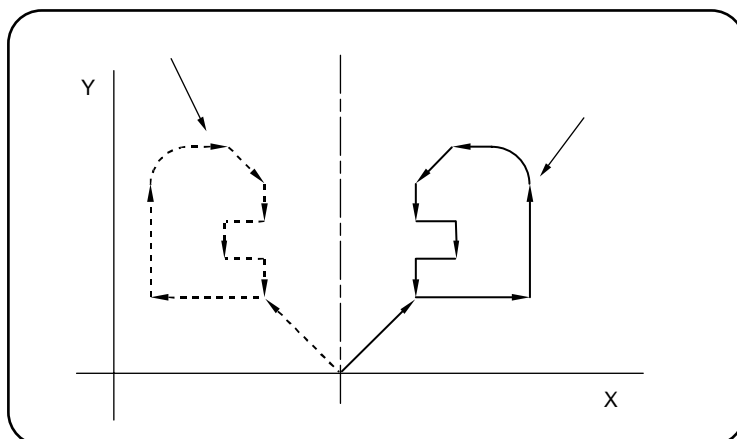
<u>G62</u> Mirror image ON	X1 Y1 Z1 ;	The mirror image is applied on the all axes specified in the same block as G62. The mirror center is the position where the G62 command is issued.
----------------------------------	------------	--

The G command mirror image cancel program format is as follows.

<u>G50.1</u> Mirror image cancel	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u> ; Command axis	The coordinate word indicates the axis for which mirror image is turned OFF. The coordinate value is ignored.
--	--	---

or

<u>G62</u> Mirror image cancel	X0 Y0 Z0 ; Mirror axis	The mirror image on the all axes specified in the same block as G62 is canceled.
--------------------------------------	---------------------------	--



(3) Detailed description

- (a) The coordinate word in G51.1 is assigned for the mirror image command axes while the coordinate values are assigned for the mirror image center coordinates in either absolute or incremental values.
- (b) The coordinate word in G50.1 indicates the axes for which mirror image will not be applied and the coordinate values are ignored.

13. PROGRAMMING SUPPORT FUNCTIONS

13.5 Mirror Image; G50.1/G51.1/G62

- (c) With the G62 command, the mirror image ON/OFF is applied on the all axes specified in the same block as G62. The mirror center is the position where the G62 command is issued.
- (d) When mirror image is applied to only one axis in the assigned plane, the rotation direction or offset direction is reversed for arc or tool radius compensation and coordinate rotation.
- (e) These functions are processed on the local coordinate system and so the mirror image center will be moved by counter presetting or by any change in the workpiece coordinates.
- (f) When a zero point return command (G28 or G30) has been executed during mirror image, the mirror image is valid for operations up to the middle point but not for operations from the middle point to the zero point.
- (g) When a return command (G29) is issued from the zero point during mirror image, the axis passes through the middle point passed through with the previous zero point return command, unrelated to the present mirror status.
- (h) Mirror image is not applied to the G53 command.
- (i) With the G62 command, either 0 or 1 must be specified after the axis. Any number other than 0 is regarded as 1. Therefore, each of the following has the same effect as X1:

G62 X-10;

G62 X50;

The G62 command allows applying mirror image ON and OFF for each axis respectively in one block.

G62 X1 Y0 Z1;The X and Z axes are ON while the Y axis is OFF

(4) Notes

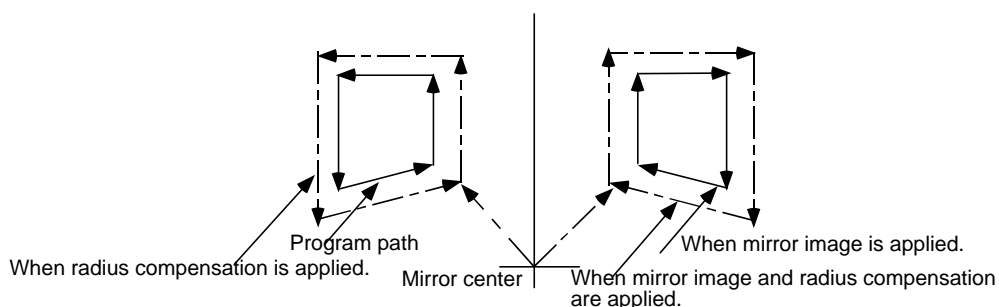
When mirror image is canceled anywhere except at the mirror image center, the program command position and machine position will deviate, as shown in the figure below. This state continues until positioning is executed with the absolute value command (G90 mode) in the program or a G28 or G30 command is assigned for zero point return. The mirror center is set with an absolute value commanded in the program and so if another mirror image is assigned in this state, the center may be set at an unforeseen position.

Cancel mirror image above the mirror center and, after cancellation, assign a positioning command using absolute values.

(5) Combination with other functions

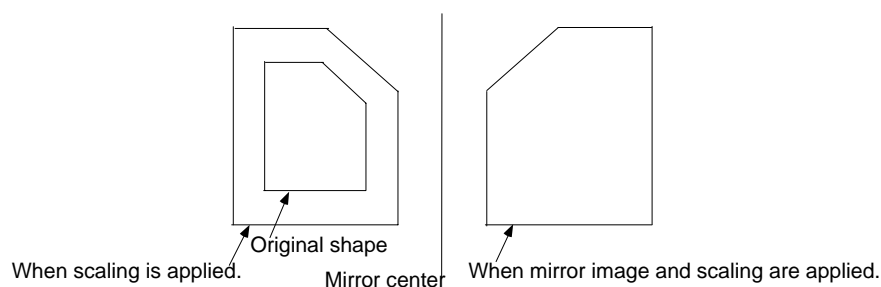
(a) Radius compensation

Mirror imaging is processed after a radius compensation command has been applied, and cutting proceeds as follows.



(b) Scaling

Mirror image applies to a shape after scaling so that cutting proceeds as follows.



13. PROGRAMMING SUPPORT FUNCTIONS
13.6 Variable Commands

13.6 Variable Commands

The M2 format variables (Dn(Ln), common variable n= 1 - 80, local variable n = 101 - 132) are converted into the present format before execution by adding the convertor function.

13.6.1 Outline

The program can have flexibility and general-purpose capabilities by designating variables, instead of giving direct numerical values to particular addresses in a program, and by assigning the values or those variables as required when executing a program.

The variables can also be operated (added, subtracted, multiplied, divided).

13.6.2 Detailed description

(1) Number of variable sets

	Number of sets	Number of set specifications
1	100 sets	#100 to #149, #500 to #549
2	200 sets	#100 to #199, #500 to #599
3	300 sets	#100 to #199, #500 to #699
4	600 sets	#100 to #199, #500 to #999
5	50+50 * 2 sets	#100 to #149: Independent for system, #500 to #549: Common for system
6	100+100 * 2 sets	#100 to #199: Independent for system, #500 to #599: Common for system

(2) Variable expressions

Variable : # Numerical value **Example) #100**
(Numerical value = 1, 2, 3,)
: # [expression] **Example) # [#100 + #200]**

Expression : Numerical value
: Variable
: Expression operator expression #100 + #101
: – (minus) expression –#120
: [Expression] [#110]
: Function [expression] SIN [#110]

- (a) The standard operators are "+", "-", "*", and "/". User macro specifications are required for other operators.
- (b) The user macro specifications are required for logical operators (OR, AND, etc.) and functions (SIN, COS, etc.)
- (c) An error will occur if the variable number is negative.

13. PROGRAMMING SUPPORT FUNCTIONS
13.6 Variable Commands

(3) Types of variables

The following table gives the types of variables.

Type of variable	Number	Function	Remarks
Common variables	100 to 149, 500 to 549	Can be used in common throughout main, sub and macro programs.	Type A : 100 sets
Local variables	1 to 33	Can be used for local variables in macro programs.	User macro specifications required.
System variables	1000 to	Application is fixed by system.	User macro specifications required.
Fixed cycle variables	1 to 99	Local variables in fixed cycle programs (Note) Only variables #15 to #21 can be used by the user.	Fixed cycle specifications required.

- (a) All common variables are retained even when the power is turned OFF.
 (b) The common variables are divided into the following two types.
- Common variables 1: 500 to 549, variables which can be used in common through all systems
 - Common variables 2: 100 to 149, variables which can be used in common in the programs of the system

(4) Variable quotations

- (a) Variables can be used for all addresses except O, N and / (slash).

When the variable value is used directly:

X#1 Value of #1 is used as the X value.

When the complement of the variable value is used:

X - #2 Value with the #2 sign changed is used as the X value.

When defining variables:

#3 = #5 Variable #3 uses the value of variable #5.

#1 = 1000 Variable #1 uses the value 1000 (which is treated as 1000.).

When defining the variable operation expression:

#1 = #2 + #3 - 100 The value of the operation result of #2 + #3 - 100. is used as the #1 value.

X [#1 + #3 + 1000] The value of the operation result of #1 + #3 + 1000. is used as the X value.

- (b) Notes

- 1) A variable cannot be defined in the same block as an address. It must be defined in a separate block.
- 2) Up to five sets of square brackets [] may be used.
- 3) There are no restrictions on the number of characters and number of variables for variable definition.
- 4) The variable values should be within a range from 0 to ±99999999.
If this range is exceeded, the operations may not be conducted properly.
- 5) The variable definitions are valid from the moment that the variables are actually defined.
- 6) Variable quotations are always regarded as having a decimal point at the end.

- (c) Variable name definition

Variable names can be set for #500 to #519.

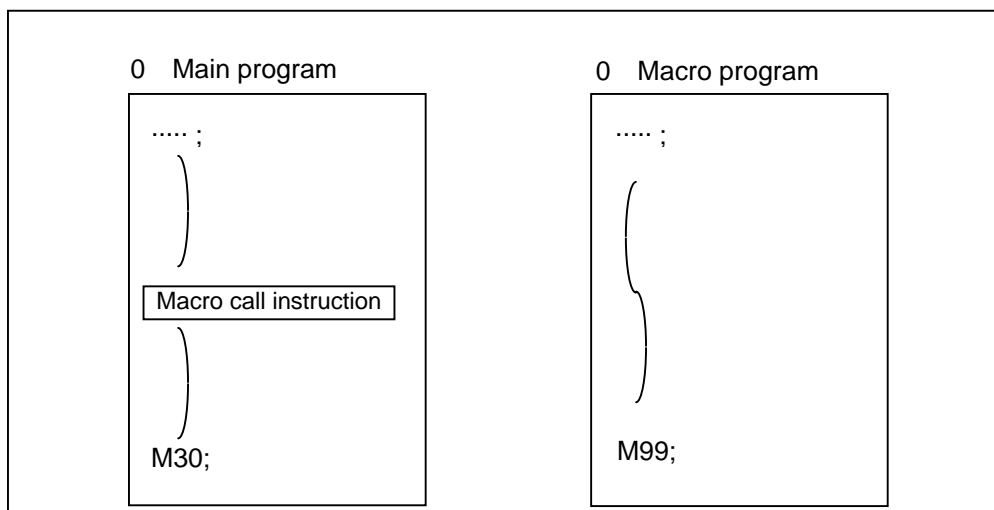
Note that variable names should be 7 alphanumeric characters or less, starting with an alphabetic character.

SETVn [NAME1, NAME2,] ; n: Head number of variable for attaching name
 GOOX [#NAME1] ; NAME1: Name with #n defined
 GOOY [#NAMEK] ; NAMEK: Name with #k defined
 :

13.7 User Macro Specifications

13.7.1 User macro commands

By combining the user macros with variable commands, it is possible to use macro program call, arithmetic operation, data input/output with PLC, control, decision, branch and many other instructions for measurement and other such applications.



Macro programs use variables, arithmetic instructions and control instructions to create subprograms which function to provide special-purpose control.

These special-purpose control functions (macro programs) are called by the macro call instructions exactly when required from the main program.

The following G codes are available for the macro call commands.

G code	Function
G65	User macro Simple call
G66	User macro Modal call A (Movement command call)
G66.1	User macro Modal call B (Per-block call)
G67	User macro Modal call cancel

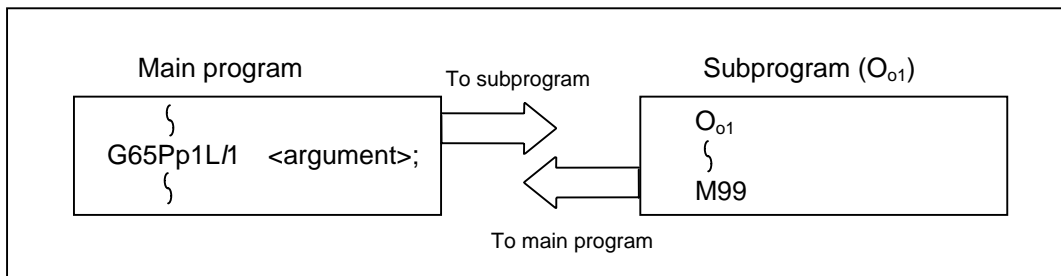
- (a) When the G66 (or 66.1) command is entered, the specified user macro subprogram will be called after each block has been executed (or after the movement command in the block) with the movement commands has been executed until the G67 (cancel) command is entered.
- (b) The G66 (or G66.1) and G67 commands must be paired in the same program.

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

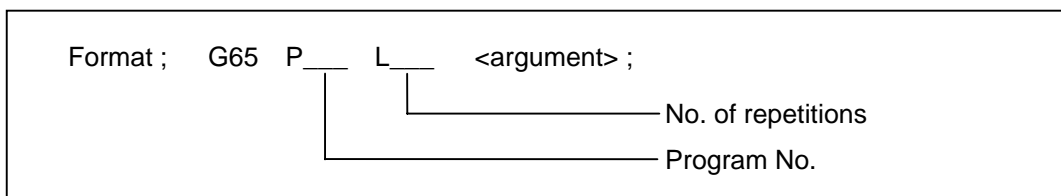
13.7.2 Macro call instruction

Included among the macro call commands are the simple calls which apply only to the instructed block and also modal calls (types A and B) which apply to each block in the call modal.

(1) Simple macro calls



M99 is used to conclude the user macro subprogram.



- (a) When the argument must be transferred as a local variable to a user macro subprogram, the actual value should be designated after the address. Regardless of the address, a sign and decimal point can be used in the argument. There are 2 ways in which arguments are designated.

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

1) Argument designation I

Format : A__ B__ C__ ···· X__ Y__ Z__

- Arguments can be designated using any address except G, L, N, O and P.
- Except for I, J and K, there is no need for designation in alphabetical order.
- I, J and K must be designated in alphabetical order.
I__ J__ K__ Correct
J__ I__ K__ Incorrect
- Address which do not need to be designated can be omitted.
- The following table shows the correspondence between the addresses which can be designated by argument designation I and the variable numbers in the user macro main body.

Address and variable number correspondence		Call instructions and usable address	
Argument designation I address	Variable in macro	G65, G66	G66.1
A	#1	○	○
B	#2	○	○
C	#3	○	○
D	#7	○	○
E	#8	○	○
F	#9	○	○
G	#10	×	× *
H	#11	○	○
I	#4	○	○
J	#5	○	○
K	#6	○	○
L	#12	×	× *
M	#13	○	○
N	#14	×	× *
O	#15	×	×
P	#16	×	× *
Q	#17	○	○
R	#18	○	○
S	#19	○	○
T	#20	○	○
U	#21	○	○
V	#22	○	○
W	#23	○	○
X	#24	○	○
Y	#25	○	○
Z	#26	○	○

- : Can be used.
- × : Cannot be used.
- * : Can be used while G66.1 command is modal.

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

2) Argument designation II

Format : A__ B__ C__ I__ J__ K__ I__ J__ K__

- In addition to address A, B and C, up to 10 groups of arguments with I, J, K serving as 1 group can be designated.
- When the same address is duplicated, designate the addresses in the specified order.
- Addresses which do not need to be designated can be omitted.
- The following table shows the correspondence between the addresses which can be designated by argument designation II and the variable numbers in the user macro main body.

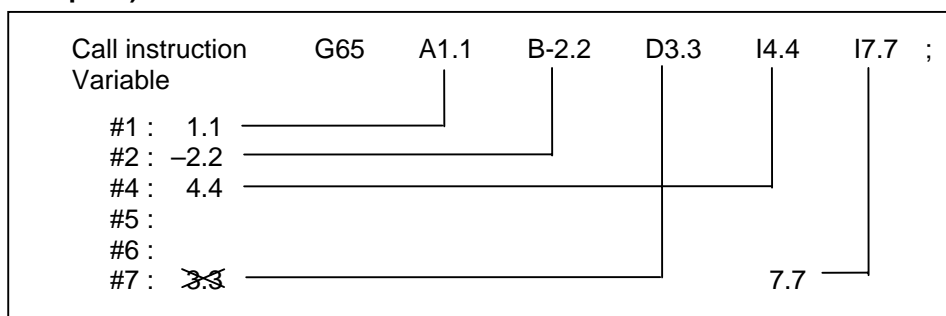
Argument designation II address	Variable within macro	Argument designation II address	Variable within macro
A	# 1	J5	#17
B	# 2	K5	#18
C	# 3	I6	#19
I1	# 4	J6	#20
J1	# 5	K6	#21
K1	# 6	I7	#22
I2	# 7	J7	#23
J2	# 8	K7	#24
K2	# 9	I8	#25
I3	#10	J8	#26
J3	#11	K8	#27
K3	#12	I9	#28
I4	#13	J9	#29
J4	#14	K9	#30
K4	#15	I10	#31
I5	#16	J10	#32
		K10	#33

Note 1) The numbers 1 through 10 accompanying I, J and K denote the sequence of the commanded groups and they are not required for the actual instructions.

3) Using arguments designations I and II together

If addresses corresponding to the same variable are commanded when both types I and II are used to designate arguments, the latter address is valid.

(Example 1)

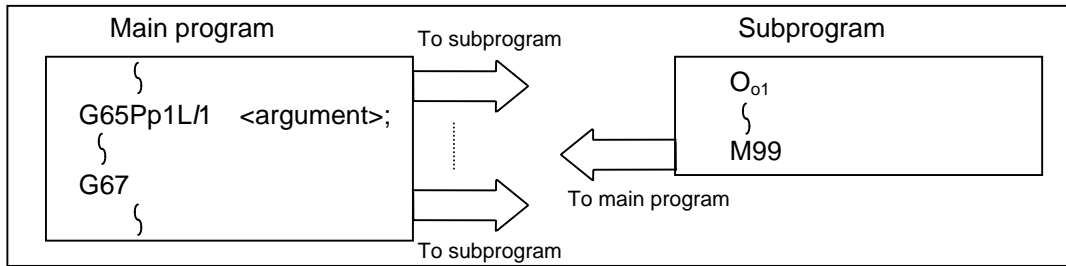


In the above example, the last I7.7 argument is valid when both arguments D3.3 and I7.7 are commanded for the #7 variable.

13. PROGRAMMING SUPPORT FUNCTIONS

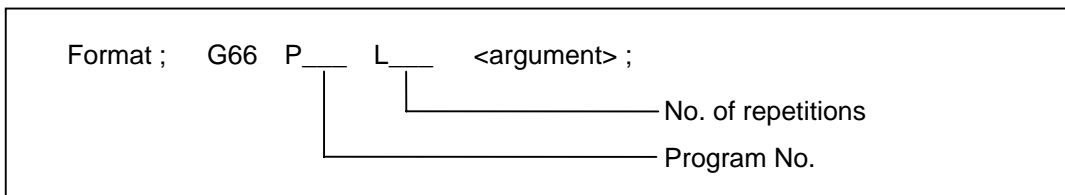
13.7 User Macro Specifications

(2) Modal call A (movement command call)



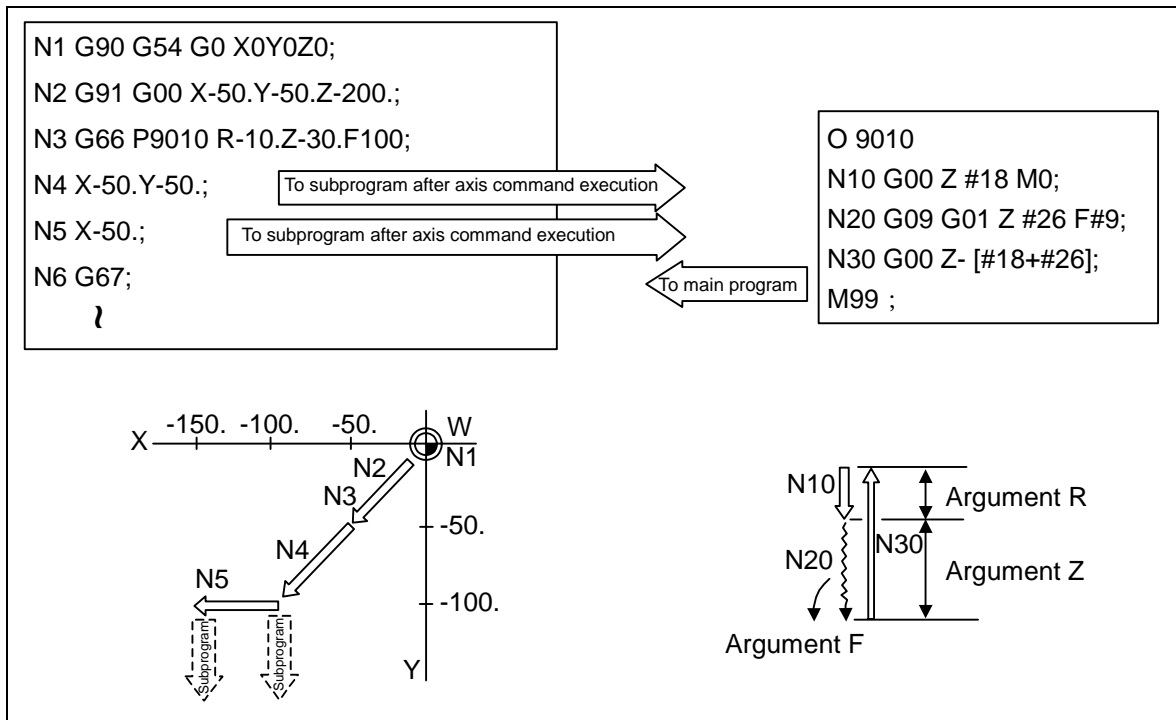
When the block with a movement command is commanded between G66 and G67, the movement command is first executed and then the designated user macro subprogram is executed. The number of times the subprogram is executed is λ 1 times with each call.

The argument is the same as for a simple call.



- (a) When the G66 command is entered, the specified user macro subprogram will be called after the movement command in the block with the movement commands has been executed until the G67 (cancel) command is entered.
- (b) The G66 and G67 commands must be paired in the same program.
A program error will result when G67 is issued without the G66 command.

(Example) Drill cycle



Note 1) After the axis command is executed in the main program, the subprogram is executed.

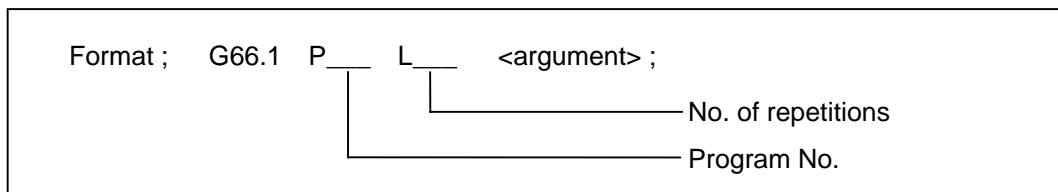
Note 2) The subprogram is not executed in the blocks following G67.

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

(3) Modal call B (for each block)

The specified user macro subprogram is called unconditionally for each command block which is assigned between G66.1 and G67 and the subprogram is executed λ 1 times.



- (a) In the G66.1 mode, everything except the O, N and G codes in the various command blocks which are read are handled as the argument without being executed. Any G code designated last or any N code commanded after anything except O and N will function as the argument.
- (b) The same applies as when G65P__ is assigned at the head of a block for all significant blocks in the G66.1 mode.

(Example 1)

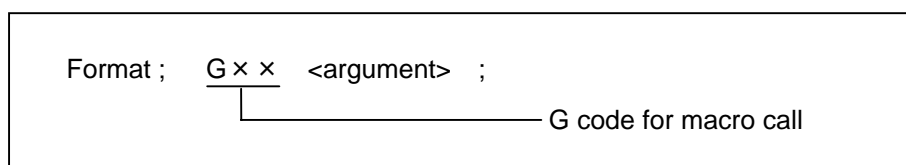
N100 G01 G90 X100. Y200. F400 R1000; in the G66.1 P1000; mode is the same as:
N100 G65 P1000 G01 G90 X100. Y200. F400 R1000;

Note 1) The Call is performed even in the G66.1 command block in the G66.1 mode and the correspondence between the argument address and the variable number is the same as for G65 (simple call).

- (c) The range of the G and N command values which can be used anew as variables in the G66.1 mode is subject to the restrictions applying to values as normal NC command values.
- (d) O, sequence numbers N and modal G codes are updated as modal information.

(4) G code macro call

User macro subprogram with prescribed program numbers can be called merely by issuing the G code command.



- (a) The above instruction functions in the same way as the instructions below, and parameters are set for each G code to determine the correspondence with the instructions.
 - a. M98P△△△△ ;
 - b. G65P△△△△△ <argument> ;
 - c. G66P △△△△△ <argument> ;
 - d. G66.1P△△△△△ <argument> ;

When the parameters corresponding to c and d above are set, issue the cancel command (G67) either in the user macro or after the call code has been commanded so as to cancel the modal call.

13. PROGRAMMING SUPPORT FUNCTIONS

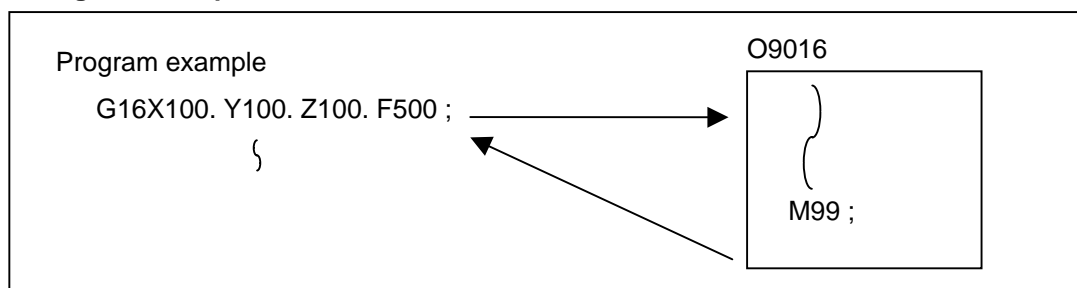
13.7 User Macro Specifications

- (b) The correspondence between the "x x" which conducts the macro call and the program number PΔΔΔΔ of the macro to be called is set by parameter.
- (c) Up to 10 G codes from G100 to G255 can be used with this instruction. (G01 to 99 can also be used with parameter #120032 Gmac_P).

Note 1) G101 to G110 and G200 to G202 are user macro I codes, but if the parameters are set as the G code call codes, the G code call will be the priority, and these codes cannot be used for user macro I.

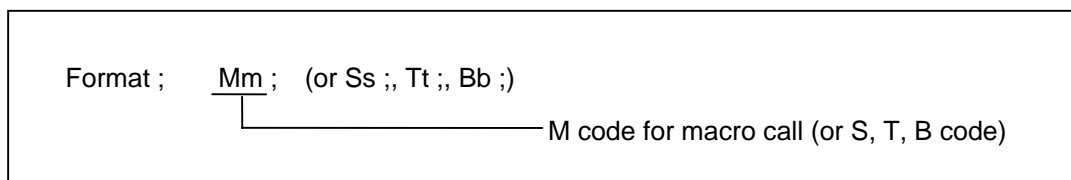
- (d) These commands cannot be issued during a user macro subprogram which has been called by a G code.

Program example



(5) Miscellaneous command macro call (for M, S, T, B code macro call)

The user macro subprogram of the specified program number can be called merely by issuing an M (or S, T, B) code. (Only entered codes apply for M but all S, T and B codes apply.)



- (a) The above instruction functions in the same way as the instructions below, and parameters are set for each M code to determine the correspondence with the instructions. (Same for S, T and B codes)
 - a. M98PΔΔΔΔ ;
 - b. G65PΔΔΔΔ Mm ;
 - c. G66P ΔΔΔΔ Mm ;
 - d. G66.1PΔΔΔΔ Mm ;

When the parameters corresponding to c and d above are set, issue the cancel command (G67) either in the user macro or after the call code has been commanded so as to cancel the modal call.

- (b) The correspondence between the "Mm" which conducts the macro call and the program number PΔΔΔΔ of the macro to be called is set by parameter. Up to 10 M codes from M00 to M95 can be entered. Note that the codes to be registered are the codes basically required for the machine, and codes excluding M0, M1, M2, M30 and M96 to M99.
- (c) As with M98, it is displayed on the CRT screen of the setting and display unit but the M codes and MF are not output.

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

- (d) Even if the miscellaneous command entered above is issued during a user macro subprogram called by the M code, macro call will not result and it will be handled as an ordinary miscellaneous command.
- (e) All S, T and B codes call the subprograms in the prescribed program numbers of the corresponding S, T and B functions.

(6) Macro call command nesting depth

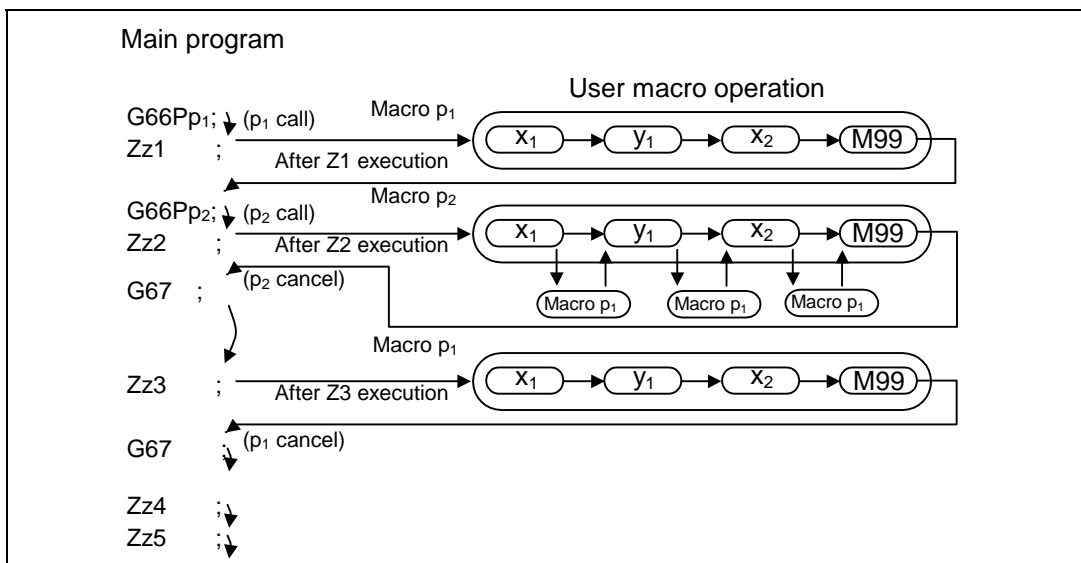
Up to 4 nesting levels are available for macro subprogram calls based on simple call or modal call.

The argument with a macro call instruction is valid only on the called macro level. Since the nesting depth for macro calls extends up to 4 levels, the argument can be used as a local variable for the program with each respective macro call.

Note 1) When a G65, G66, G66.1 G code macro call or miscellaneous command macro call is conducted, this is regarded as nesting level 1 and the level of the local variables is also incremented by one.

Note 2) The designated user macro subprogram is called every time the movement command is executed with modal call A. However, when the G66 command has been duplicated, the next user macro subprogram is called every time an axis is moved even with movement commands in the macro. User macro subprograms are called in sequence from the subprogram commanded last.

(Example 1)



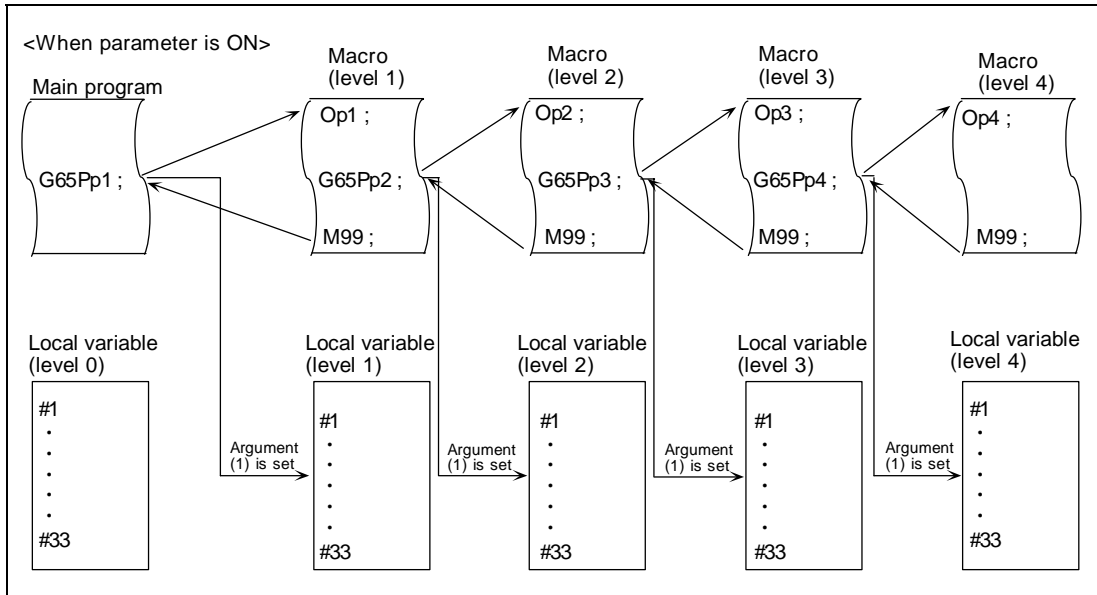
13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

(7) Local variables

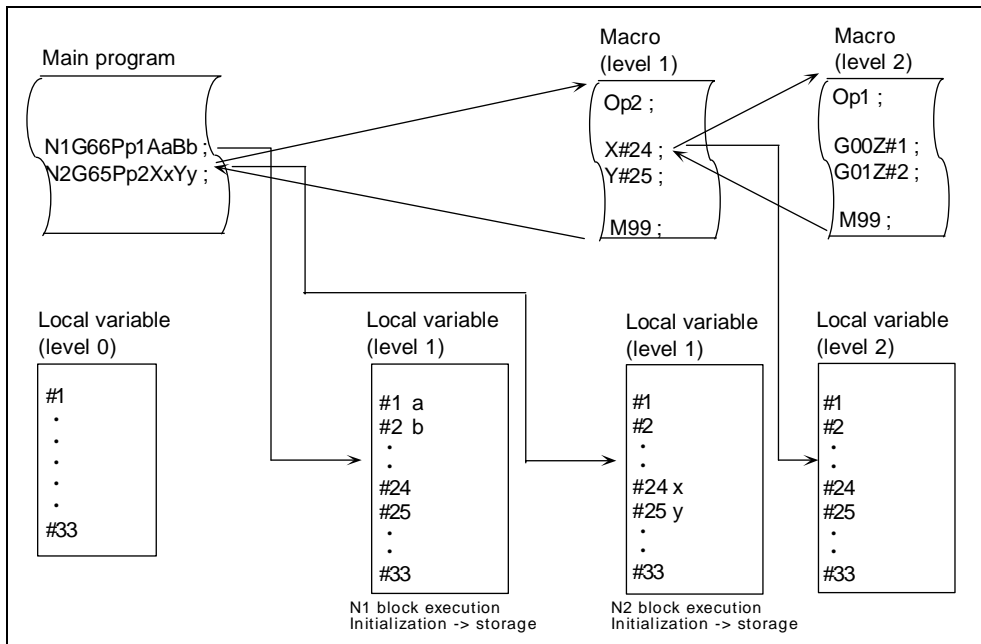
When a macro is called out with the simple call (G65), modal call A (G66), modal call B (G66.1) or G code/MSTB code call, the macro multiple (level) will increase by one. The local variable level will also increase by one. The local variable is set if there is an argument in the corresponding level when the macro program is called. The relation of the macro multiple (level) and local variable level is shown below.

(Example 1) For multiple call



(Example 2) For continuous call

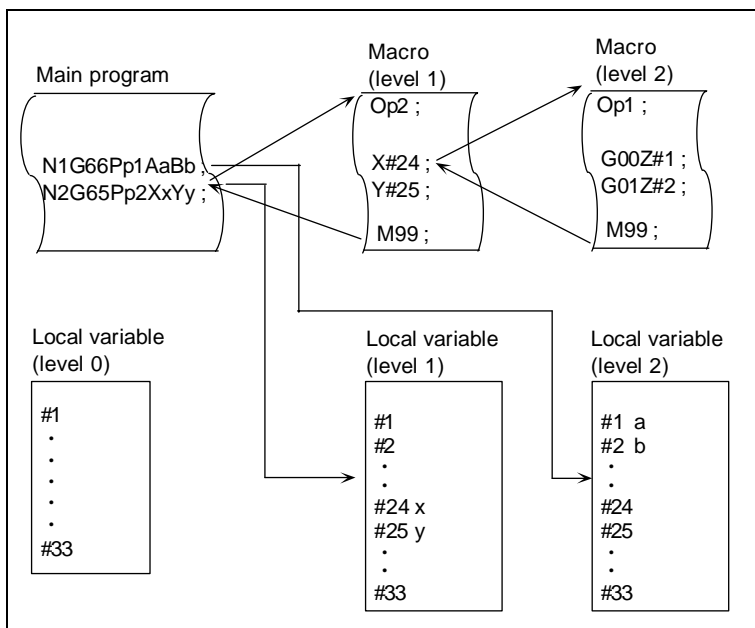
(a) Types stored in local variables in order of main program execution



13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

(b) Types stored in local variables in order of macro program call order



(Note) The set local variable is initialized with macro return (M99) when using simple call (G65). When using modal call A (G66) or B (G66.1), the local variable is initialized with cancel (G67).

The types (a) and (b) can be changed with the parameter (#110138 macro call LVAR type).

(8) Differences between M98 and G65 commands

M98 is simply a subprogram call command. Thus, observe the following points.

- (a) The argument can be designated for G65 but not for M98.
- (b) The sequence number can be designated for M98 but no for G65, G66 and G66.1.
- (c) M98 executes a subprogram after all the commands except M, P, H and L in the M98 block have been executed, but G65 branches to the subprogram without any further operation.
- (d) When any address except O, N, P, H or L is included in the M98 block, single block stop results. This is not the case with G65.
- (e) The level of the M98 local variables is fixed but it can be varied in accordance with the nesting depth for G65. (#1, for instance, has the same significance either before or after M98 but a different significance in each case with G65.)
- (f) The M98 nesting depth extends up to 8 levels in combination with G65, G66 and G66.1. The G65 nesting depth extends up to only 4 levels in combination with G66 and G66.1.

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

13.7.3 Variables

Both the variable specifications and user macro specifications are required for the variables which are used with the user macros.

The offset amounts of the local, common and system variables among the variables for this MELDAS NC system except #33 are retained even when the unit's power is turned OFF. (Common variables can also be cleared by parameter #310004 Com-var RST clear.)

(1) Use of multiple variables

When the user macro specifications applied, variable numbers can be turned into variables (multiple use of variables) or replaced by <formula>. Only one of the four basic arithmetic rule (+, -, ×, ÷) operations can be conducted with <formula>.

(Example 1) Multiple use of variables

#1 = 10 #10 = 20 #20 = 30 ;	#[[#1]] = #[#10] from #1 = 10.
#5 = #[[#1]] ;	#[#10] = #20 from #10 = 20.
	Therefore, #5 = #20 or #5 = 30.
#1 = 20 #10 = 20 #20 = 30 #5 = 1000 ;	#[[#1]] = #[#10] from #1 = 10.
#[[#1]] = #5 ;	#[#10] = #20 from #10 = 20.
	Therefore, #20 = #5 or #20 = 1000.

(Example 2) Example of multiple designation of variables

#10 = 5	<Formula> ##10 = 100; is handled in the same
In which case ##10 = 100 ; #5 = 100	manner as # [#10] = 100.

(Example 3) Replacing variable numbers with <formula>

#10 = 5 ;	
#[#10 + 1] = 1000 ;	In which case, #6 = 1000.
#[#10 - 1] = -1000 ;	In which case, #4 = -1000.
#[#10*3] = 100 ;	In which case, #15 = 100.
#[#10/2] = -100 ;	In which case, #2 = -100.

(2) Undefined variables

Variables applying with the user macro specifications such as variables which have not been used even once after the power was turned ON or local variables not quoted by the G65, G66 or G66.1 commands can be used as <vacant>. Also, variables can forcibly be set to <vacant>. Variable #0 is always used as the <vacant> variable and cannot be defined in the left-side member.

(a) Arithmetic expressions

#1 = #0 ;	#1 = <vacant>
#2 = #0 + 1 ;	#2 = 1
#3 = 1 + #0 ;	#3 = 1
#4 = #0*10 ;	#4 = 0
#5 = #0 + #0 ;	#5 = 0

It should be borne in mind that <vacant> in an arithmetic expression is handled in the same way as 0.

<Vacant> + <Vacant> = 0
<Vacant> + <Constant> = Constant
<Constant> + <Vacant> = Constant

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

(b) Variable quotations

When undefined variables only are quoted, they are ignored up to the address.

When #1 = <Vacant>

G0 X#1 Y1000 ; Equivalent to G0 Y1000 ;
 G0 X#1 + 10 Y1000 ; Equivalent to G0 X10 Y1000 ;

(c) Conditional expressions

<Vacant> and 0 are not equivalent for EQ and NE only. (#0 means <vacant>.)

When #101 = <Vacant>	When #101 = 0
#101 EQ #0 <Vacant> = <Vacant> established	#101 EQ #0 0 = <Vacant> not established
#101 NE 0 <Vacant> ≠ 0 established	#101 NE 0 0 ≠ 0 not established
#101 GE #0 <Vacant> ≥ <Vacant> established	#101 GE #0 0 ≥ <Vacant> established
#101 GT 0 <Vacant> > 0 not established	#101 GT 0 0 > 0 not established
#101 LE #0 <Vacant> ≤ <Vacant> established	#101 LE #0 0 ≤ <Vacant> established
#101 LT 0 <Vacant> < 0 not established	#101 LT 0 0 < 0 not established

Note 1) EQ and NE should be used only for integers. For comparison of numeric values with decimals, GE, GT, LE, and LT should be used.

List of condition establishment and establishment failure

(conditional expression containing non-defined variables)

Right side \ Left side	EQ		NE		GT		LT		GE		LE	
	Blank	Con-stant	Blank	Con-stant	Blank	Con-stant	Blank	Con-stant	Blank	Con-stant	Blank	Con-stant
Blank	○	x	x	○	x	x	x	○	○	x	○	x
Constant	x	○	○	x	○	x	x	○	○	x	x	○

○ ····· Conditional expression established
 x ····· Conditional expression establishment failed

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

13.7.4 Types of variables

(1) Common variables

Common variables can be used commonly from any position. Number of the common variables sets depends on the specifications. Refer to "13.6 Variable commands" for details.

(2) Local variables (#1 to #33)

These can be defined as an argument when a macro subprogram is called or used locally within main programs and subprograms. They can be duplicated regardless of the relationship existing between macros (up to 4 levels).

G65 P_{p₁} L_{l₁} <argument> ;

Where P₁ = Program number
l₁ = Number of repetitions

(Note) The argument is assumed to be Aa1 Bb1 Cc1 Zz1.

The following table shows the correspondences between the addresses designated by argument and the local variable numbers used in the user macro main bodies.

[Argument specification I]

Call command		Argument address	Local variable number	Call command		Argument address	Local variable number
G65 G66	G66.1			G65 G66	G66.1		
○	○	A	#1	○	○	Q	#17
○	○	B	#2	○	○	R	#18
○	○	C	#3	○	○	S	#19
○	○	D	#7	○	○	T	#20
○	○	E	#8	○	○	U	#21
○	○	F	#9	○	○	V	#22
x	x*	G	#10	○	○	W	#23
○	○	H	#11	○	○	X	#24
○	○	I	#4	○	○	Y	#25
○	○	J	#5	○	○	Z	#26
○	○	K	#6			-	#27
x	x*	L	#12			-	#28
○	○	M	#13			-	#29
x	x*	N	#14			-	#30
x	x	O	#15			-	#31
x	x*	P	#16			-	#32
						-	#33

"x" in the above table denotes an argument address which cannot be used. However, provided that the G66.1 mode has been established, an argument address denoted by the asterisk can be added for use.

"-" denotes that a corresponding address is not available.

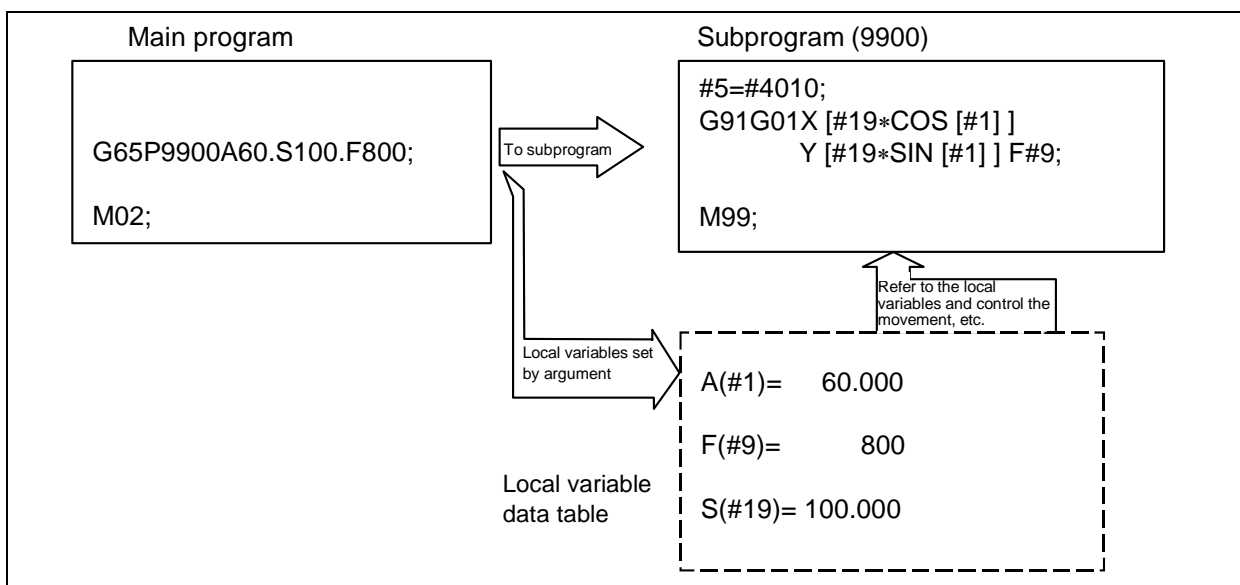
13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

[Argument specification II]

Argument specification II address	Variable in macro	Argument specification II address	Variable in macro
A	# 1	I6	#19
B	# 2	J6	#20
C	# 3	K6	#21
I1	# 4	I7	#22
J1	# 5	J7	#23
K1	# 6	K7	#24
I2	# 7	I8	#25
J2	# 8	J8	#26
K2	# 9	K8	#27
I3	#10	I9	#28
J3	#11	J9	#29
K3	#12	K9	#30
I4	#13	I10	#31
J4	#14	J10	#32
K4	#15	K10	#33
I5	#16		
J5	#17		
K5	#18		

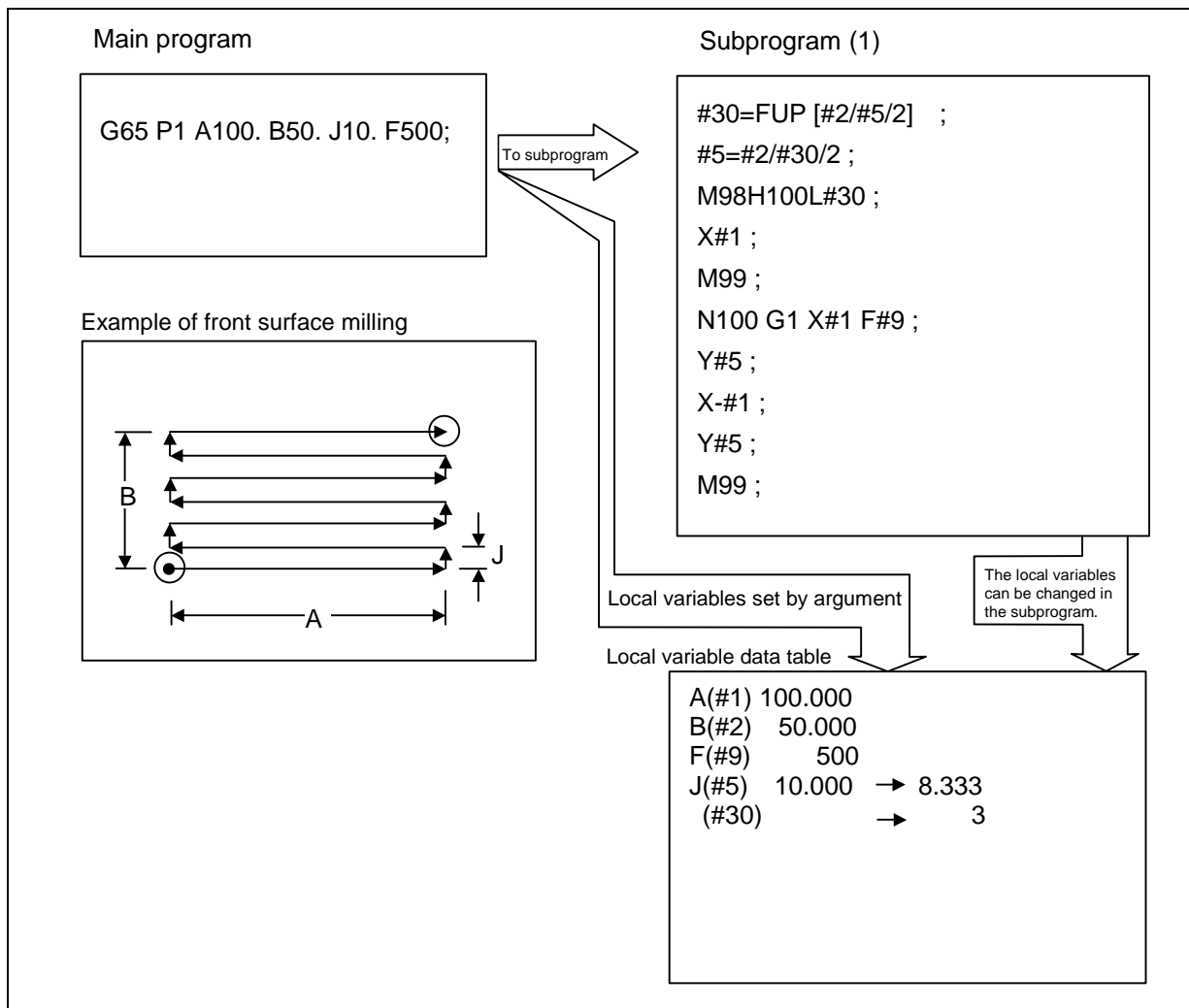
Subscripts 1 to 10 for I, J, and K indicate the order of the specified command sets. They are not required to specify instructions.

Local variables in subprograms can be defined by means of the <argument> designation during macro call.



13. PROGRAMMING SUPPORT FUNCTIONS 13.7 User Macro Specifications

The local variables can be used freely in that subprogram.

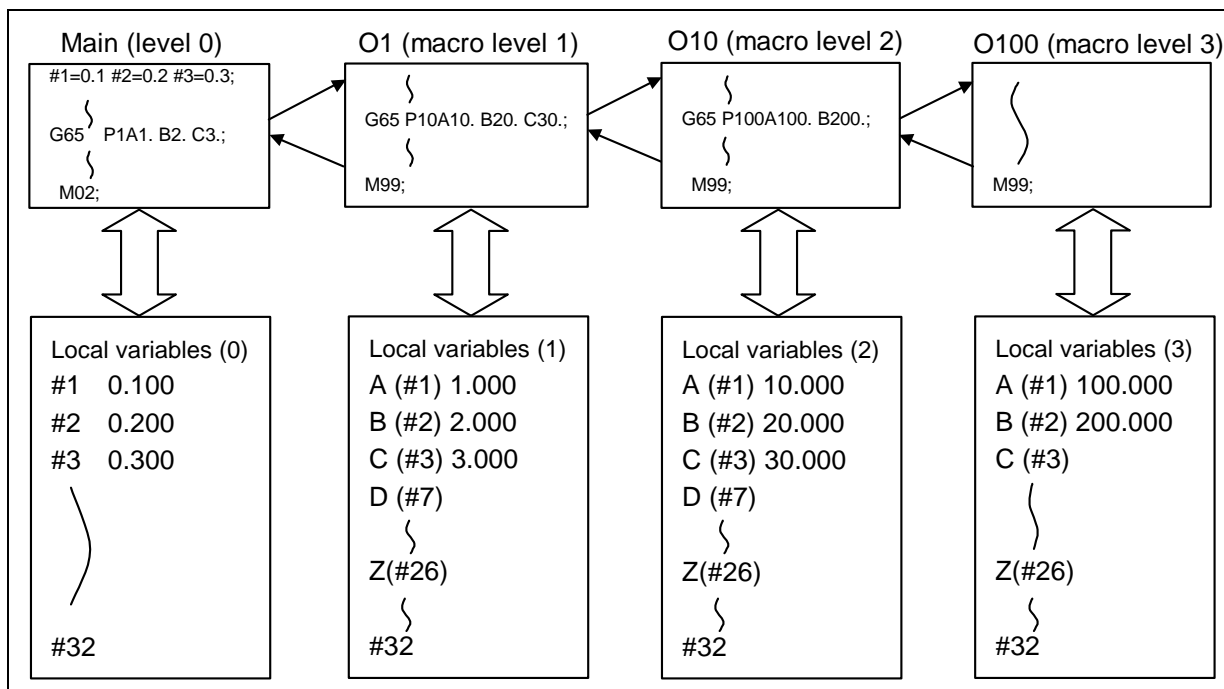


In the front surface milling example, argument J is programmed as the milling pitch 10.mm. However, this is changed to 8.333mm to create an equal interval pitch. The results of the No. of reciprocation data calculation is set in local variable #30.

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

Local variables can be used independently on each of the macro call levels (4 levels).
 Local variables are also provided independently for the main program (macro level 0).
 Arguments cannot be used for the level 0 local variables.



The status of the local variables appear on the setting and display unit.
 Refer to the Instruction Manual for details.

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(3) Macro interface inputs (#1000 ~ #1041)

The status of the interface input signals can be ascertained by reading out the values of variable numbers 1000 to 1041. A variable value which has been read out can be only one of 2 values: 1 or 0 (1: contact closed, 0: contact open). All the input signals from #1000 to #1031 can be read at once by reading out the value of variable number 1032. Variable numbers #1000 to #1041 are for readout only, and cannot be placed in the left side member of their arithmetic formula. Input here refers to input to the control unit.

System variable	No. of points	Interface input signal	System variable	No. of points	Interface input signal
#1000	1	Register R124 bit 0	#1016	1	Register R125 bit 0
#1001	1	Register R124 bit 1	#1017	1	Register R125 bit 1
#1002	1	Register R124 bit 2	#1018	1	Register R125 bit 2
#1003	1	Register R124 bit 3	#1019	1	Register R125 bit 3
#1004	1	Register R124 bit 4	#1020	1	Register R125 bit 4
#1005	1	Register R124 bit 5	#1021	1	Register R125 bit 5
#1006	1	Register R124 bit 6	#1022	1	Register R125 bit 6
#1007	1	Register R124 bit 7	#1023	1	Register R125 bit 7
#1008	1	Register R124 bit 8	#1024	1	Register R125 bit 8
#1009	1	Register R124 bit 9	#1025	1	Register R125 bit 9
#1010	1	Register R124 bit 10	#1026	1	Register R125 bit 10
#1011	1	Register R124 bit 11	#1027	1	Register R125 bit 11
#1012	1	Register R124 bit 12	#1028	1	Register R125 bit 12
#1013	1	Register R124 bit 13	#1029	1	Register R125 bit 13
#1014	1	Register R124 bit 14	#1030	1	Register R125 bit 14
#1015	1	Register R124 bit 15	#1031	1	Register R125 bit 15

System variable	No. of points	Interface input signal
#1032	32	Register R124, R125
#1033	32	Register R126, R127
#1034	32	Register R128, R129
#1035	32	Register R130, R131
#1036	32	Register R140, R141
#1037	32	Register R142, R143
#1038	32	Register R144, R145
#1039	32	Register R146, R147
#1040	32	Register R148, R149
#1041	32	Register R150, R151

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(4) Macro interface outputs (#1100 ~ #1141)

The interface output signals can be sent by substituting values in variable numbers 1100 to 1141. An output signal can be only 0 or 1.

All the output signals from #1100 to #1131 can be sent at once by substituting a value in variable number 1132. ($2^0 \sim 2^{31}$)

The status of the writing and output signals can be read in order to offset the #1100 to #1141 output signals. Output here refers to the output from the NC.

System variable	No. of points	Interface input signal	System variable	No. of points	Interface input signal
#1100	1	Register R24 bit 0	#1116	1	Register R25 bit 0
#1101	1	Register R24 bit 1	#1117	1	Register R25 bit 1
#1102	1	Register R24 bit 2	#1118	1	Register R25 bit 2
#1103	1	Register R24 bit 3	#1119	1	Register R25 bit 3
#1104	1	Register R24 bit 4	#1120	1	Register R25 bit 4
#1105	1	Register R24 bit 5	#1121	1	Register R25 bit 5
#1106	1	Register R24 bit 6	#1122	1	Register R25 bit 6
#1107	1	Register R24 bit 7	#1123	1	Register R25 bit 7
#1108	1	Register R24 bit 8	#1124	1	Register R25 bit 8
#1109	1	Register R24 bit 9	#1125	1	Register R25 bit 9
#1110	1	Register R24 bit 10	#1126	1	Register R25 bit 10
#1111	1	Register R24 bit 11	#1127	1	Register R25 bit 11
#1112	1	Register R24 bit 12	#1128	1	Register R25 bit 12
#1113	1	Register R24 bit 13	#1129	1	Register R25 bit 13
#1114	1	Register R24 bit 14	#1130	1	Register R25 bit 14
#1115	1	Register R24 bit 15	#1131	1	Register R25 bit 15

System variable	No. of points	Interface input signal
#1132	32	Register R24, R25
#1133	32	Register R26, R27
#1134	32	Register R28, R29
#1135	32	Register R30, R31
#1136	32	Register R40, R41
#1137	32	Register R42, R43
#1138	32	Register R44, R45
#1139	32	Register R46, R47
#1140	32	Register R48, R49
#1141	32	Register R50, R51

Note 1) The last values of the system variables #1100 to #1141 sent are retained as 1 or 0. (They are not cleared even with resetting.)

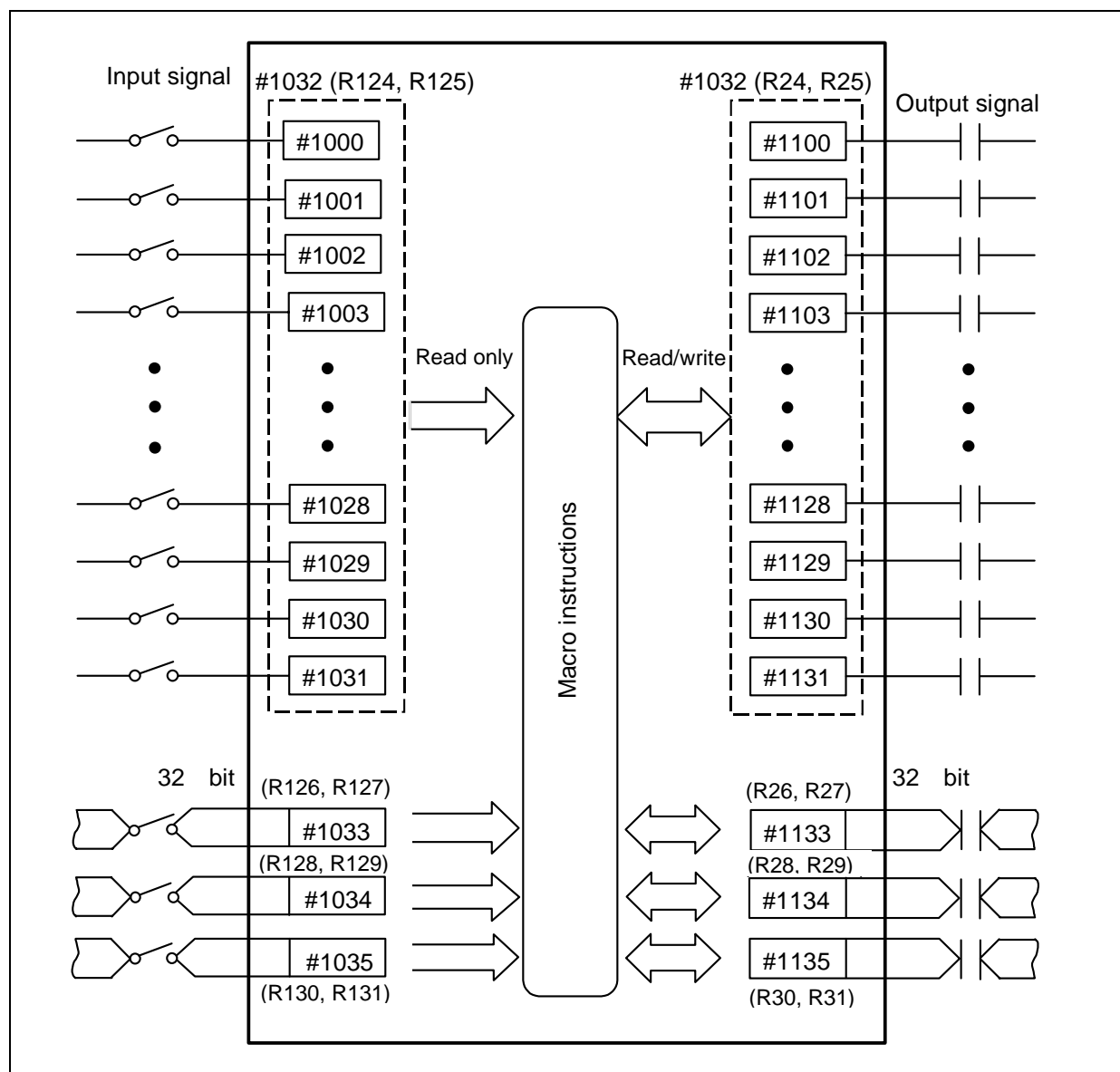
Note 2) The following applies when any number except 1 or 0 is substituted into #1100 to #1131.

<Vacant> is treated as 0.

Any number except 0 and <vacant> is treated as 1.

Any value less than 0.00000001 is indefinite.

13. PROGRAMMING SUPPORT FUNCTIONS 13.7 User Macro Specifications



(5) Normal line control parameter (#1900, #1901)

The normal line control C axis turning diameter and normal line control C axis insertion diameter can be read and written with the variable numbers #1900 and #1901.

Variable No.	Details
#1900	Machining parameter "#330015 Normal C axis turn R"
#1901	Machining parameter "#330016 Norm C insrt radius"

(6) Tool offset

Variable number range		Type 1	Type 2
#10001 ~ #10000 + n	#2001 ~ #2000 + n	○	○ (Length dimension)
#11001 ~ #11000 + n	#2201 ~ #2200 + n	×	○ (Length wear)
#16001 ~ #16000 + n	#2401 ~ #2400 + n	×	○ (Radius dimension)
#17001 ~ #17000 + n	#2601 ~ #2600 + n	×	○ (Radius wear)

13. PROGRAMMING SUPPORT FUNCTIONS 13.7 User Macro Specifications

Tool data can be read and values substituted using the variable numbers.

Either the numbers in the #10000 order or #2000 order can be used.

The last 3 digits of the variable numbers correspond to the tool offset number.

n corresponds to the No. of tool offset sets.

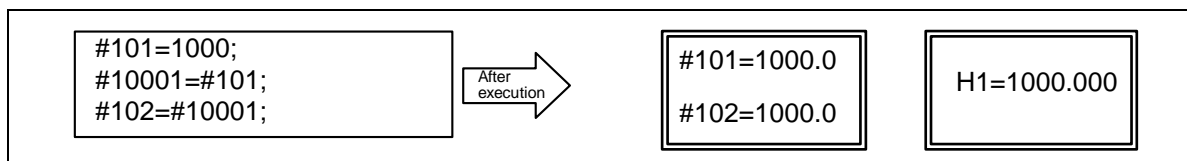
If there are 4000 tool offset sets and type 2 is being used, avoid variable Nos. in the #2000 order, and instead use the #10000 order.

The tool offset data are configured as data with a decimal point in the same way as for other variables. Consequently, this decimal point must be commanded when data below the decimal point is to be entered.

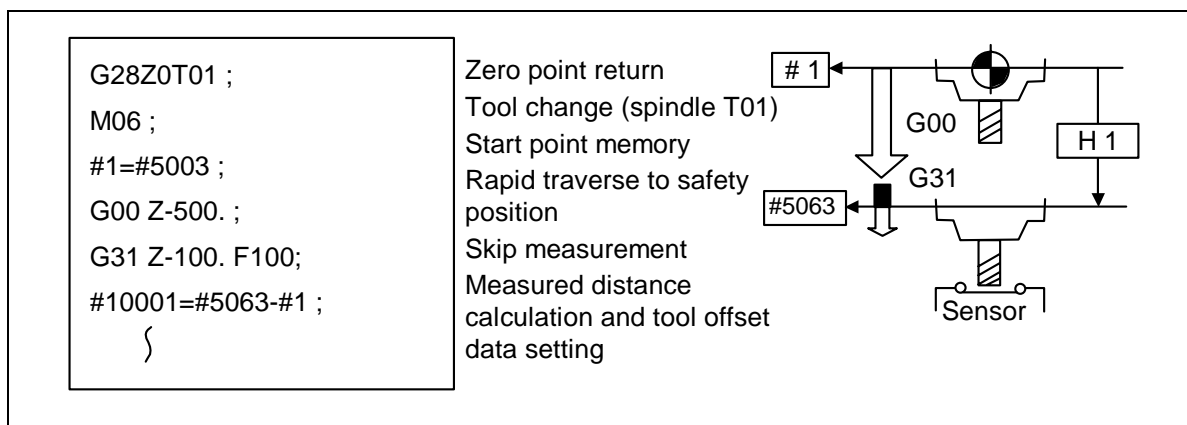
Programming example

Common variables

Tool offset data



(Example 1) Calculation and tool offset data setting



Note) In this example, no consideration is given to the delay in the skip sensor signal. #5003 is the Z-axis start point position and #5063 is the Z-axis skip coordinates, and indicated is the position at which the skip signal is input while G31 is being executed.

13. PROGRAMMING SUPPORT FUNCTIONS

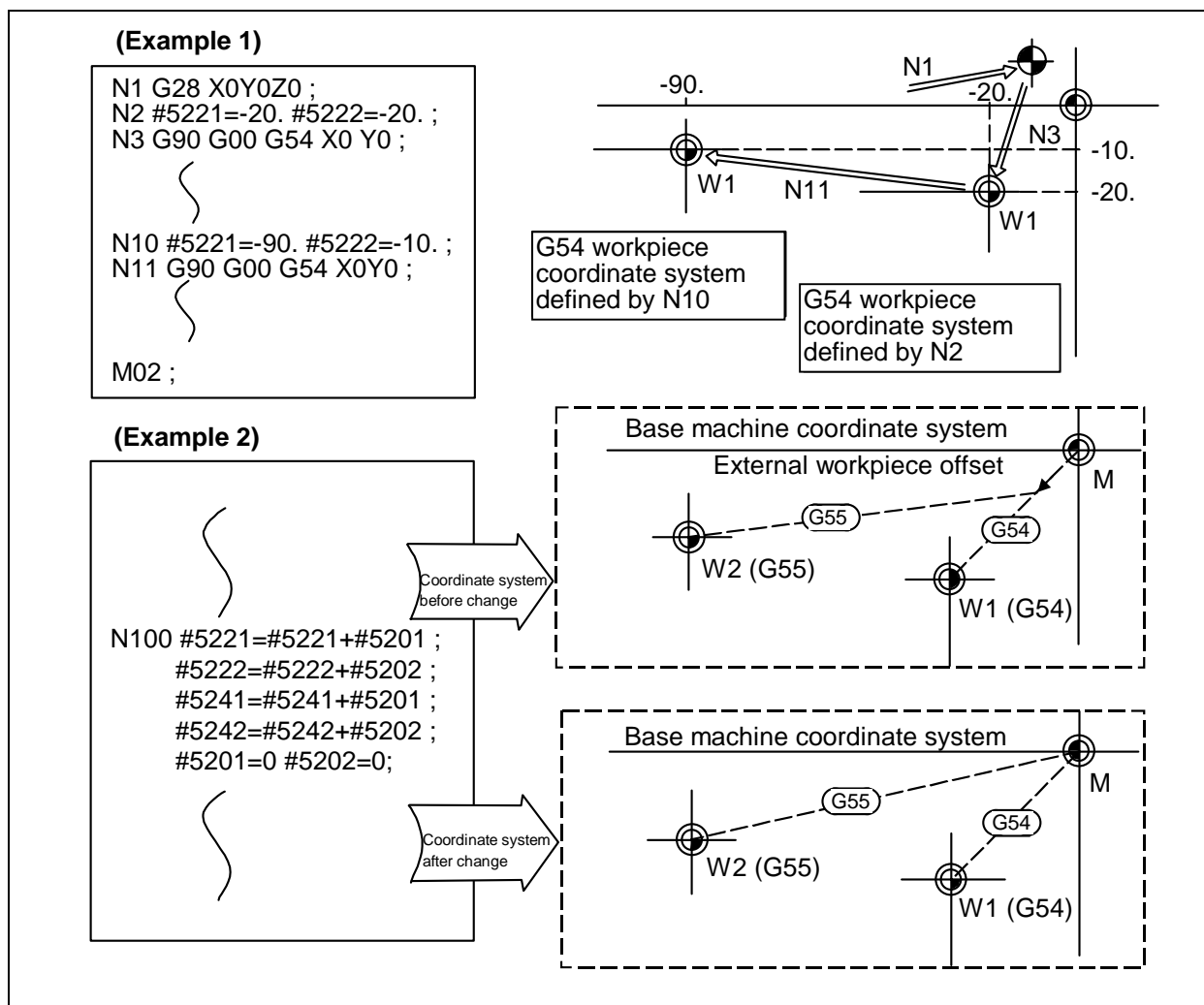
13.7 User Macro Specifications

(7) Workpiece coordinate system offset

By using variable numbers 5201 to 5324, it is possible to read out the workpiece coordinate system offset data or to substitute values.

Note) The number of axes which can be controlled differs according to the specifications.

Axis No.	Axis 1	Axis 2	Axis 3	Axis 4	Remarks
External workpiece offset	#5201	#5202	#5203	#5204	External workpiece offset specifications are required.
G54	#5221	#5222	#5223	#5224	
G55	#5241	#5242	#5243	#5244	
G56	#5261	#5262	#5263	#5264	
G57	#5281	#5282	#5283	#5284	
G58	#5301	#5302	#5303	#5304	
G59	#5321	#5322	#5323	#5324	



This is an example where the external workpiece offset values are added to the workpiece coordinate (G54, G55) system offset values without changing the position of the workpiece coordinate systems.

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(8) Extended workpiece coordinate system offset

The extended workpiece coordinate system offset data can be read, and values can be substituted using the variable numbers 7000 to 7944.

(Note) This is valid when the extended workpiece coordinate system specifications are provided.

	Axis 1 to Axis 4
P1	#7001 to #7004
P2	#7021 to #7024
P3	#7041 to #7044
P4	#7061 to #7064
P5	#7081 to #7084
P6	#7101 to #7104
P7	#7121 to #7124
P8	#7141 to #7144
P9	#7161 to #7164
P10	#7181 to #7184
P11	#7201 to #7204
P12	#7221 to #7224
P13	#7241 to #7244
P14	#7261 to #7264
P15	#7281 to #7284
P16	#7301 to #7304
P17	#7321 to #7324
P18	#7341 to #7344
P19	#7361 to #7364
P20	#7381 to #7384
P21	#7401 to #7404
P22	#7421 to #7424
P23	#7441 to #7444
P24	#7461 to #7464

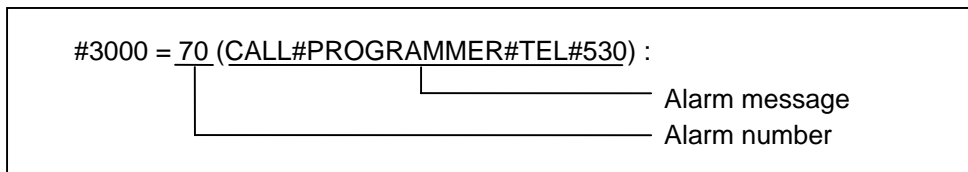
	Axis 1 to Axis 4
P25	#7481 to #7484
P26	#7501 to #7504
P27	#7521 to #7524
P28	#7541 to #7544
P29	#7561 to #7564
P30	#7581 to #7584
P31	#7601 to #7604
P32	#7621 to #7624
P33	#7641 to #7644
P34	#7661 to #7664
P35	#7681 to #7684
P36	#7701 to #7704
P37	#7721 to #7724
P38	#7741 to #7744
P39	#7761 to #7764
P40	#7781 to #7784
P41	#7801 to #7804
P42	#7821 to #7824
P43	#7841 to #7844
P44	#7861 to #7864
P45	#7881 to #7884
P46	#7901 to #7904
P47	#7921 to #7924
P48	#7941 to #7944

13. PROGRAMMING SUPPORT FUNCTIONS 13.7 User Macro Specifications

(9) Alarm (#3000)

The NC system can be forcibly set to the alarm state by using variable number 3000.

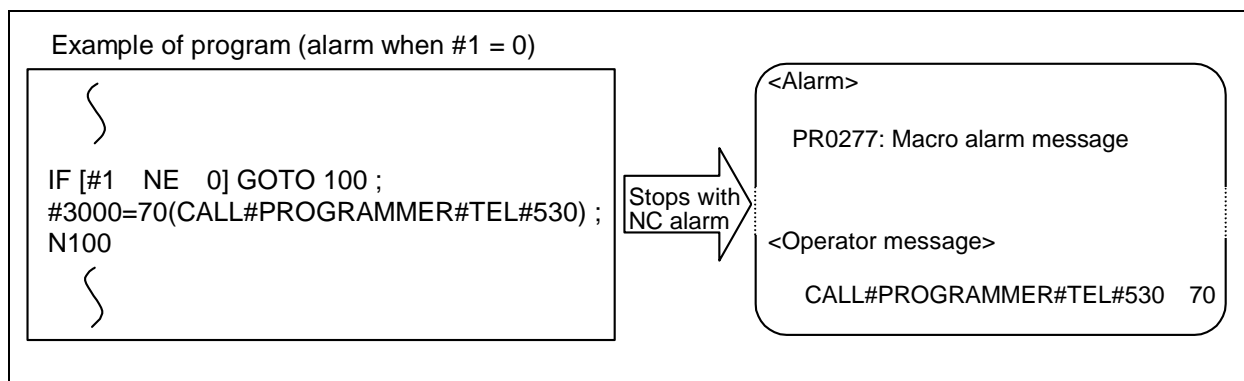
Command format



Any alarm number from 1 to 9999 can be specified.

The alarm message must be less than 31 characters long.

The "PR0277 Macro alarm message" appears in the <alarm> column on diagnosis screen 1 while the alarm number and alarm message CALL #PROGRAMMER #TEL#530 is indicated in the <operator message>.



Note 1) Alarm number 0 is not displayed and any number exceeding 9999 cannot be indicated.

Note 2) The characters following the first alphabet letter in the right member is treated as the alarm message. Therefore, a number cannot be designated as the first character of an alarm message. It is recommended that the alarm messages be enclosed in round parentheses.

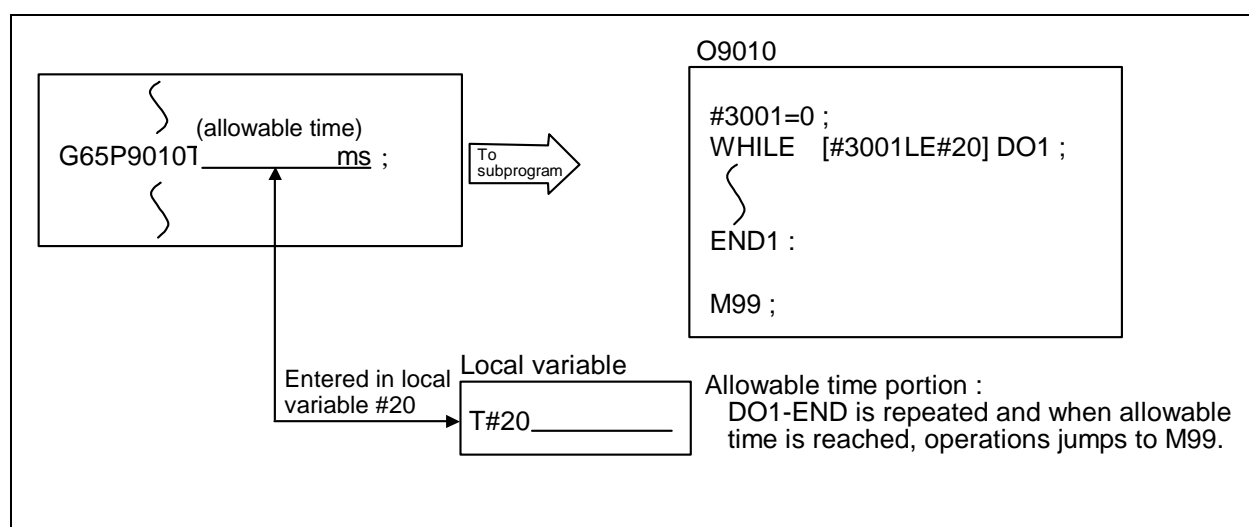
13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(10) Integrating (run-out) time (#3001, #3002)

The integrating (run-out) time can be read during automatic operation or automatic start or values can be substituted by using variable numbers 3001 and 3002.

Type	Variable number	Unit	Contents when power is turned ON	Initialization of contents	Count condition
Integrating (run-out) time 1	3001	1ms	Same as when power is turned OFF	Value substituted for variable	At all times while power is ON
Integrating (run-out) time 2	3002				In-automatic start

The integrating run time returns to zero in about 2.44×10^{11} ms (approximately 7.7 years).



(11) Suppression of single block stop and miscellaneous function finish signal waiting

By substituting the values below in variable number 3003, it is possible to suppress single block stop in the subsequent blocks or to advance to the next block without waiting for the miscellaneous function (M, S, T, B) finish (FIN) signal.

#3003	Single block stop	Miscellaneous function finish signal
0	Not suppressed	Awaited
1	Suppressed	Awaited
2	Not suppressed	Not awaited
3	Suppressed	Not awaited

Note 1) #3003 is cleared to zero by NC reset.

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(12) Feed hold, feedrate override, G09 valid/invalid

By substituting the values below in variable number 3004, it is possible to make the feed hold, feedrate override and G09 functions either valid or invalid in the subsequent blocks.

Contents (value) \ #3004	Bit 0	Bit 1	Bit 2
	Feed hold	Feedrate override	G09 check
0	Valid	Valid	Valid
1	Invalid	Valid	Valid
2	Valid	Invalid	Valid
3	Invalid	Invalid	Valid
4	Valid	Valid	Invalid
5	Invalid	Valid	Invalid
6	Valid	Invalid	Invalid
7	Invalid	Invalid	Invalid

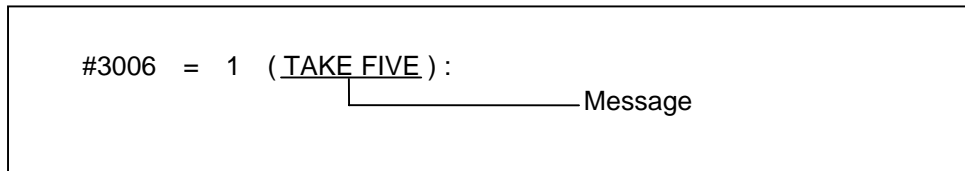
Note 1) Variable number #3004 is set to zero by NC reset.

Note 2) The functions are valid when the above bits are 0 and invalid when they are 1.

(13) Message display and stop

By using variable number 3006, the execution is stopped after the previous block has been executed and, if message display data have been commanded, then the corresponding message will be indicated.

Command format



The message should not be longer than 31 characters and it should be enclosed within round () parentheses.

(14) Mirror image

By reading variable number 3007, it is possible to ascertain the status of mirror image at a particular point in time for each axis.

The axes correspond to the bits of #3007. When the bits are 0, it means that the mirror image function is not valid; when they are 1, it means that it is valid.

#3007

Bit	15	14	13	12	11	10	9	8	7	6	5	4	3	2	1	0
nth axis													4	3	2	1

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(15) Miscellaneous function lock state

The miscellaneous function lock state can be seen by reading the variable number #3008.

The #3008 details indicate the following:

0: Miscellaneous function lock OFF

1: Miscellaneous function lock ON

The ON/OFF status of the miscellaneous function lock is determined by the miscellaneous function lock signal (Y75A) and miscellaneous function lock invalid (#3009).

Miscellaneous function lock signal (Y75A)	OFF	ON	OFF	ON
Miscellaneous function lock invalid (#3009)	OFF	OFF	ON	ON
Miscellaneous function lock state (#3008)	OFF	ON	OFF	OFF

(16) Miscellaneous function lock invalid

By substituting the following values in the variable number #3009, the miscellaneous function lock signal (Y75A) can be validated and invalidated in the following blocks.

Details	#3009	Bit 0
	Miscellaneous function lock	
0	Valid	
1	Invalid	

(Note) #3009 is set to 0 when reset.

(17) Clock function

The current time (year, month, date, hour, minute, second) can be read and written using the variable numbers #3011 and #3012.

Type	Variable number
Year, month, date	#3011
Hour, minute, second	#3012

Example) For 18 hours, 27 minutes, 52 seconds on May 14, 1992
#3011 = 19920514 ;
#3012 = 182752 ;

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(18) G command modals

Using variable numbers 4001 to 4021, it is possible to read the G modal commands which have been issued up to the block immediately before.
 Similarly, it is possible to read the modals in the block being executed with variable numbers #4201 to #4221.

Variable number		Function
Pre-read block	Execution block	
#4001	#4201	Interpolation mode : G00:0, G01:1, G02:2, G03:3
#4002	#4202	Plane selection : G17:17, G18:18, G19:19
#4003	#4203	Absolute/incremental : G90:90, G91:91
#4004	#4204	No variable No.
#4005	#4205	Feed designation : G94:94, G95:95
#4006	#4206	Inch/metric : G20:20, G21:21
#4007	#4207	Tool nose R compensation : G40:40, G41:41, G42:42
#4008	#4208	Tool length offset : G43:43, G44:44, G49:49
#4009	#4209	Fixed cycle : G80:80, G73~74:73~74, G76:76, G81~G89:81~89
#4010	#4210	Return level : G98:98, G99:99
#4011	#4211	Scaling : G50:50, G51:51
#4012	#4212	Workpiece coordinate system: G54~G59:54~59
#4013	#4213	Acceleration/deceleration: G61~G64:61~64
#4014	#4214	Macro modal call : G66:66, G66.1:66.1, G67:67
#4015	#4215	Normal line control : G40.1:40.1, G41.1:41.1, G42.1:42.1
#4016	#4216	Coordinate rotation : G68:68, G69:69
#4017	#4217	Constant surface speed control : G96:96, G97:97
#4018	#4218	
#4019	#4219	Mirror image : G50.1:50.1, G51.1:51.1
#4020	#4220	
#4021	#4221	

(Example)

```
G28 X0 Y0 Z0 ;
G90 G1 X100. F1000;
G91 G65 P300 X100. Y100.;
M02;
O300;
#1 = #4003; → Group 3G modal (pre-read) #1 = 91.0
#2 = #4203; → Group 3G modal (now being executed) #2 = 90.0
G#1X#24Y#25;
M99;
%
```


13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

(19) Other modals

Using variable numbers 4101 to 4120, it is possible to read the modal commands assigned up to the block immediately before.

Similarly, it is possible to read the modals in the block being executed with variable numbers #4301 to #4320.

Variable number		Modal information	Variable number		Modal information
Pre-read	Execution		Pre-read	Execution	
#4101	#4301		#4111	#4311	Tool length offset No.H
#4102	#4302		#4112	#4312	
#4103	#4303		#4113	#4313	Miscellaneous function M
#4104	#4304		#4114	#4314	Sequence number N
#4105	#4305		#4115	#4315	Program number O
#4106	#4306		#4116	#4316	
#4107	#4307	Tool radius compensation No. D	#4117	#4317	
#4108	#4308		#4118	#4318	
#4109	#4309	Feedrate F	#4119	#4319	Spindle function S
#4110	#4310		#4120	#4320	Tool function T

(20) Position information

Using variable numbers 5001 to 5104, it is possible to read the servo deviation amounts, tool position offset amount, skip coordinates, workpiece coordinates, machine coordinates and end point coordinates in the block immediately before.

Position information / Axis No.	End point coordinate of block immediately before	Machine coordinate	Workpiece coordinate	Skip coordinate	Tool position offset amount	Servo deviation amount
1	#5001	#5021	#5041	#5061	#5081	#5101
2	#5002	#5022	#5042	#5062	#5082	#5102
3	#5003	#5023	#5043	#5063	#5083	#5103
4	#5004	#5024	#5044	#5064	#5084	#5104
Remarks (reading during movement)	Yes	No	No	Yes	No	Yes

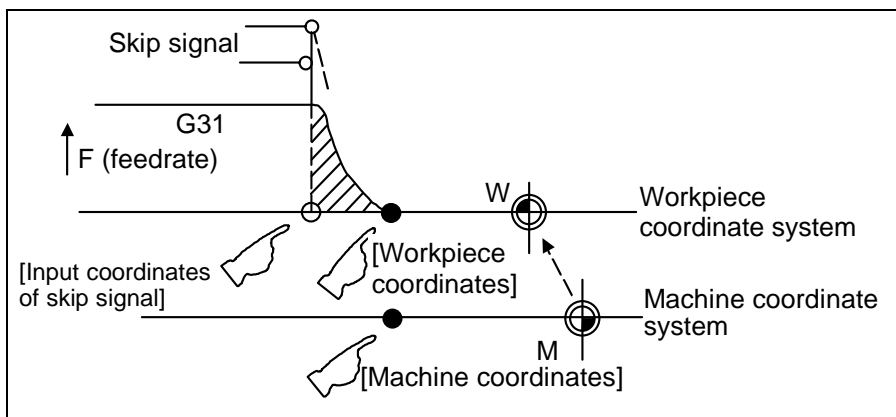
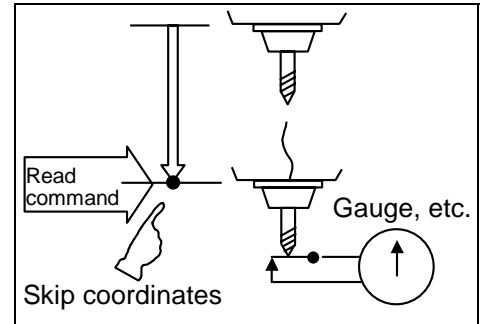
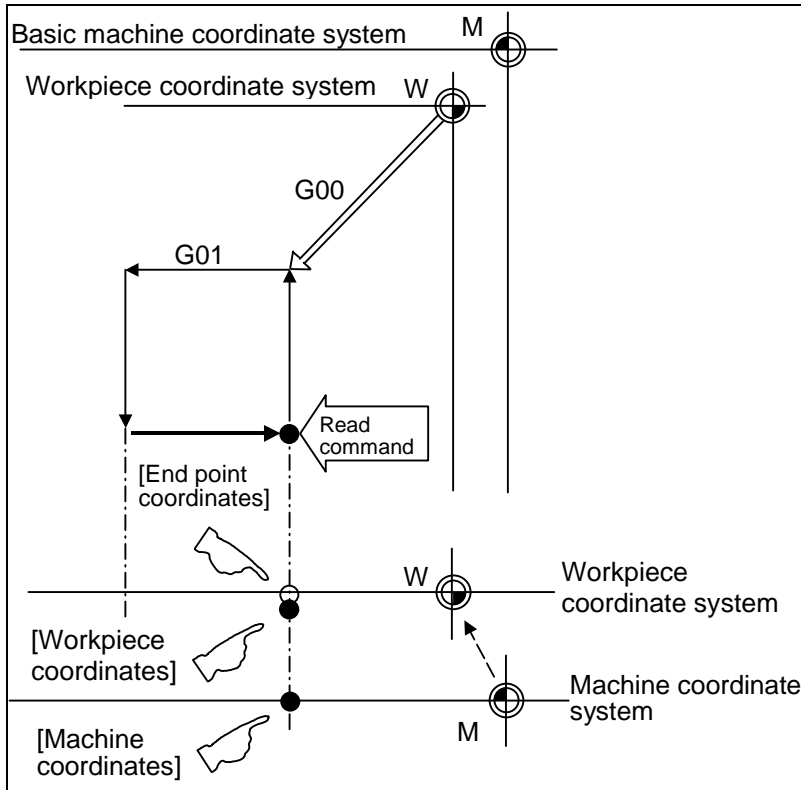
Note 1) The number of axes which can be controlled differs according to the specifications.

Note 2) The tool position offset position is the value obtained with only the G45 to G48 position compensation amount.

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

- 1) The positions of the end point coordinates and skip coordinates are positions in the workpiece coordinate system.
- 2) The end point coordinates, skip coordinates and servo deviation amounts can be read even during movement. However, it must first be checked that movement has stopped before reading the machine coordinates and the workpiece coordinates.
- 3) The position where the skip signal is turned ON in the G31 block is indicated for the skip coordinates. The end point position is indicated when the skip signal has not been turned ON. (For further details, refer to the section on tool length measurement.)
- 4) The tool nose position where the tool offset and other such factors are not considered is indicated as the end point position. The tool reference point position with consideration given to tool offset is indicated for the machine coordinates, workpiece coordinates and skip coordinates.



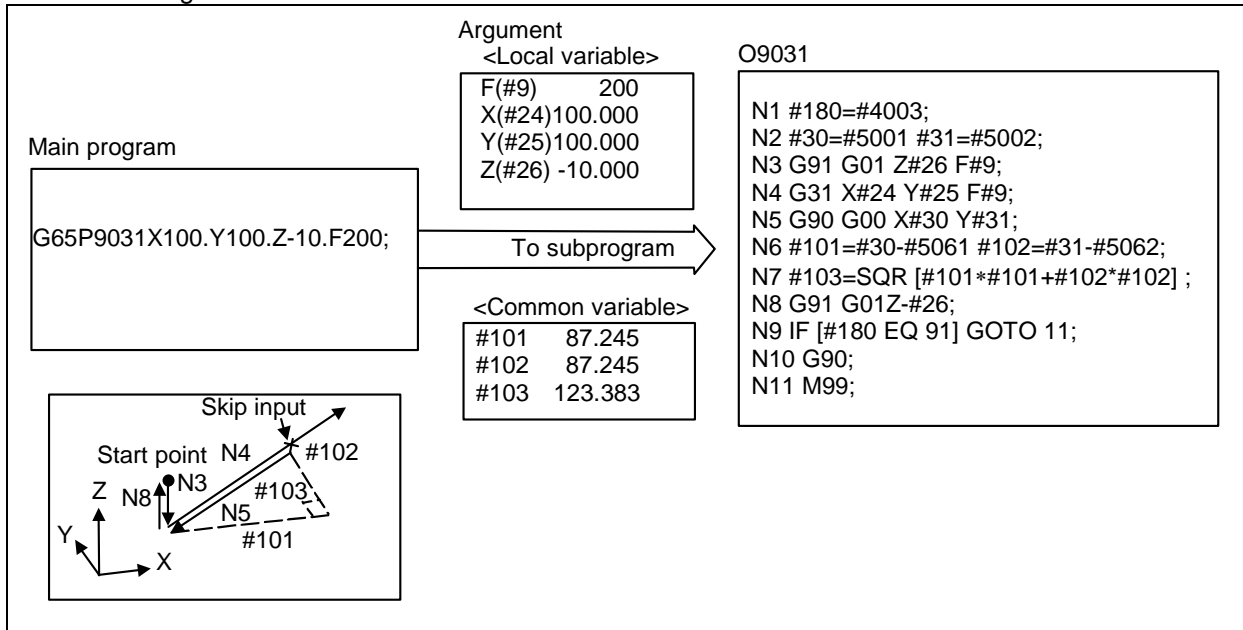
For "●", check stop and then proceed to read.
 For "○", reading is possible during movement.

The position of the skip signal input coordinates is the position in the workpiece coordinate system. The coordinates in variable numbers #5061 to #5064 memorize the moments when the skip input signal during movement was input and so they can be read at any subsequent time. For further details, reference should be made to the section on the skip function.

13. PROGRAMMING SUPPORT FUNCTIONS 13.7 User Macro Specifications

(Example 1) Example of workpiece position measurement

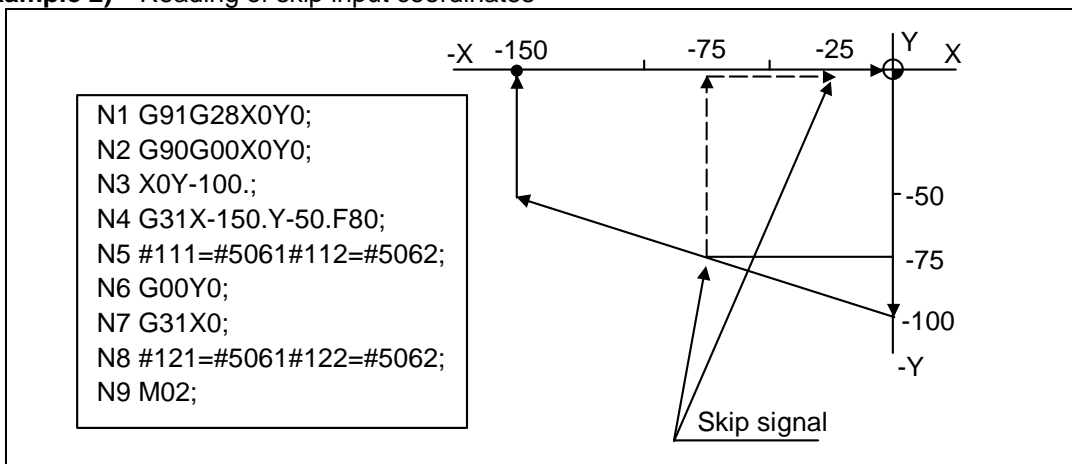
An example to measure the distance from the measured reference point to the workpiece edge is shown below.



- | | |
|--|---|
| <p>#101 X axis measurement amount
#102 X axis measurement amount
#103 Measurement linear segment amount
#5001 X axis measurement start point
#5002 Y axis measurement start point

#5061 X axis skip input point
#5062 Y axis skip input point</p> | <p>N1 G90/G91 modal recording
N2 X, Y start point recording
N3 Z axis entry amount
N4 X, Y measurement (Stop at skip input)
N5 Return to X, Y start point
N6 X, Y measurement incremental value calculation
N7 Measurement linear segment calculation
N8 Z axis escape
N9, N10 G90/G91 modal return
N11 Subprogram return</p> |
|--|---|

(Example 2) Reading of skip input coordinates



$\#111 = -75. + \epsilon$	$\#112 = -75. + \epsilon$
$\#121 = -25. + \epsilon$	$\#122 = -75. + \epsilon$

ϵ is the error caused by response delay. (Refer to the section on the skip function for details.)
#122 is the N4 skip signal input coordinates as there is no Y command at N7.

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(21) Mirror image

The external, parameter, G command mirror image valid axis and each axis center coordinate value can be read with variable numbers #30000 to #30056.

Variable No.	Details
#30000	External mirror image valid axis bit 0: 1st axis external mirror image valid axis bit 1: 2nd axis external mirror image valid axis . . bit 15: 16th axis external mirror image valid axis
#30001	1st axis external mirror image center coordinate value
#30002	2nd axis external mirror image center coordinate value
.	.
.	.
#30016	16th axis external mirror image center coordinate value
#30020	Parameter mirror image valid axis bit 0: 1st axis parameter mirror image valid axis bit 1: 2nd axis parameter mirror image valid axis . . bit 15: 16th axis parameter mirror image valid axis
#30021	1st axis parameter mirror image center coordinate value
#30022	2nd axis parameter mirror image center coordinate value
.	.
.	.
#30036	16th axis parameter mirror image center coordinate value
#30040	G command mirror image valid axis bit 0: 1st axis G command mirror image valid axis bit 1: 2nd axis G command mirror image valid axis . . bit15: 16th axis G command mirror image valid axis
#30041	1st axis G command mirror image center coordinate value
#30042	2nd axis G command mirror image center coordinate value
.	.
.	.
#30056	16th axis G command mirror image center coordinate value

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(22) Coordinate rotation

The parameter and program coordinate rotation information can be read using the variable numbers #30060 to 30076.

Variable No.	Details
#30060	Parameter coordinate rotation center horizontal axis No.
#30061	Parameter coordinate rotation center vertical axis No.
#30062	Parameter coordinate rotation center coordinate value horizontal axis
#30063	Parameter coordinate rotation center coordinate value vertical axis
#30064	Parameter coordinate rotation angle
#30065	Parameter coordinate rotation SIN data
#30066	Parameter coordinate rotation COS data
#30067	Parameter coordinate rotation horizontal axis length
#30068	Parameter coordinate rotation vertical axis length
#30070	Program coordinate rotation center horizontal axis No.
#30071	Program coordinate rotation center vertical axis No.
#30072	Program coordinate rotation center coordinate value horizontal axis
#30073	Program coordinate rotation center coordinate value vertical axis
#30074	Program coordinate rotation angle
#30075	Program coordinate rotation SIN data
#30076	Program coordinate rotation COS data

(23) Scaling

The scaling information can be read using the variable numbers #30080 to #30097

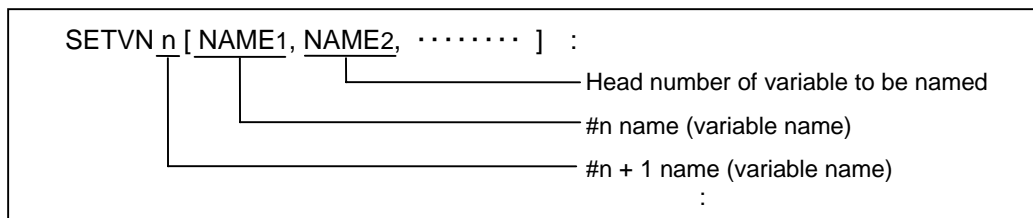
Variable No.	Details
#30080	Scaling valid axis bit 0: 1st axis scaling valid bit 2: 2nd axis scaling valid . . bit 15: 16th axis scaling valid
#30081	1st axis scaling center coordinate value
#30082	2nd axis scaling center coordinate value
.	.
.	.
#30096	16th axis scaling center coordinate value
#30097	Scaling magnification

13. PROGRAMMING SUPPORT FUNCTIONS 13.7 User Macro Specifications

(24) Variable name setting and quotation

Any name (variable name) can be given to common variables #500 to #519. It must be composed of not more than 7 alphanumeric characters and it must begin with a letter. Do not use "#" in variable names. It causes an alarm when the program is executed.

Command format



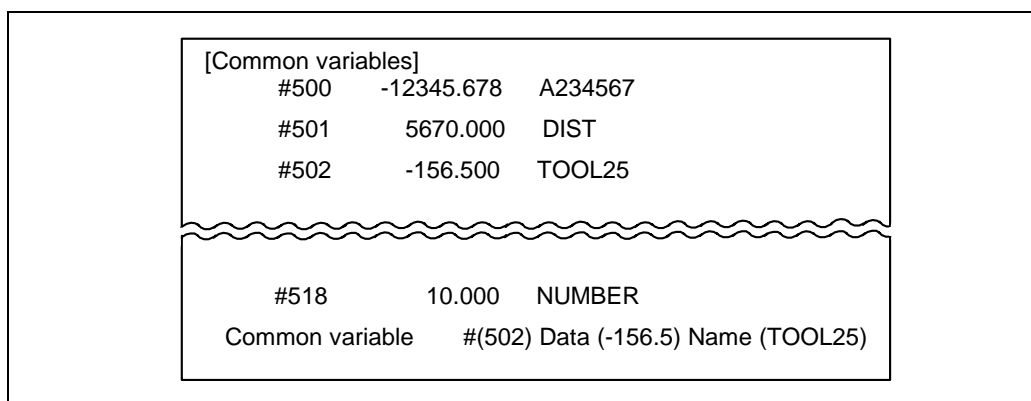
Variable names are separated by a comma (,).

- (a) Once variable names have been set, they will not be cleared even when the power is turned OFF.
- (b) Variables in programs can be quoted by their variable names. In cases like this, the variables should be enclosed in square parentheses.

(Example 1) G01X [#POINT1] ;
[#NUMBER] = 25 ;

- (c) The variable numbers, data and variable names appear on the screen of the setting and display unit.

(Example 2) Program SETVN500 [A234567, DIST, TOOL25] ;



13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

13.7.5 Arithmetic commands

A variety of arithmetic operations can be performed between variables.

$\#i = \langle \text{formula} \rangle$

$\langle \text{Formula} \rangle$ is a combination of constants, variables, functions and operators.

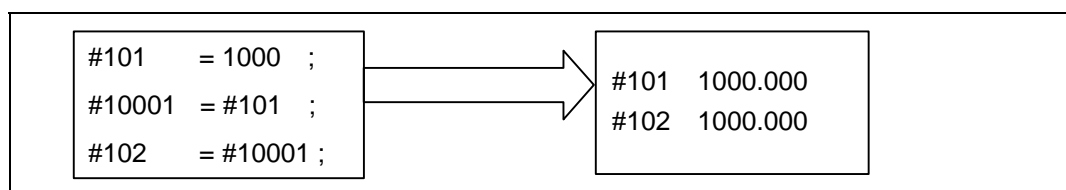
Constants can be used instead of $\#j$ and $\#k$ below.

(1) Definition and substitution of variables	$\#i = \#j$	Definition, substitution
(2) Addition arithmetic	$\#i = \#j + \#k$ $\#i = \#j - \#k$ $\#i = \#j \text{ OR } \#k$ $\#i = \#j \text{ XOR } \#k$	Addition Subtraction Logical sum (at every bit of 32 bits) Exclusive OR (at every bit of 32 bits)
(3) Multiplication arithmetic	$\#i = \#j * \#k$ $\#i = \#j / \#k$ $\#i = \#j \text{ MOD } \#k$ $\#i = \#j \text{ AND } \#k$	Multiplication Division Remainder Logical product (at every bit of 32 bits)
(4) Functions	$\#i = \text{SIN } [\#k]$ $\#i = \text{COS } [\#k]$ $\#i = \text{TAN } [\#k]$ $\#i = \text{ATAN } [\#j]$ $\#i = \text{ACOS } [\#j]$ $\#i = \text{SQRT } [\#k]$ $\#i = \text{ABS } [\#k]$ $\#i = \text{BIN } [\#k]$ $\#i = \text{BCD } [\#k]$ $\#i = \text{ROUND } [\#k]$ $\#i = \text{FIX } [\#k]$ $\#i = \text{FUP } [\#k]$ $\#i = \text{LN } [\#k]$ $\#i = \text{EXP } [\#k]$	Sine Cosine Tangent (sin/cos used for tan) Arctangent (ATAN or ATN may be used) Arc-cosine Square root (SQRT or SQR may be used) Absolute value Conversion from BCD to BIN Conversion from BIN to BCD Rounding off (ROUND or RND may be used) Discard fractions less than 1 Add for fractions less than 1 Natural logarithm Exponent with e (=2.718) as bottom

Note 1) A value without a decimal point is basically treated as a value with a decimal point at the end (1 = 1.000).

Note 2) Offset amounts from #10001 and workpiece coordinate system offset values from #5201 are handled as data with a decimal point. Consequently, data with a decimal point will be produced even when data without a decimal point have been defined in the variable numbers.

(Example)



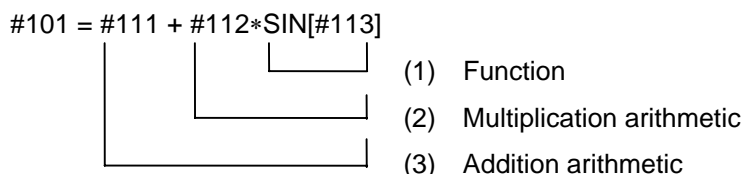
Note 3) The $\langle \text{formula} \rangle$ after a function must be enclosed in the square parentheses.

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

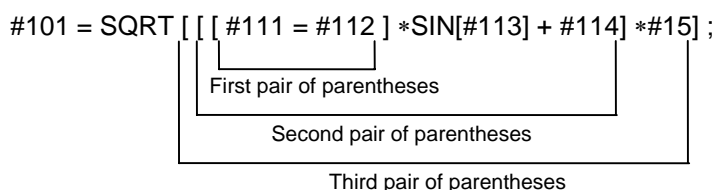
(1) Sequence of arithmetic operations

The sequence of the arithmetic operations (1) to (3) is, respectively, the functions followed by the multiplication arithmetic followed in turn by the addition arithmetic.



(2) Designation of arithmetic operation sequence

The part to be given priority in the operation sequence should be enclosed in square parentheses. Up to 5 pairs of such parentheses including those for the functions may be used.



(3) Examples of arithmetic commands

(1) Main Program and argument designation	G65 P100 A10 B20. ; #101 = 100.000 #102 = 200.000 ;	#1 10.0000 #2 20.0000 #101 100.0000 #102 200.0000	
(2) Definition and substitution (=)	#1 = 1000 #2 = 1000. #3 = #101 #4 = #102 #5 = #5041	#1 1000.0000 #2 1000.0000 #3 100.0000 #4 200.0000 #5 -10.0000	} From common variables From offset amount
(3) Addition and subtraction (+,-)	#11 = #1 + 1000 #12 = #2 - 50. #13 = #101 + #1 #14 = #5041 - 3. #15 = #5041 + #102	#11 2000.0000 #12 950.0000 #13 1100.0000 #14 -13.0000 #15 190.0000	
(4) Logical sum (OR)	#3 = 100 #4 = #3OR14	#3 = 01100100 14 = 00001110 #4 = 01101110 = 110	
(5) Exclusive OR (XOR)	#3 = 100 #4 = #3XOR14	#3 = 01100100 14 = 00001110 #4 = 01101010 = 106	

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(6) Multiplication and division (*,/)	#21 = 100*100 #22 = 100.*100 #23 = 100*100 #24 = 100.*100. #25 = 100/100 #26 = 100./100. #27 = 100/100. #28 = 100./100. #29 = #5041*#101 #30 = #5041/#102	#21 10000.0000 #22 10000.0000 #23 10000.0000 #24 10000.0000 #25 1.0000 #26 1.0000 #27 1.0000 #28 1.0000 #29 -1000.0000 #30 -0.0500
Remainder (MOD)	#31 = #19MOD#20	#19/#20 = 48/9 = 5 with 3 over
(7) Logical product (AND)	#9 = 100 #10 = #9AND15	#9 = 01100100 15 = 00001111 #10 = 00000100 = 4
(8) Sin (SIN)	#501 = SIN [60] #502 = SIN [60.] #503 = 1000*SIN [60] #504 = 1000*SIN [60.] #505 = 1000.*SIN [60] #506 = 1000.*SIN [60.] Note: SIN [60] is equivalent to SIN [60.]	#501 0.860 #502 0.860 #503 866.0254 #504 866.0254 #505 866.0254 #506 866.0254
(9) Cosine (COS)	#541 = COS [45] #542 = COS [45.] #543 = 1000*COS [45] #544 = 1000*COS [45.] #545 = 1000.*COS [45] #546 = 1000.*COS [45.] Note: COS [45] is equivalent to COS [45.]	#541 0.7071 #542 0.7071 #543 707.1068 #544 707.1068 #545 707.1068 #546 707.1068
(10) Tangent (TAN)	#551 = TAN [60] #552 = TAN [60.] #553 = 1000*TAN [60] #554 = 1000*TAN [60.] #555 = 1000.*TAN [60] #556 = 1000.*TAN [60.] Note: TAN [60] is equivalent to TAN [60.]	#551 1.7321 #552 1.7321 #553 1732.0508 #554 1732.0508 #555 1732.0508 #556 1732.0508
(11) Arctangent (ATAN or ATN)	#561 = ATAN [173205/100000] #562 = ATAN [173205/100.] #563 = ATAN [173.205/100000] #564 = ATAN [173.205/100.] #565 = ATAN [1.732]	#561 60.0000 #562 60.0000 #563 60.0000 #564 60.0000 #565 59.9993
(12) Arc-cosine (ACOS)	#521 = ACOS [100./141.421] #522 = ACOS [100./141.421] #523 = ACOS [1000./1414.213] #524 = ACOS [10./14.142] #525 = ACOS [0.707]	#521 44.9999 #522 44.9999 #523 45.0000 #524 44.9995 #525 45.0087

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(13) Square root (SQR or SQRT)	#571 = SQRT [1000] #572 = SQRT [1000.] #573 = SQRT [10.*10.+20.*20.] #574 = SQRT[#14*#14+#15*#15] Note: In order to increase the accuracy, proceed with the operation inside parentheses.	#571 31.6228 #572 31.6228 #573 22.3607 #574 190.4442
(14) Absolute value (ABS)	#576 = -1000 #577 = ABS [#576] #3 = 70.#4 = -50. #580 = ABS [#4 - #3]	#576 -1000.0000 #577 1000.0000 #580 120.0000
(15) BIN, BCD	#1 = 100 #11 = BIN [#1] #12 = BCD [#1]	#11 64.0000 #12 256.0000
(16) Rounding off (ROUND or RND)	#21 = ROUND [14/3] #22 = ROUND [14./3] #23 = ROUND [14/3.] #24 = ROUND [14./3.] #25 = ROUND [-14/3] #26 = ROUND [-14./3] #27 = ROUND [-14/3.] #28 = ROUND [-14./3.]	#21 5.0000 #22 5.0000 #23 5.0000 #24 5.0000 #25 -5.0000 #26 -5.0000 #27 -5.0000 #28 -5.0000
(17) Discarding fractions below decimal point (FIX)	#21 = FIX [14/3] #22 = FIX [14./3] #23 = FIX [14/3.] #24 = FIX[14./3.] #25 = FIX [-14/3] #26 = FIX [-14./3] #27 = FIX [-14/3.] #28 = FIX [-14./3.]	#21 4.0000 #22 4.0000 #23 4.0000 #24 4.0000 #25 -4.0000 #26 -4.0000 #27 -4.0000 #28 -4.0000
(18) Adding fractions less than 1 (FUP)	#21 = FUP [14/3] #22 = FUP [14./3] #23 = FUP [14/3.] #24 = FUP [14./3.] #25 = FUP [-14/3] #26 = FUP [-14./3] #27 = FUP [-14/3.] #28 = FUP [-14./3.]	#21 5.0000 #22 5.0000 #23 5.0000 #24 5.0000 #25 -5.0000 #26 -5.0000 #27 -5.0000 #28 -5.0000
(19) Natural logarithms (LN)	#101 = LN [5] #102 = LN [0.5] #103 = LN [-5]	#101 1.6094 #102 -0.6931 Error "PR0282"
(20) Exponents (EXP)	#104 = EXP [2] #105 = EXP [1] #106 = EXP [-2]	#104 7.3891 #105 2.7183 #106 0.1353

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

13.7.6 Control commands

The flow of programs can be controlled by IF-GOTO- and WHILE-DO-.

(1) Branching

Format : IF [conditional expression] GOTO n; (n = sequence number in the program)

When the condition is satisfied, control branches to "n" and when it is not satisfied, the next block is executed.

IF [conditional expression] can be omitted and, when it is, control passes to "n" unconditionally.

The following types of [conditional expressions] are available.

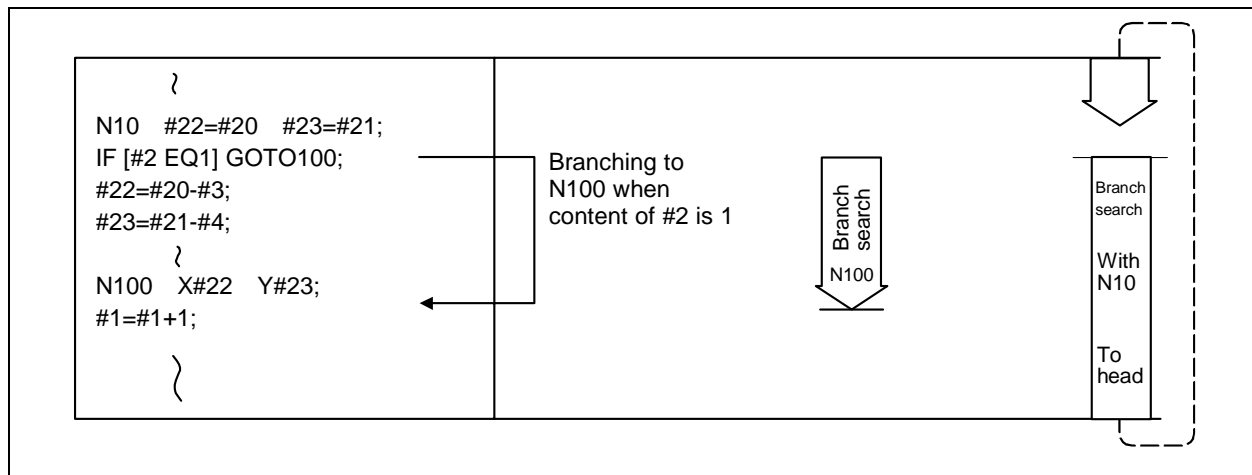
#i EQ #j	=	When #i and #j are equal
#i NE #j	≠	When #i and #j are not equal
#i GT #j	>	When #i is greater than #j
#i LT #j	<	When #i is less than #j
#i GE #j	≥	When #i is #j or more
#i LE #j	≤	When #i is #j or less

"n" of GOTO n must always be in the same program. Alarm "PR0231 No sequence No." will result if it is not. A formula or variable can be used instead of #i, #j and "n".

In the block with sequence number "n" which will be executed after a GOTO n command, the sequence number must always be at the head of the block.

Otherwise, alarm "PR0231 No sequence No." will result.

If "/" is at the head of the block and Nn follows, control can be branched to the sequence number.



Note 1) When the sequence number of the branch destination is searched, the search is conducted up to the end of the program (% code) from the block following IF ... ; and if it is not found, it is then conducted from the top of the program to the block before IF ... ;. Therefore, branch searches in the opposite direction to the program flow will take longer to execute compared with branch searches in the forward direction.

Note 2) EQ and NE should be used only for integers. For comparison of numeric values with decimals, GE, GT, LE, and LT should be used.

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

(2) Iteration

Format : WHILE [conditional expression] DOm ; (m = 1, 2, 3 127)
 }
 END m ;

While the conditional expression is established, the blocks from the following block to ENDM are repeatedly executed; when it is not established, execution moves to the block after ENDM. DOm may come before WHILE, WHILE [conditional expression] DOm and ENDM must be used as a pair. IF WHILE [conditional expression] is omitted, these blocks will be repeatedly ad infinitum. The repeating identification numbers range from 1 to 127 (DO1, DO2, DO3, DO127). Up to 27 nesting levels can be used.

<p>(1) Same identifier number can be used any number of times.</p> <div style="margin-top: 20px;"> <p>Possible</p> <pre style="border: 1px solid black; padding: 5px; width: fit-content; margin: 5px auto;">WHILE ~ DO1 ; } END1 ;</pre> <p>Possible</p> <pre style="border: 1px solid black; padding: 5px; width: fit-content; margin: 5px auto;">WHILE ~ DO1 ; } END1 ;</pre> </div>	<p>(2) Any number may be used for the WHILE –DOm identifier number.</p> <div style="margin-top: 20px;"> <pre style="border: 1px solid black; padding: 5px; width: fit-content; margin: 5px auto;">WHILE ~ DO1 ; } END1 ;</pre> <p style="text-align: center;">}</p> <pre style="border: 1px dashed black; padding: 5px; width: fit-content; margin: 5px auto;">WHILE ~ DO3 ; } END3 ;</pre> <p>Possible</p> <p style="text-align: center;">}</p> <pre style="border: 1px dashed black; padding: 5px; width: fit-content; margin: 5px auto;">WHILE ~ DO2 ; } END2 ;</pre> <p style="text-align: center;">}</p> <pre style="border: 1px dashed black; padding: 5px; width: fit-content; margin: 5px auto;">WHILE ~ DO1 ; } END1 ;</pre> </div>
<p>(3) Up to 27 nesting levels for WHILE – DOm. "m" is any number from 1 to 127 for the nesting depth.</p> <div style="margin-top: 20px;"> <p>Possible</p> </div> <p>Note : With nesting, "m" which has been used once cannot be used.</p>	<p>(4) The number of WHILE – DOm nesting levels cannot exceed 27.</p> <div style="margin-top: 20px;"> <p style="text-align: right;">Not possible</p> </div>

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

<p>(5) WHILE – DOm must be designated first and ENDm last.</p> <div style="text-align: center;"> <p>Not possible</p> </div>	<p>(6) WHILE – DOm and ENDm must correspond on a 1:1 (pairing) basis in the same program.</p> <div style="text-align: center;"> <p>Not possible</p> </div>
<p>(7) Two WHILE – DOm's must not overlap.</p> <div style="text-align: center;"> <p>Not possible</p> </div>	<p>(8) Branching externally is possible from the WHILE – DOm range.</p> <div style="text-align: center;"> <p>Possible</p> </div>
<p>(9) No branching is possible inside WHILE – DOm.</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;"> <p>Not possible</p> </div> <div style="text-align: center;"> <p>Not possible</p> </div> </div>	<p>(10) Subprograms can be called by M98, G65 or G66 between WHILE – DOm's.</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;"> <p>Main program</p> </div> <div style="text-align: center;"> <p>Subprogram</p> </div> </div> <p style="text-align: center;">Possible</p>
<p>(11) Calls can be initiated by G65 or G66 between WHILE – DOm's and commands can be issued again from 1. Up to 27 nesting levels are possible for the main program and subprograms.</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;"> <p>Main program</p> </div> <div style="text-align: center;"> <p>Subprogram</p> </div> </div> <p style="text-align: center;">Possible</p>	<p>(12) A program error will occur at M99 if WHILE and END are not paired in the subprogram (including macro subprogram).</p> <div style="display: flex; justify-content: space-around;"> <div style="text-align: center;"> <p>Main program</p> </div> <div style="text-align: center;"> <p>Subprogram</p> <p>Don ENDn constitutes illegal usage.</p> </div> </div>

Note) As the fixed cycles G73 and G83 and the special fixed cycle G34 use WHILE, these will be added multiple times.

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

13.7.7 External output commands

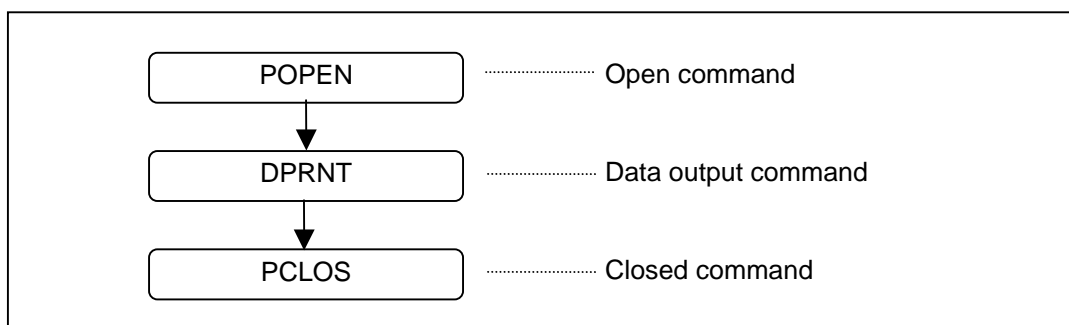
(1) Outline

The external output commands are designed to output the variable values or characters via the RS-232C interface.

(a) Types

- POPEN For preparing the processing of data outputs
- PCLOS For terminating the processing of data outputs
- BPRNT For character output and variable value binary output
- DPRNT For character output and digit-by-digit variable numerical output

(b) Command sequence



(2) Open command : POPEN

Command format : POPEN ;

- (a) The command is issued before the series of data output commands.
- (b) The DC2 control code and % code are output from the NC system to the external output device.
- (c) Once POPEN; has been issued, it will remain valid until PCLOS; is issued.

(3) Close command : PCLOS

Command format : PCLOS ;

- (a) This command is issued when all the data outputs are completed.
- (b) The DC4 control code and % code are output from the NC unit to the external output device.
- (c) This command is used together with the open command and it should not be issued unless the open mode has been established.
- (d) Issue the close command at the end of the program even when operation has been suspended by resetting or some other operation during data output.

13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

(4) Data output command **BPRNT**

Command format

BPRNT [l 1 # v 1 [c 1] l 2 # v 2 [c 2] ·····]

l_ : Character string

v_ : Variable number

c_ : Significant digits below decimal point

- (a) Character output and binary output of variable values are executed.
- (b) The designated character string is output as ISO codes.
 - Alphanumeric characters (A to Z, 0 to 9) and special characters (+, -, *, /) can be used.
 - Note that asterisk (*) is output as a space code.
- (c) All variables are saved with decimal points, so the number of digits required after the decimal point are commanded in brackets []. The variable value is handled as 4-byte (32-bit) data, and is output as binary data from the high-order byte. - (minus) data is handled as the complement of this data.

(Example) If three digits are designated for 12.3456, the fourth digit after the decimal point will be rounded up, and $[12.346 \times 10^3] = 12346$ (0000303A) will be output as binary data.

(Example) When zero digits is designated with -100.0, -100 (FFFFFF9C) will be output as binary data.
- (d) After the designated data is output, the EOB code will be output as an ISO code.
- (e) A <blank> variable will be interpreted as 0.

(5) Data output command : **DPRNT**

Command format

DPRNT [l 1 # v 1 [d 1 c 1] l 2 # v 2 [d 2 c 2] ·····]

l_ : Character string

v_ : Variable number

d_ : Significant digits above decimal point

c_ : Significant digits below decimal point $c + d \leq 8$

- (a) The character output and decimal output of the variable values are done with ISO codes.
- (b) The commanded character string is output as is by the ISO code.

Alphanumerics (A ~ Z, 0 ~ 9) and special characters (+, -, *, /) can be used.
- (c) The required significant digits above and below the decimal point in the variable values are commanded within square parentheses. As a result, the variable values equivalent to the commanded number of digits including the decimal point are output in ISO code in decimal notation from the high-order digits. Trailing zeroes are not omitted.
- (d) Leading zeroes are suppressed.

The leading zeroes can also be replaced by blank if so specified with a parameter (#110090 DPRINT leading 0). This can justify printed data on the last column.

13. PROGRAMMING SUPPORT FUNCTIONS

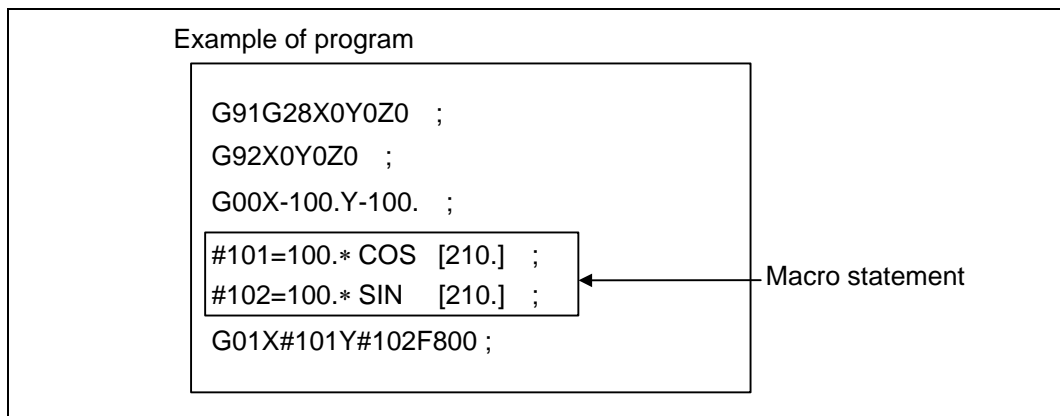
13.7 User Macro Specifications

13.7.8 Checkpoints

When the user macro commands are employed, it is possible to use the M, S, T and other NC control commands together with the arithmetic, decision, branching and other macro commands for preparing the machining programs. When the former commands are made into executable statements and the latter commands into macro statements, the macro statement processing should be accomplished as quickly as possible in order to minimize the machining time, because such processing is not directly related to machine control.

As a result, the parameter "#320002 macro single" can be set and the macro statements can be processed in parallel with the execution of the executable statement.

(The parameter can be set OFF during normal machining to process all the macro statements together or set ON during a program check to execute the macro statements block by block. This enables the setting to be made in accordance with the intended objective in mind.)

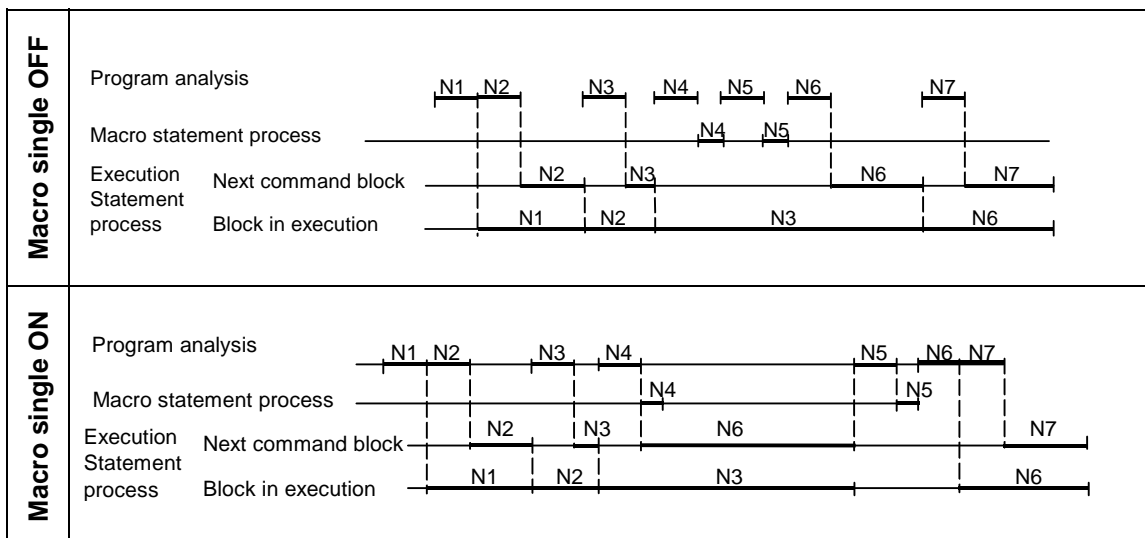


Macro statements are:

1. Arithmetic commands (block including =)
2. Control commands (block including GOTO, DO-END, etc.)
3. Macro call commands (including macro calls based on G codes and cancel command (G65, G66, G66.1, G67))

Executable statements are all those statements which are not macro statements.

Flow of processing



13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

Machining program display

Macro single OFF	<p>Macro single ON</p> <div style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <pre>[In execution] N3 G00 X-100. Y-100. ; [Next command]N6 G01 X#101 Y#102 F800 ;</pre> </div>	<p>N4, N5 and N6 are processed in parallel with the control of the executable statement of N3, N6 is an executable statement and so it is displayed as the next command. If the N4, N5 and N6 analysis is in time during N3 control, the machine movement will be continuously controlled.</p>
Macro single ON	<p>Macro single OFF</p> <div style="border: 1px solid black; padding: 5px; margin: 10px 0;"> <pre>[In execution] N3 G00 X-100. Y-100. ; [Next command] N4 #101=100. *COG [210.] ;</pre> </div>	<p>N4 is processed in parallel with the control of the NC executable statement of N3, and it is displayed as the next command. N5 and N6 is executed after N3 has finished, and so the machine control is held on standby during the N5 and N6 analysis time.</p>

13. PROGRAMMING SUPPORT FUNCTIONS
13.7 User Macro Specifications

13.7.9 Actual examples of using user macros

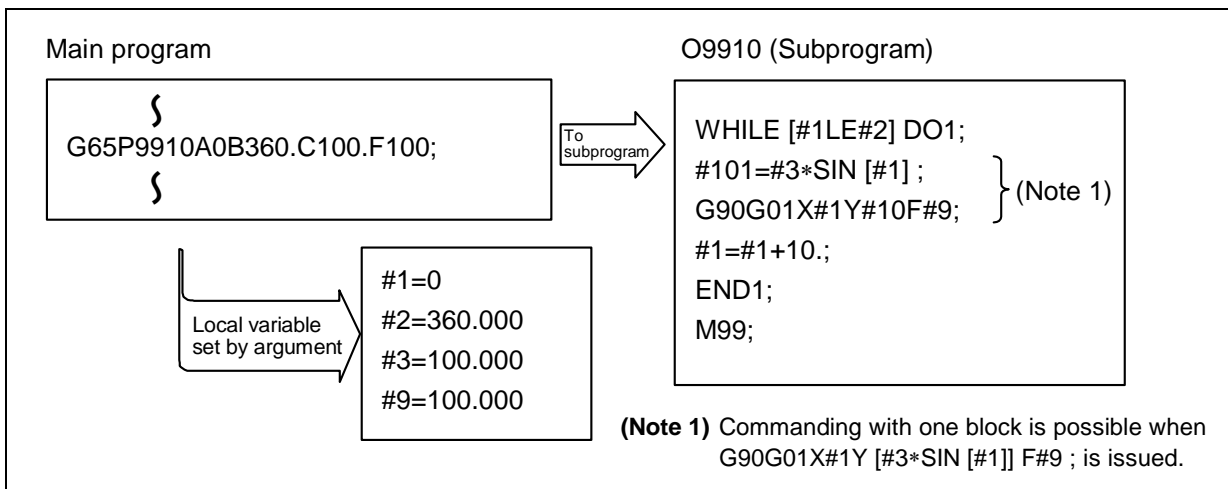
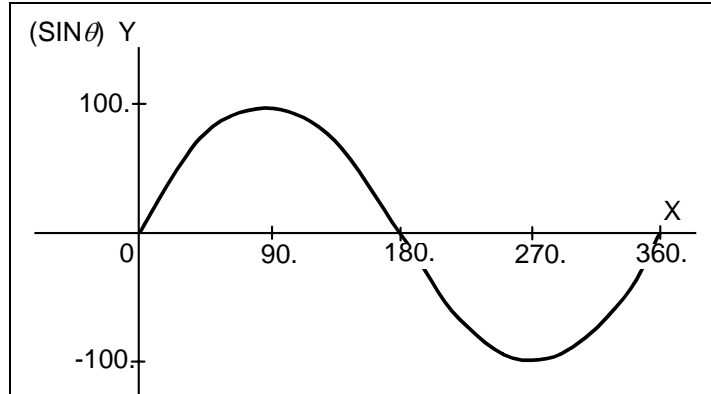
The following three examples will be described.

- Example 1. SIN curve
- Example 2. Bolt hole circle
- Example 3. Grid

(Example 1) SIN curve

```
G65 Pp1 Aa1 Bb1 Cc1 Ff1 ;

a1 ; Initial value 0°
b1 ; Final value 360°
c1 ; R of R*SINθ
f1 ; Feedrate
```

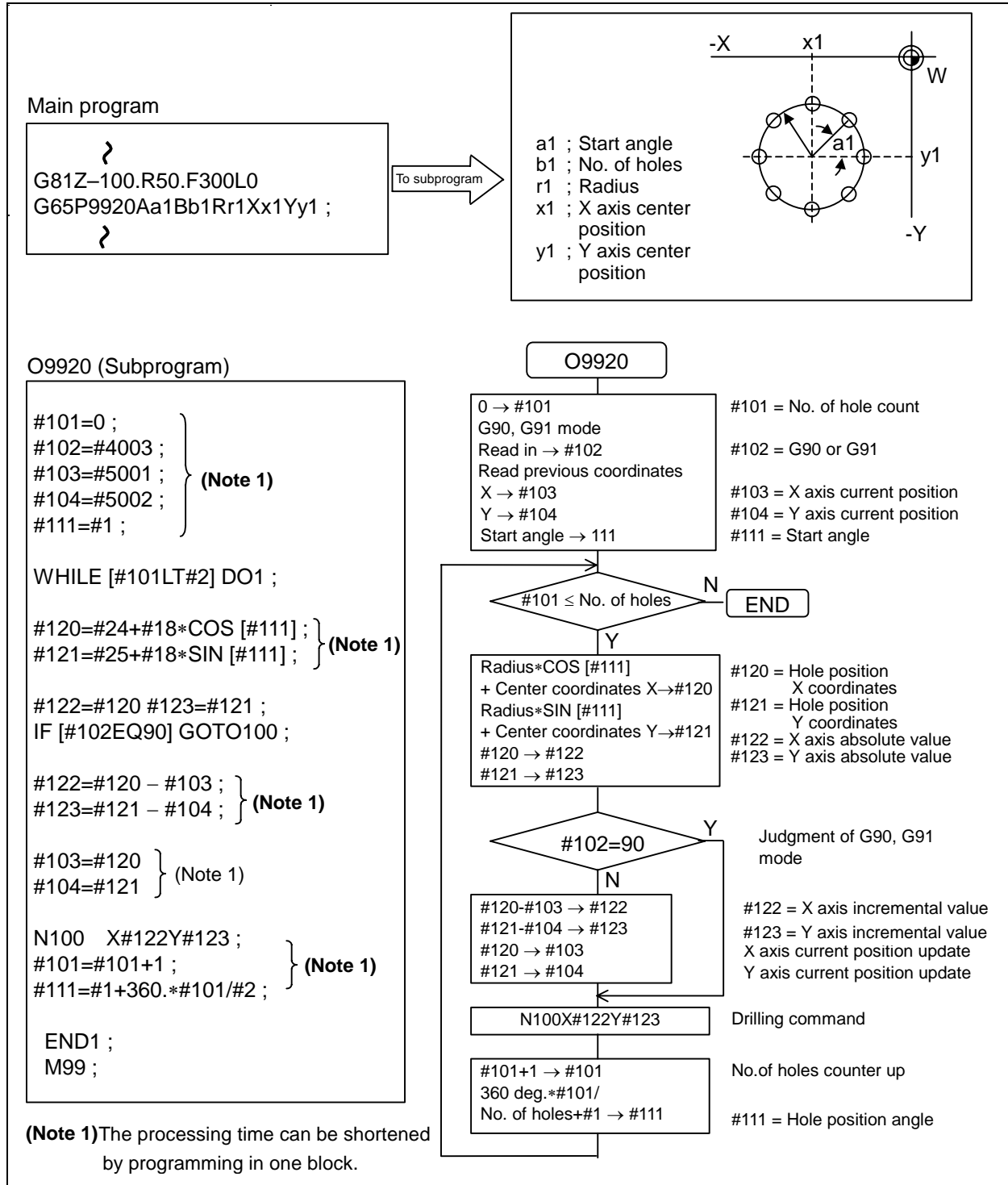


13. PROGRAMMING SUPPORT FUNCTIONS

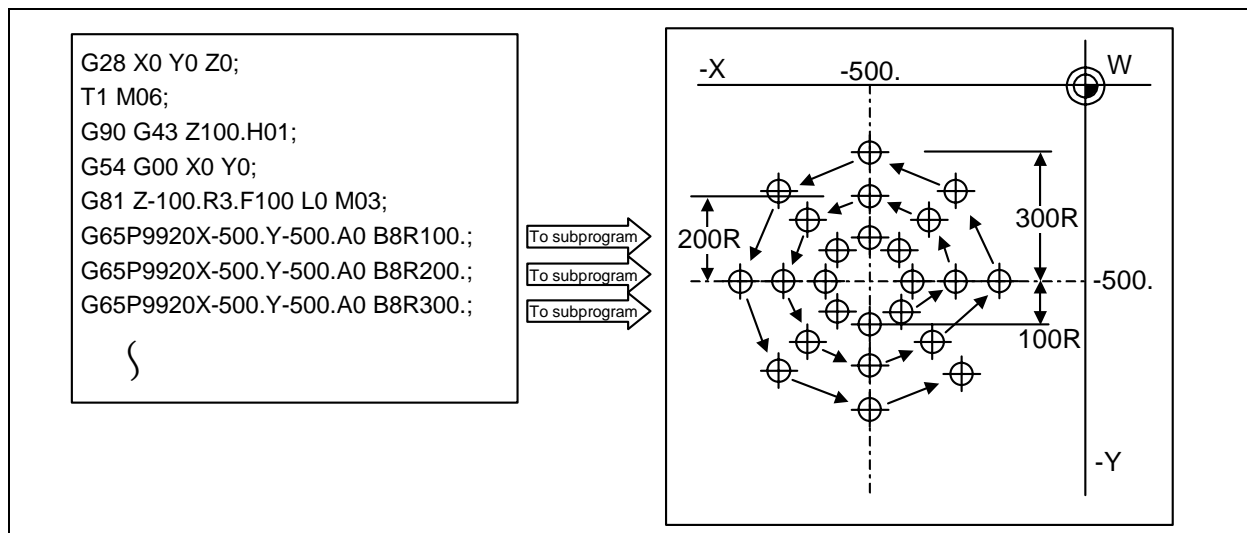
13.7 User Macro Specifications

(Example 2) Bolt hole circle

After defining the hole data with fixed cycle (G72 to G89), the macro command is issued as the hole position command.

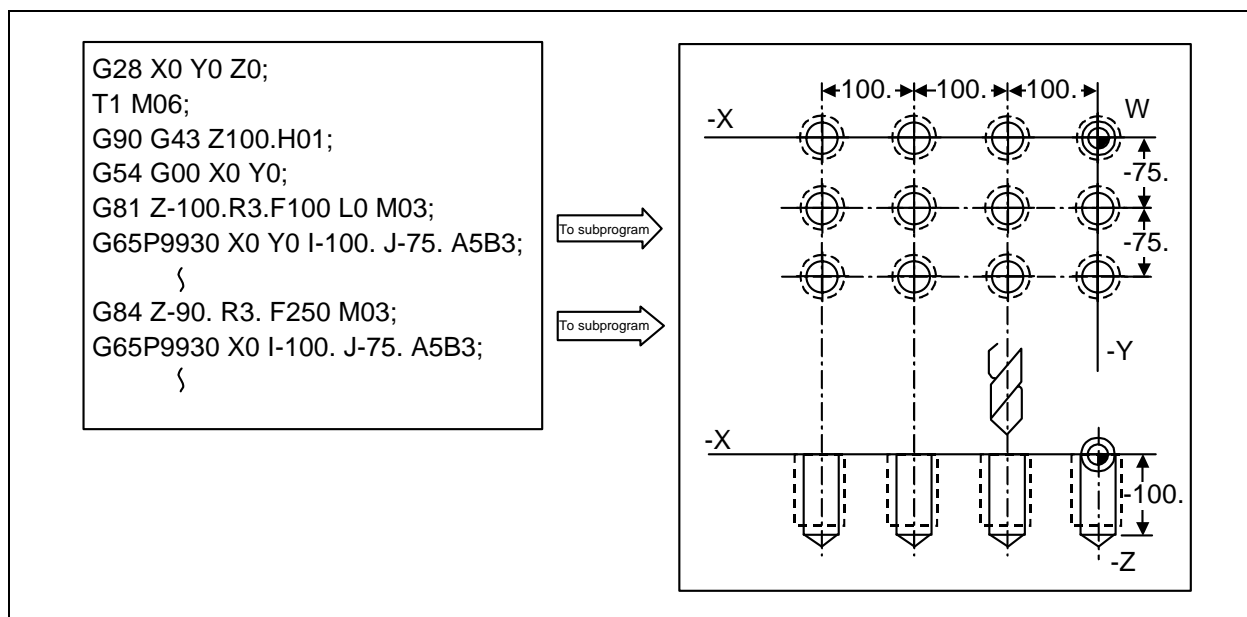
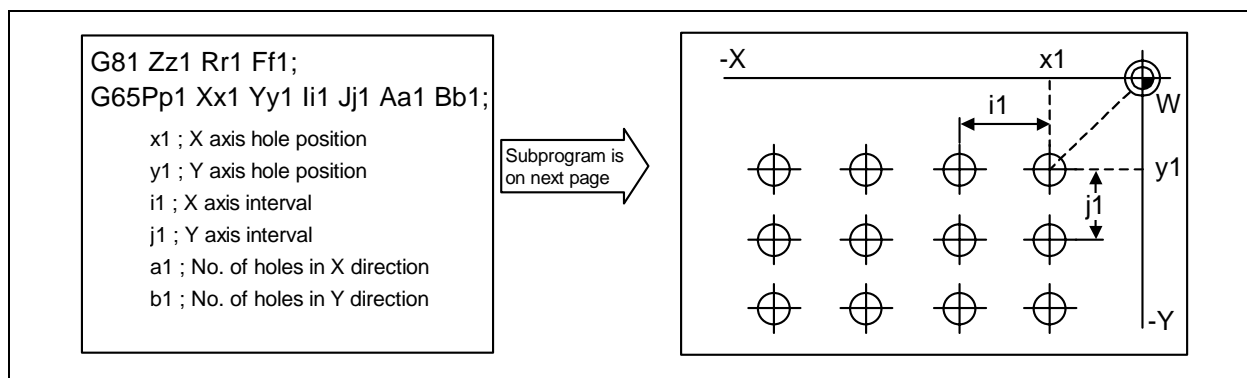


13. PROGRAMMING SUPPORT FUNCTIONS 13.7 User Macro Specifications



(Example 3) Grid

After defining the hole data with the fixed cycle (G72 to G89), macro call is commanded as a hole position command.



13. PROGRAMMING SUPPORT FUNCTIONS

13.7 User Macro Specifications

O9930 (Subprogram)

```

#101=#24 ;
#102=#25 ;

#103=#4 ;
#104=#5 ;

#106=#2 ;

WHILE [#106GT0] DO1 ;

#105=#1 ;

WHILE [#105GT0] DO2 ;

G90 X#101 Y#102 ;

#101=#101+#103 ;
#105=#105-1 ;

END2 ;

#101=#101-#103 ;
#102=#102+#104 ;

#103=-#103 ;
#106=#106-1 ;

END1 ;

M99 ;
    
```

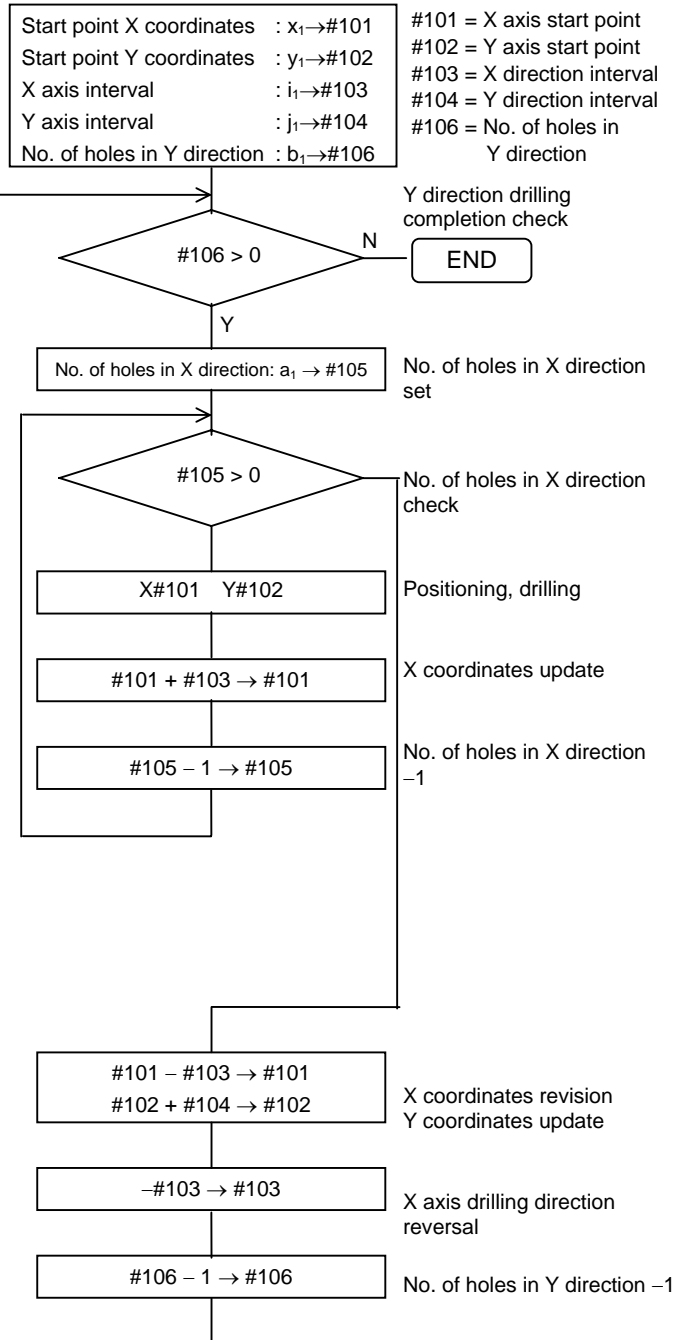
(Note 1)

(Note 1)

(Note 1)

(Note 1) The processing time can be shortened by programming in one block.

O9930



13.8 Linear Angle Command

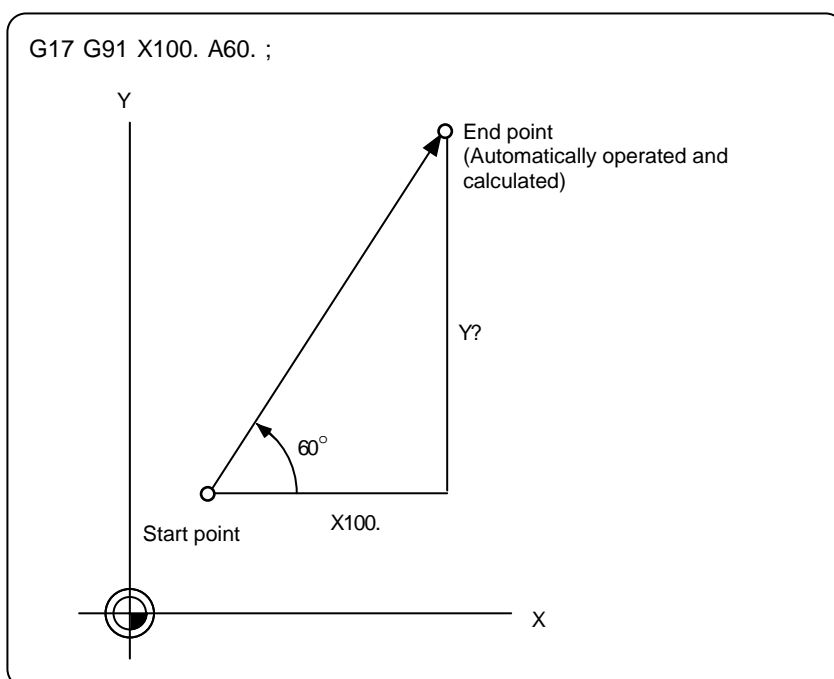
(1) Outline

The end point is automatically calculated by commanding the linear angle and one element of the end point coordinates (one element of the selected plane).

(2) Command format

<u>G17</u>	<u>Xx1</u>	<u>Aa1</u> ;
Selected plane	One element	Angle
Or G17	Yy1	Aa1 ;

(Example)



(3) Detailed description

- (a) The angle is from the + direction of the 1st axis horizontal axis on the selected plane. The counter-clockwise (CCW) direction is considered to be + and the clockwise direction (CW) –.
- (b) Either of the axes on the selected plane is commanded for the end point.
- (c) The angle is ignored when the angle and the coordinates of both axes are commanded.
- (d) When only the angle has been commanded, this is treated as a geometric command.
- (e) The angle of either the start point or end point may be used.
- (f) If address A is used for the axis name or as the 2nd miscellaneous function, the setting is interpreted as the A axis command or 2nd miscellaneous function command, and cannot be used for the linear angle.
- (g) This function is valid only for the G01 command; it is not valid for other interpolation or positioning commands.

(4) Notes

- (a) If the workpiece coordinate system selection and linear angle command are issued in the same block, the end point will be the coordinate value in the new coordinate system.

13.9 Geometric Command

(1) Outline

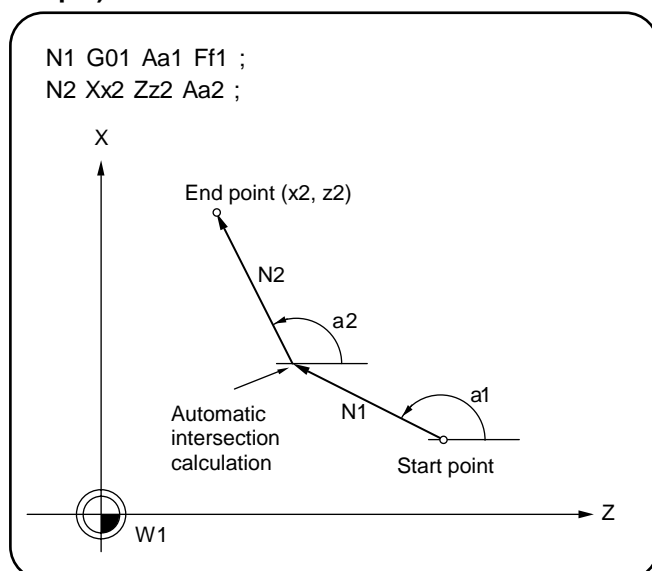
(a) Geometric command I

When it is difficult to determine the intersecting point of two straight lines during continuous linear interpolation command, the intersecting point of the two straight lines can be automatically calculated by creating a program that commands the angles of the straight lines.

(2) Command format

N1	<u>G17</u>	<u>Aa1</u> ;
	Selected plane	Angle
N2	<u>Xx2, Yy2,</u>	<u>Aa2</u> ;
	Next block's end point coordinates	Angle

(Example)



(3) Detailed description

- (a) The slope is expressed as the angle that is formed with the 1st axis horizontal axis + direction on the selected plane. The counterclockwise (CCW) direction is considered to be + and the clockwise direction (CW) -.
- (b) The range of the slopes in the 1st block(a1) and 2nd block(a2) is $-360.000 \leq a \leq 360.000$.
- (c) The slope of the 2nd block (a2) can be commanded on the start or end point side. The start or end point side of the commanded slope is identified automatically inside the NC.
- (d) The feedrate can be commanded for each block.
- (e) Single block stop is possible at the end point of the 1st block.
- (f) The end point coordinates of 2nd block are (Xx2, Yy2).

(4) Notes

- (a) Program error results when the geometric command is not on the selected plane.
- (b) The end point coordinates of the 2nd block should be commanded with absolute values. When incremental values are used, the program error will occur.
- (c) When the angle where the two straight lines intersect is less than 1° , program error will result.
- (d) The program error will occur when the plane is changed in the 1st block and 2nd block.
- (e) A program error will occur if a linear command is not issued in the 2nd block.
- (f) When A is selected for the axis name, or when A is selected for the 2nd miscellaneous function name, this function will be ignored.

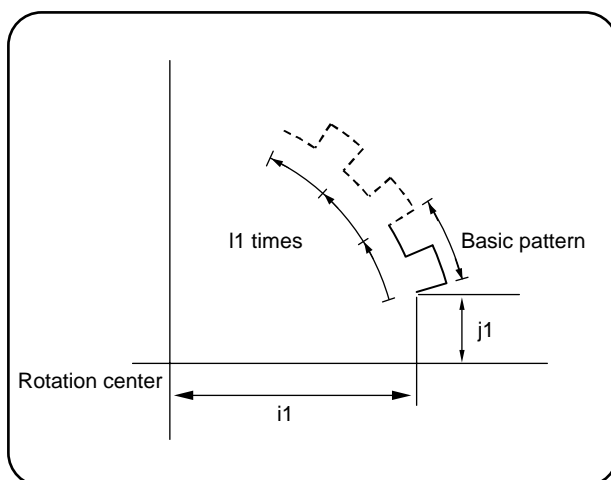
13.10 Pattern Rotation; G22 I_J_

(1) Outline

When the same pattern is repeatedly used on a concentric circle, one rotation machining pattern can be registered as a subprogram. By designating the rotation center when calling the subprogram from the main program, the analog rotation type path can be easily created on the concentric circle by this function. This allows programs to be created easily.

(2) Command format

<u>M98</u>	<u>i1</u> <u>J1</u>	<u>L1 (Aa1)</u> <u>Hh1</u>	<u>Pp1</u> ;
Call command	Rotation center coordinates	Subprogram number and sequence number	Number of repetitions



(3) Detailed description

- (a) The first subprogram called with subprogram call is executed at a rotation angle of 0°. The path follows the commands.
- (b) If the number of repetitions is set to two or more times, the rotation angle is obtained from the called subprogram's start point/end point and the rotation center coordinates. This is used as the rotation center coordinates reference. Using the path of the first subprogram as a basic pattern, rotation and arrangement corresponding to the designated number of calls can be carried out.
- (c) When rotating, all blocks in the subprogram rotate.
- (d) If the subprogram's start point and end point are not on the same circle using the commanded center of the pattern rotation center coordinates, the subprogram's end point will be handled as the start point. The movement will be interpolated using the end point of the first movement command block in the subprogram rotated next as the commanded end point.
- (e) Both absolute values and incremental values can be used in the figure rotation subprogram. Even if commanded with an absolute value command, the rotation will be the same as when commanded with an incremental value.
- (f) I and J are commanded with the incremental amount from the start point.
- (g) A subprogram cannot be branched to from a subprogram in pattern rotation.
- (h) The pattern rotation is carried out on the workpiece coordinates so the figure can be shifted with the G92, G52, G54 to G59 (workpiece coordinate system shift) commands.
- (i) During pattern rotation, functions on the rotation plane axis' machine coordinate system such as reference point return and uni-directional positioning cannot be carried out. However, functions on the machine coordinate system can be used for axes other than that on the rotation plane.

13. PROGRAMMING SUPPORT FUNCTIONS

13.10 Pattern Rotation; G22 I_J_

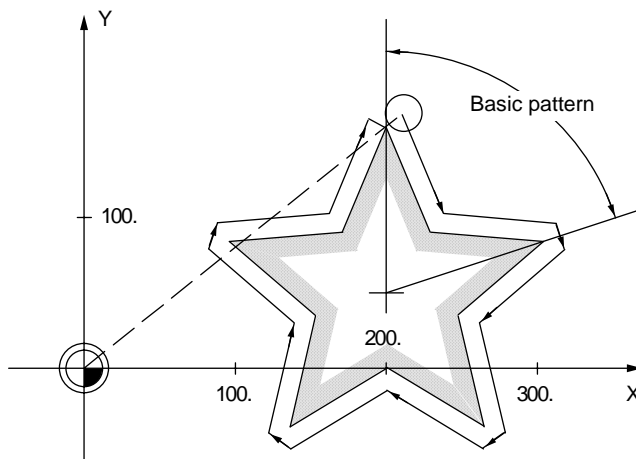
(4) Notes

- (a) A program error will occur if a pattern rotation command is used during pattern rotation.
- (b) Pattern rotation and program coordinate rotation cannot be commanded simultaneously.
A program error will occur.

(5) Program example

```
Main program (L1000)
(O)
N01 G90 G54 G00 X0 Y0 ;
N02 G01 G41 X200. Y150. D01 F500 ;
N03 G01 Z-50. F300 ;
N04 G22 L2200 L5 J-100. ;
(A)
N05 G90 G01 Z50. F500 ;
N06 G40 ;
N07 G00 X0 Y0 ;

Subprogram (L2200)
(O)
N01 G91 G01 X29.389 Y-59.549 ;
N02 X65.717 Y-9.549 ;
N03 G23 ;
```



13.11 Program Coordinate Rotation; G68.1/G69.1

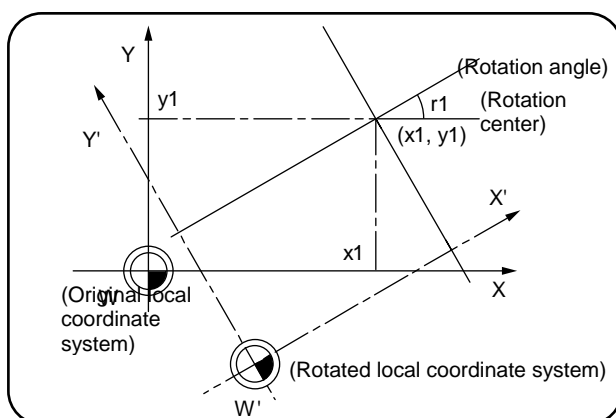
(1) Outline

When machining a complicated shape at a position rotated in respect to the coordinate system, the shape before rotation can be programmed on the local coordinate system, the parallel movement amount and rotation angle designated with the program coordinate rotation command, and the rotated shaped machined.

(2) Command format

Gn <u>G68.1</u>	<u>Xx1</u> <u>Yy1</u>	<u>Rr1</u> ;	Coordinate	rotation
Call command	Rotation center coordinates	Rotation angle	ON		
G69.1 ;				Coordinate rotation cancel	

- Gn : Plane selection code G17, G18, G19
Xx1, Yy1 : Two axes corresponding to the plane selected from the rotation center coordinates X, Y and Z. Designate with an absolute position.
Rr1 : Rotation angle. The counterclockwise direction is +.



(3) Detailed description

- (a) The plane selection (G17, G18, G19) does not need to be commanded in the G68.1 block. It will be valid even when designated earlier.
- (b) If the rotation center coordinates (x1, y1) are omitted, the position where the G68.1 command was executed will be the rotation center.
- (c) The rotation center coordinates (x1, y1) are commanded with an absolute value.
- (d) Rotation angle r1 is the minimum setting value between -360° and 360° .
- (e) The rotation takes place in the counterclockwise direction by the angle designated in rotation angle r1.
- (f) Rotation angle r1 is modal data, so it will not change until a new angle is commanded. Thus, the rotation angle r1 command can be omitted. If the first G68.1 is omitted, R will be set to 0.
- (g) The rotation angle r1 will enter the absolute mode or incremental mode depending on the G90 or G91 command.
- (h) The coordinate rotation command during coordinate rotation is processed as center coordinates and rotation angle changes.

(4) Notes

- (a) The program coordinate rotation cannot be executed simultaneously with figure rotation. The program error will occur if coordinate rotation is commanded during figure rotation, or figure rotation commanded during coordinate rotation.
- (b) The program error will occur if the plane selection is commanded during the coordinate rotation mode.
- (c) The program error will occur if the tool position offset is commanded during the coordinate rotation mode.

13. PROGRAMMING SUPPORT FUNCTIONS

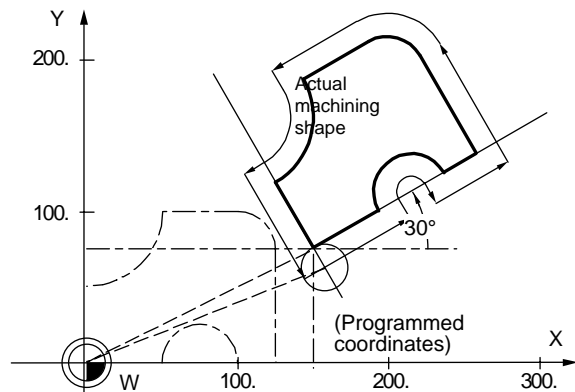
13.11 Program Coordinate Rotation; G68.1/G69.1

(5) Example

```
N01 G28 X Y Z ;  
N02 G54 G52 X150. Y75. ; Local coordinate designation  
N03 G90 G01 G42 X0 Y0 ; Radius compensation ON  
N04 G68.1 X0 Y0 R30. ; Coordinate rotation ON  
N05 G22 H101 ; Subprogram execution  
N06 G69.1 ; Coordinate rotation cancel  
N07 G54 G52 X0 Y0 ; Local coordinate system cancel  
N08 G00 G40 X0 Y0 ; Radius compensation cancel  
N09 M02 ; End
```

Subprogram
(Shape programmed with original coordinate system)

```
N101 G90 G01 X50. F200 ;  
N102 G02 X100. R25. ;  
N103 G01 X125. ;  
N104 Y75. ;  
N105 G03 X100. Y100. R25. ;  
N106 G01 X50. ;  
N107 G02 X0 Y50. R50. ;  
N108 G01 X0 Y0 ;  
N109 G23 ;
```



13. PROGRAMMING SUPPORT FUNCTIONS

13.12 Parameter Coordinate Rotation

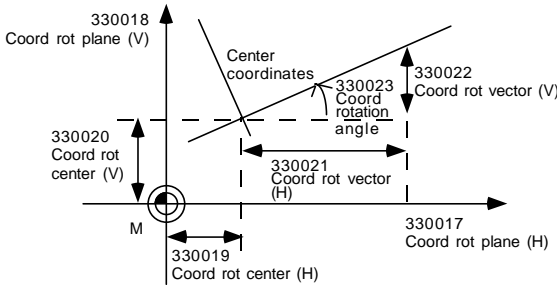
13.12 Parameter Coordinate Rotation

(1) Outline

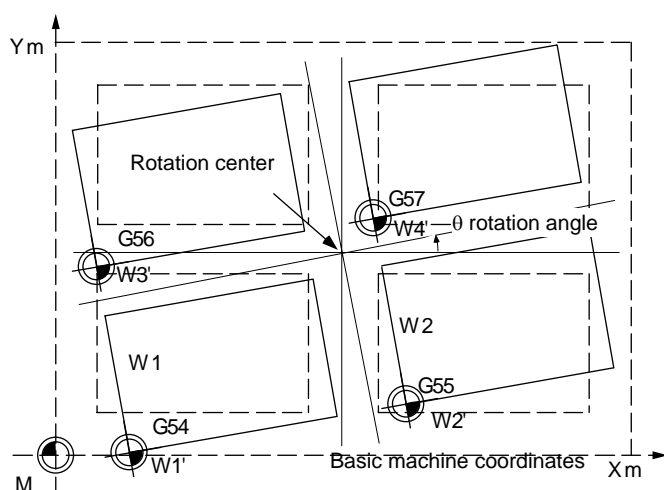
If a deviation occurs between the workpiece reference line and machine coordinate system coordinate axis while the workpiece is installed, the machine can be controlled to rotate the machining program coordinates according to the workpiece reference line deviation. The coordinate rotation amount is set with parameters. The parameters can be set from the setting display unit or with the G10 command. Note that when using the G10 command, the separate additional specifications "Program parameter input" are required.

(2) Setting parameters

Set the plane, center coordinates, vector component and angle for the coordinate rotation control.

No.	Item	Setting details	Setting range (unit)
33 0017	Coord rot plane (H)	Set the plane, center coordinates, vector component and angle for coordinate rotation control.	Axis control address names such as X, Y, Z, U, V, W, A, B and C.
33 0018	Coord rot plane (V)	Coordinate rotation plane (horizontal axis): Set the axis name for the 1st axis during rotation measurement.	
33 0019	Coord rot center (H)	If no axis name is set, the X axis will be automatically set.	-999999.999 to 999999.999mm
33 0020	Coord rot center (V)	Coordinate rotation plane (vertical axis): Set the axis name for the 2nd axis during rotation measurement.	-999999.999 to 999999.999mm
33 0021	Coord rot vector (H)	If no axis name is set, the Y axis will be automatically set.	-999999.999 to 999999.999mm
33 0022	Coord rot vector (V)		-999999.999 to 999999.999mm
33 0023	Coord rotation angle	 <p>(Note) The angle is automatically calculated when the vector component is set, but it is also possible to set the angle directly. In that case, the vector components for both the horizontal and vertical axes appear as "0".</p>	-180.000 to 180.000°

(3) Detailed description



- (a) The rotation center coordinate position designates the machine coordinate system position.
- (b) All workpiece coordinate systems from G54 to G59 rotate with the rotation command. Think of this as using the rotation center position designated with the machine coordinate system as the axis, and rotating the entire workpiece coordinates.
- (c) The coordinate value counter indicates the position in the local coordinate system before rotation.
- (d) When settings are made with the setting and display unit, the parameter settings are validated after cycle start. If set with the G10 command, the parameter settings are immediately validated.

13. PROGRAMMING SUPPORT FUNCTIONS

13.12 Parameter Coordinate Rotation

(4) Notes

- (a) If rotation angle zero is commanded while carrying out coordinate rotation, it will be canceled at the next movement command regardless of the G90 or G91 setting.
- (b) Command the first movement after this command with the G00 or G01 mode. If an arc command is issued, the arc start point will not be rotated. However, only the arc end point will rotate. This will cause the start point radius and end point radius to differ, and "PR0070: Arc end point deviation large" to occur.

(5) Example

To use for compensating positional deviation of pallet changer

```

N01 G28 X0 Y0 Z0 ;
N02 G22 L9000 ;      Pallet deviation measurement
                    (A) Parallel movement amount shift
N03 G90 G53 X0 Y0;  Parallel movement amount definition
N04 G92 X0 Y0 ;      Rotation amount definition
N05 G10 K15. ;       G54 workpiece
N06 G90 G54 G00 X0 Y0 ; machining
N07 G22 H101 ;       G55 workpiece
N08 G90 G55 G00 X0 Y0 ; machining
N09 G22 H101 ;       G56 workpiece
N10 G90 G56 G00 X0 Y0 ; machining
N11 G22 H101 ;       G57 workpiece
N12 G90 G57 G00 X0 Y0; machining
N13 G22 H101 ;
N14 G27 X Y Z ;
N15 M02 ;

Machining shape program
N101 G91 G01 G42 D01 F300 ;
N102 X100 ;
N103 G03 Y50. R25. ;
N104 G01 X-100. ;
N105 G03 Y-50. R25. ;
N106 G01 G40;
N107 G23 ;

```

13.13 Special Fixed Cycles; G34/G35/G36/G37

(1) Outline

Special fixed cycles are used for multiple drilling of pattern shapes. These cycles are used in combination with the hole drilling fixed cycles, and the hole drilling pattern shape is commanded with this special fixed cycle.

The special fixed cycles include the following:

- Bolt hole cycle
- Right angle
- Arc
- Grid

(2) Detailed description of common functions

- (a) Before commanding the special fixed cycle, always command the fixed cycle for drilling, and register the hole machining data. (If there is no positioning data, the hole drilling fixed cycle will not be executed, and only the data will be registered.)
- (b) When the special fixed cycle is executed, the axis will be positioned to the hole drilling position. The hole drilling operation is executed with the hole drilling fixed cycle.
- (c) Even after the special fixed cycle is executed, the registered hole drilling fixed cycle is held until canceled.
- (d) If the special fixed cycle is executed when not in the hole drilling fixed cycle mode, only positioning will be executed, and the hole drilling operation will not take place.
- (e) If a special fixed cycle is commanded without commanding the hold drilling fixed cycle, positioning will follow the modal G code of the 01 group.

13. PROGRAMMING SUPPORT FUNCTIONS
13.13 Special Fixed Cycles; G34/G35/G36/G37

(3) Detailed description of each function

(a) Bolt hole circle (G34)

1) Function

"n" number of holes are drilled on the circumference of radius R centered on the coordinates designated by X and Y. The circumference is divided into "n" divisions starting at the point formed by the X axis and angle θ . The drilling data of the G81 or other fixed cycle is retained for the drilling operation at each hole position.

All movements between hole positions are carried out in the G00 mode. G34 does not hold the data after the command is completed.

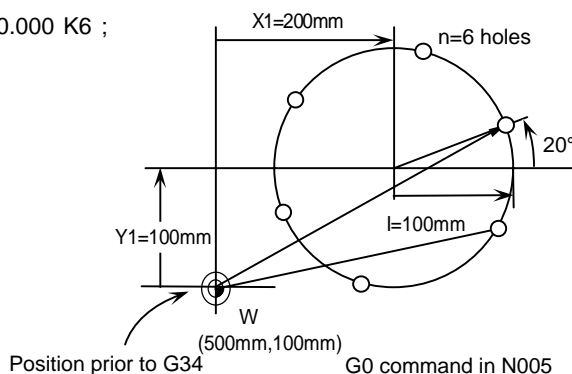
2) Command format

G34 X x1 Y y1 I r J θ K n ;
 X, Y Bolt hole circle center position; affected by G90/G91.
 I Circle radius "r"; unit is based on minimum setting unit and supplied as a positive number.
 J Angle θ at initial drilling point; counterclockwise direction is taken to be positive.
 K Number "n" of holes to be drilled. Any number from 1 to 9999 can be assigned; 0 cannot be assigned. An alarm will occur if 0 is set. A positive value will position in the counterclockwise direction, and a negative value will position in the clockwise direction.

3) Example

When input setting unit is 0.001mm

```
N001 G91 ;
N002 G81 Z-10.000 R5.000 L0 F200 ;
N003 G90 G34 X200.000 Y100.000 I100.000 J20.000 K6 ;
N004 G80 ; ..... (G81 cancel)
N005 G90 G0 X500.000 Y100.000 ;
```



As shown in the above example, the tool position applied upon completion of the G34 command is above the final hole. Thus, when moving it to the next position, it is convenient to command the movement amount in the absolute value mode.

(The coordinate value must be calculated to command the movement amount in the incremental mode.)

13. PROGRAMMING SUPPORT FUNCTIONS
13.13 Special Fixed Cycles; G34/G35/G36/G37

(b) Line at angle (G35)

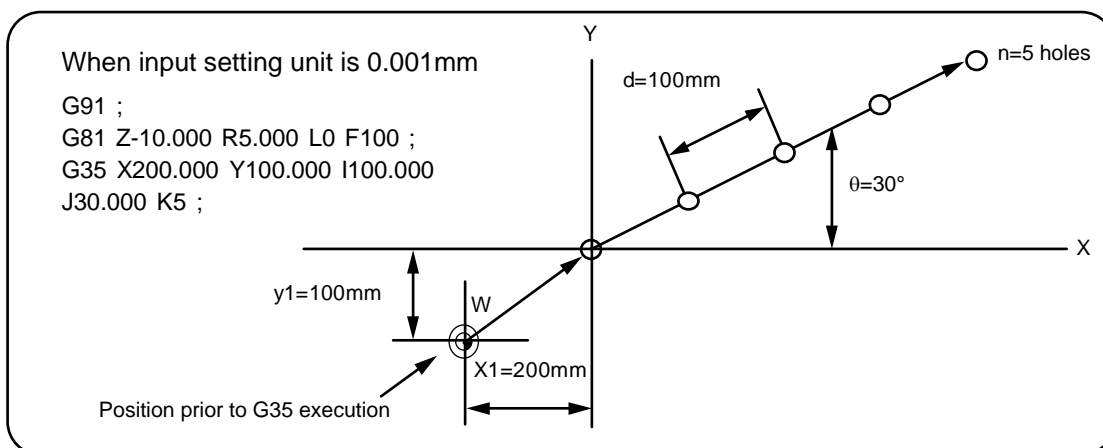
1) Function

"n" number of holes are drilled at distance "d" in the direction forming angle θ with the X axis, with the position assigned by X and Y serving as the start point. The drilling data (hole machining mode and hole machining data) must be retained first since the drilling operation at each hole position is dependent upon the hole drilling fixed cycle. All movements between hole positions are carried out in the G00 mode. The data is not retained upon completion of the G35 command.

2) Command format

G35 X x1 Y y1 I d J θ K n ;
 X, Y Start point coordinate assignment; affected by G90/G91.
 I Distance "d"; unit is based on minimum setting unit and when "d" is negative, drilling proceeds in the symmetrical direction of the point centered on the start point.
 J Angle θ ; counterclockwise direction is taken to be positive.
 K Number "n" of holes including the start point. Any number from 1 to 9999 can be assigned.

3) Example



4) Notes

- a. The program error will occur when the K command is K0 or when it is absent.
- b. The last 4 digits are valid when the value of K is more than 4 digits.
- c. Any address except G, L, N, X, Y, I, J, K, F, M, S, T or B programmed in the same block as the G35 command will be ignored.
- d. If there is a group 0 G command in the same block as the G35 command, the one commanded afterwards takes precedence.
- e. When a G22/G23 command is programmed in the same block as the G35 command, it will be ignored and the G35 command will be executed.
- f. If a G72 to G89 command is assigned in the same block as the G35 command, the hole drilling fixed cycle will be ignored and the G35 command will be executed.

13. PROGRAMMING SUPPORT FUNCTIONS
13.13 Special Fixed Cycles; G34/G35/G36/G37

(c) Arc (G36)

1) Function

"n" number of holes are drilled at angle distance $\Delta\theta$ starting at the point where angle θ is formed with the X axis along a circumference with radius "r" and centered on the coordinates assigned by X and Y. As with the bolt hole circle fixed cycle, the drilling data must be retained beforehand since the drilling operation at each hole position depends on the hole drilling fixed cycle.

All movements between hole positions are carried out in the G00 mode. The data is not retained upon completion of the G36 command.

2) Command format

G36 X x1 Y y1 I r J θ P $\Delta\theta$ K n ;
 X, Y Circle center coordinates; affected by G90/G91.
 I Circle radius "r", unit is based on minimum setting unit and supplied as a positive number.
 J Angle θ at initial drilling point; counterclockwise direction is taken to be positive.
 P Angle distance $\Delta\theta$; counterclockwise when positive and clockwise when negative.
 K Number "n" of holes to be drilled. Any number from 1 to 9999 can be assigned.

3) Example

When input setting unit is 0.001mm

```
N001 G91 ;
N002 G81 Z-10.000 R5.000 F100 ;
N003 G36 X300.000 Y100.000 I300.000 J10.000
P15.000 K6 ;
```

Position prior to G36 execution

Y1=100mm

X1=300mm

$\Delta\theta=15^\circ$

$\theta=10^\circ$

n=6 holes

13. PROGRAMMING SUPPORT FUNCTIONS
13.13 Special Fixed Cycles; G34/G35/G36/G37

(d) Grid (G37)

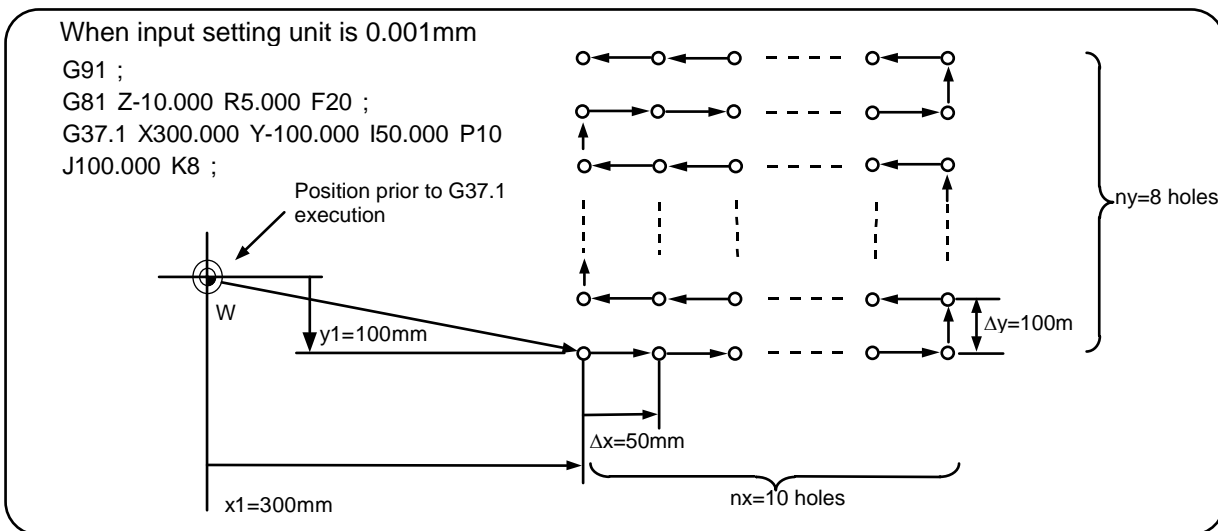
1) Function

The holes are drilled at points on the "nx" grid at intervals Δx parallel to the X axis, with the position assigned by X and Y as the start point. The hole drilling data (hole machining mode and hole machining data) must be retained beforehand since the hole drilling operation at each hole position depends on the hole drilling cycle. All movements between hole positions are carried out in the G00 mode. The data is not retained upon completion of the G37.1 command.

2) Command format

```
G37 X x1 Y y1 I  $\Delta x$  P nx J  $\Delta y$  K ny ;
X, Y ..... Start point coordinate assignment; affected by G90/G91.
I ..... X-axis interval  $\Delta x$ ; unit is based on minimum setting unit; positive direction
intervals as seen from start point when  $\Delta x$  is positive and negative
direction when it is negative.
P ..... Number "nx" of holes to be drilled in X-axis direction. Any number from 1
to 9999 can be assigned.
J ..... Y-axis interval  $\Delta y$ ; unit is based on minimum setting unit; positive direction
intervals as seen from start point when  $\Delta y$  is positive and negative
direction when it is negative.
K ..... Number "ny" of holes to be drilled in Y-axis direction. Any number from 1
to 9999 can be assigned.
```

3) Example



4) Notes

- The program error will occur when the P or K command is P0 or K0, or when they are absent.
- The last 4 digits are valid when the value of P or K is more than 4 digits.
- Any address except G, L, N, X, Y, I, P, J, K, F, M, S, T, or B programmed in the same block as the G37 command will be ignored.
- If there is a group 0 G command in the same block as the G37 command, the one commanded afterwards takes precedence.
- When a G22/G23 command is programmed in the same block as the G37 command, it will be ignored and the G37 command will be executed.
- If a G72 to G89 command is assigned in the same block as the G37 command, the hole drilling fixed cycle will be ignored and the G37 command will be executed.

13.14 Circular Cutting; G12/G13

(1) Outline

Circular cutting is a function which performs a series of operations: the tool is made to depart from the center of the circle and while cutting the inner circumference of the circle, it describes a perfect circle, and then it is returned to the center of the circle. The position where G12/13 is commanded, is the circle center.

(2) Command format

The program format is as follows.

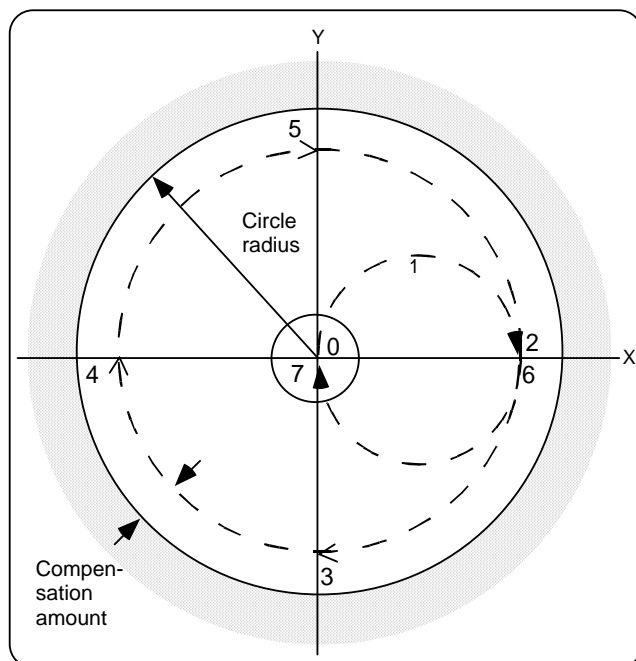
<u>G12/G13</u>	<u>I</u>	<u>D</u>	<u>F</u> ;
Circular cutting command	Perfect circle radius	Compensation number	Speed

G code	Function
G12	CW (Clockwise)
G13	CCW (Counterclockwise)

I : Circle radius (incremental value), sign is ignored
D: Compensation number

(3) Detailed description

(a) The operation is as shown below.



With G12 command (tool center path)
0→1→2→3→4→5→6→7→0
With G13 command (tool center path)
0→7→6→5→4→3→2→1→0

(b) The "+" compensation amount sign denotes reduction and the "-" sign expansion.
(c) Circular cutting is performed on the plane currently selected by G17, G18 and G19.

(4) Notes

- A program error will occur when compensation number "D" is not assigned or when an illegal number is assigned.
- A program error occurs when [radius (I) – compensation amount] is zero or a negative value.
- If G12 or G13 is specified while the radius compensation (G41, G42) is specified, the radius compensation becomes valid for the path after compensation is made according to the D commanded by G12 or G13.
- If the address H is commanded in the same block as G12 or G13, program error occurs.

13.15 Scaling; G50/G51

(1) Outline

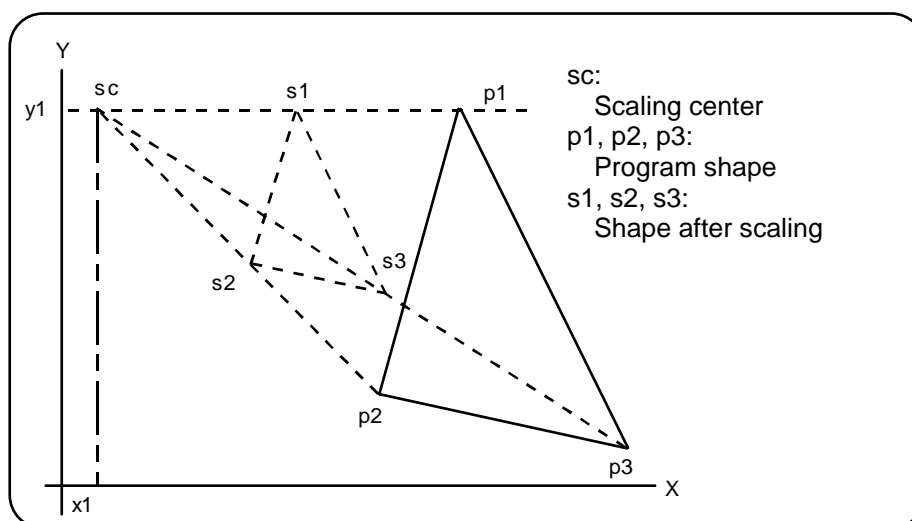
With this command, the shape commanded in the program can be enlarged or reduced by applying a magnification to the movement axis command value in the commanded range.

G code	Function
G50	Scaling cancel
G51	Scaling ON

(2) Command format

The program format is as follows.

<u>G51</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Zz1</u>	<u>Pp1</u>	;
Call command	Scaling center coordinate values			Magnification	



(3) Detailed description

(a) Designating the scaling axis, scaling center and magnification

When G51 is commanded, the scaling mode will be entered. The G51 command only designates the scaling axis, center and magnification, and does not move.

The scaling mode is entered with the G51 command, but the axis for which scaling is actually effective is the axis for which the scaling center is commanded.

1) Scaling center

- a. The scaling center follows the absolute/incremental mode (G90/G91) set at that point.
- b. The scaling center must be commanded even when the current position is to be the center.
- c. As explained above, the axis for which scaling is actually effective is the axis for which the scaling center is commanded.

2) Scaling magnification

- a. The scaling magnification is commanded with address P.
- b. Minimum command unit: 0.000001
- c. Command range: 1 to 99999999 (0.000001 to 99.999999-fold) or 0.000001 to 99.99999 are both valid.
- d. If the magnification is not commanded in the same block as G51, the magnification set in the parameters will be used.

13. PROGRAMMING SUPPORT FUNCTIONS

13.15 Scaling; G50/G51

- e. Even if this parameter is changed during the scaling mode, the changes will not be valid. The scaling will be applied with the value set when G51 is commanded.
- f. If the magnification is not set in the program or parameters, it will be calculated as 1-fold.

3) A program error will occur in the following cases.

- a. If scaling is commanded when scaling specifications are not provided.
- b. If the upper limit of the magnification command range is exceeded in the same block as G51.

(b) Scaling cancel

The scaling mode is canceled when G50 is commanded.

(4) Notes

- (a) Scaling is not applied to the offset amounts such as tool radius compensation, tool position offset or tool length offset. (The compensation and offset is calculated in respect to the scaled shape.)
- (b) Scaling is valid only for movement commands in automatic operation. It is invalid for manual movement.
- (c) For X, Y and Z, the scaling is valid only for the commanded axes. Scaling is not applied on the axis not having a command.
- (d) For the arc command, if scaling is validated for one of the two axes that configure the arc plane, scaling will be applied only on that axis.
- (e) If M02, M30 or M00 (only for M0 reset) is commanded during the scaling mode, the cancel mode will be entered.
- (f) If the coordinate system is shifted (G92, G52 command) during scaling, the scaling center will also shift by the difference amount.
- (g) If manual interrupt is applied during scaling, the manual ABS selection will be ignored, and the same operation as manual ABS OFF will take place.

(5) Combination with other functions

- (a) G27 reference point compare command
If G27 is commanded during scaling, the scaling will be canceled when the command is completed.
- (b) Reference point return command (G28, G29, G30)
If G28, G30 or reference point return is commanded during scaling, the scaling will be canceled at the middle point, and the axis will return to the reference point. If the middle point is ignored, the axis will return directly to the reference point.
If G29 is commanded during scaling, the scaling will be applied to the movement after the middle point.
- (c) G60 (uni-directional positioning) command
If G60 (uni-directional positioning) is commanded during scaling, the scaling will be applied at the final positioning amount, and will not be applied on the creep amount. In other words, the creep amount is constant regardless of the scaling application.
- (d) Workpiece coordinate system changeover
If the workpiece coordinate system is changed during scaling, the scaling center will be shifted by the difference of the new workpiece coordinate system and old workpiece coordinate offset amounts.
- (e) Figure rotation command
If figure rotation is commanded during scaling, the scaling will be applied on the figure rotation center and rotation radius.

13. PROGRAMMING SUPPORT FUNCTIONS

13.15 Scaling; G50/G51

- (f) Scaling command in figure rotation subprogram
If scaling is commanded in the figure rotation subprogram, the scaling will not be applied on the figure rotation's radius. Instead scaling will be applied only on the shape created by the subprogram.
- (g) During coordinate rotation
If scaling is commanded during coordinate rotation, the scaling center will rotate. Scaling can be executed on the rotated scaling center.
- (h) G51 command
If G51 is commanded during the scaling mode, the axis for which the center is newly commanded will also become a scaling valid axis. The magnification commanded by the latest G51 command will be valid.

13.16 Normal Line Control; G40.1/G41.1/G42.1

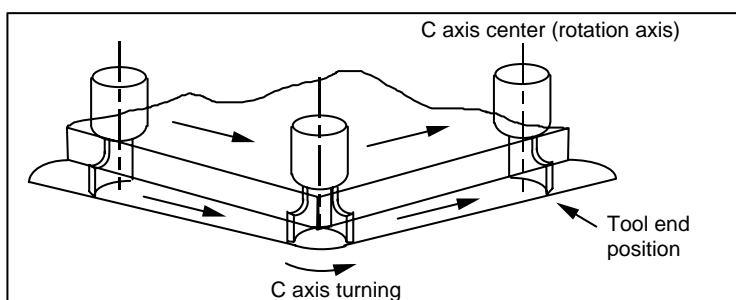
(1) Outline

If the C axis is set as the normal line control axis, the C axis (rotation axis) turning will be controlled so that the tool constantly faces the normal line direction control in respect to the XY axis movement command during program operation.

At the block seams, the C axis turning is controlled so that the tool faces the normal line direction control at the next block's start point.

During arc interpolation, the C axis turning is controlled to be synchronized with the arc interpolation speed operation.

This can be used in spring machining, or stitching such as with a sewing machine or for carpets. Only one axis is valid for the C axis (rotation axis).



The normal line control I and II can be used according to the C axis turn direction during normal line control. This is determined by the machine manufacturer specifications.

Normal line control type	Turning direction	Turning speed	Application
Type I	Direction that is 180° or less (shortcut direction)	Parameter speed	Cutting machining
Type II	As a principle, the commanded direction	Feedrate	Spring machining

(2) Command format

```
GΔΔ Xx1 Yy1 Ff1 ;
```

GΔΔ	Function
G40.1	Normal line control cancel
G41.1	Normal line control left ON
G42.1	Normal line control right ON

x1: X axis end point coordinates

y1: Y axis end point coordinates

f1 : Feedrate

The axis that carries out normal line control differs according to the machine.

The normal line control plane is fixed to the 1-axis–2-axis (I axis – J axis of basic 3 axes) plane.

The plane selection has no effect.

(3) Detailed description

(a) Definition of C axis angle

1) C axis turning operation in respect to movement command

a. Start up

After the C axis turns to a right angle of the movement direction by the XY axis command, the XY axes are moved.

b. During normal control mode

1. Block seam

No tool radius compensation	After the C axis is turned to be at a right angle of the XY axis movement in the next block, that block is moved.
With tool radius compensation	If radius compensation is applied on a linear-linear, linear-arc or arc-arc block that crosses a sharp angle, a linear block can be inserted. At this time, the C axis normal line control inserts a C axis turn movement in respect to the shape after radius compensation.

2. During block movement

- The C axis angle is kept constant during the linear command, and the C axis does not turn.
- During the arc command, the C axis turns in synchronization after the XY arc interpolation axis.

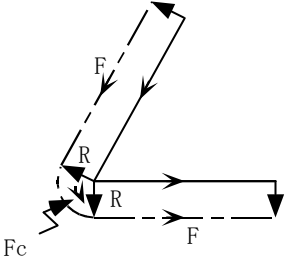
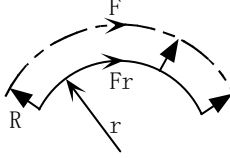
c. Cancel

The C axis does not turn, and the axis movement by the XY axis command is carried out.

13. PROGRAMMING SUPPORT FUNCTIONS
13.16 Normal Line Control; G40.1/G41.1/G42.1

2) C axis turning speed

Turning speed at block seam (Select from type 1 or type 2)

Item	Normal line control type I	Normal line control type II
C axis turning speed at block seam	Set the C axis turning speed F_c with the parameter (basic system parameter "C_Feed"). Note that if the C axis turning speed exceeds the C axis cutting clamp speed, the speed will be the C axis clamp speed.	<p>The tool end position feedrate is automatically controlled by CNC internal calculations so that it matches the F command value.</p> <p>The C axis turning speed F_c is obtained with the following expression.</p> $F_c = F/R * 180/\pi \text{ (}^\circ/\text{min)}$ <p>F : Feedrate (mm/min) R : Parameter value (mm) Length from C axis center to tool tip (Value set in machining parameter "Normal C axis turn R")</p> <p>When R = 0 $F_c = F * 180/\pi \text{ (}^\circ/\text{min)}$</p> <p>The rapid traverse rate is also obtained with the above expression.</p> <p>Note that if F_c exceeds the C axis cutting feed clamp speed, F_c will be the C axis cutting feed clamp speed.</p> 
C axis turning speed during circular interpolation	The speed becomes the circular interpolation speed (angle speed) corresponding to the X and Y axes interpolation speed. However, if the C axis turning speed exceeds the C axis cutting clamp speed, the circular interpolation speed (F command) will be clamped so that the C axis turning speed becomes the C axis cutting feed clamp speed.	<p>The tool end position feedrate is automatically controlled by CNC internal calculations so that it matches the F command value.</p> <p>The circular interpolation speed F_r is obtained with the following expression.</p> $F_r = F*r/(R+r) \text{ (mm/min)}$ <p>F : Command speed (mm/min) r : Arc radius (mm) R : Parameter value (mm) Length from C axis center to tool end</p> <p>Note that if F_r exceeds the C axis cutting feed clamp speed, the circular interpolation speed (F command) will be clamped so that the C axis turning speed becomes the C axis cutting feed clamp speed.</p> 

3) Definition of C axis turning direction

The C axis angle is 0° (degree) when the tool is facing the +X direction.

The counterclockwise direction turning is + (plus), and the clockwise direction turning is - (minus).

13. PROGRAMMING SUPPORT FUNCTIONS

13.16 Normal Line Control; G40.1/G41.1/G42.1

(4) Notes

- (a) If the spindle is to be the axis targeted for normal line control, the spindle must be changed to a servo axis. The spindle → C axis (servo ON), and C axis → spindle (servo OFF) changeover is carried out with M commands.
- (b) The normal line control axis will stop at the turning start position at the single block, cutting block start interlock and block start interlock.
- (c) The C axis movement command is ignored during normal line control.
- (d) During C axis normal line control (during the G41.1 and G42.1 modal), the C axis workpiece offset rewrite command (G92C_) cannot be issued. The program error (PR0901) will occur if commanded.
- (e) If mirror image is applied to either the 1-axis or 2-axis, the normal line control will be reversed.

13. PROGRAMMING SUPPORT FUNCTIONS
13.16 Normal Line Control; G40.1/G41.1/G42.1

(5) Relation with other functions

Function name	Notes
Uni-directional positioning	Normal line control is not applied.
Helical cutting	Normal line control is applied normally.
Spiral interpolation	The start point and end point are not on the same arc, so normal line control is not applied correctly.
Synchronous feed	When using the spindle for C axis control, the correct feedrate will not be reached.
Exact stop check	The operation will not decelerate and stop for the turning movement of the normal line control axis.
Error detect	The operation will not decelerate and stop for the turning movement of the normal line control axis.
Override	Override is applied on the turning movement by normal line control.
Figure rotation	Normal line control is applied to the shape after figure rotation.
Coordinate rotation	Normal line control is applied to the shape after coordinate rotation.
Scaling	Normal line control is applied to the shape after scaling.
Mirror image	Normal line control is applied to the shape after mirror image.
Linear angle designation	Normal line control is applied to the shape after the linear angle command.
Geometric designation	Normal line control is applied to the shape after geometric command.
Automatic reference point return	Normal line control is not applied.
Start position return	Normal line control is not applied on the movement to the middle point position. If the control parameter "G00 interpolation OFF" is OFF in respect to the movement from the middle point to a position designated in the program, normal line control is applied.
Workpiece coordinate system offset	The normal line control axis operates in the program coordinate system set during normal line control.
Local coordinate system offset	The normal line control axis operates in the program coordinate system set during normal line control.
Program restart	The block in the normal line control can be restarted.
Dry run	The feedrate is changed by the dry run signal even in respect to the turning movement of the normal line control axis.
Machining time calculation	The time of the section turned by the normal line control is also calculated.
Graphic check	The section turned by the normal line control is also drawn.

13.17 Macro Interrupt; ION / IOF

(1) Outline

By inputting a user macro interrupt from the PLC, another designated program (interrupt program) can be called as a priority to the program currently running.

Examples of this interrupt program include programs for evading abnormalities such as escaping at a tool breakage, or returning.

The following two types of interrupt can be selected with parameters.

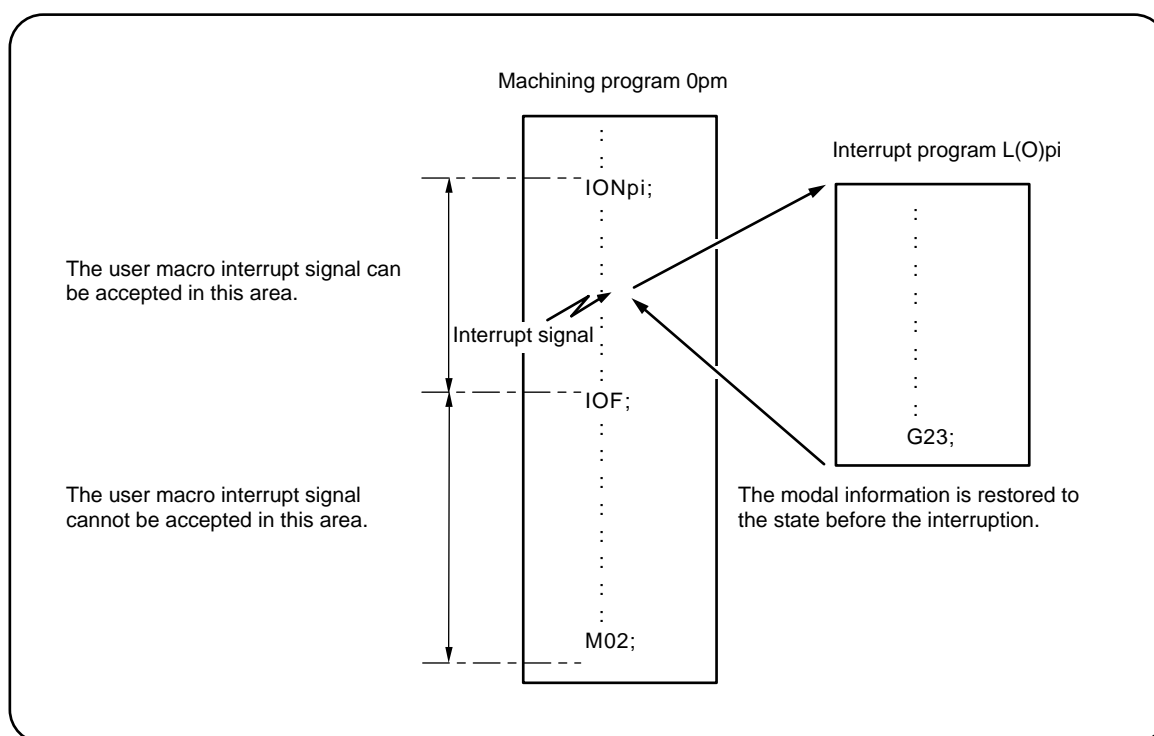
[Interrupt type 1] The running block is immediately stopped, and the interrupt program is executed.

[Interrupt type 2] The interrupt program is executed after completing the running block.

(2) Command format

The program format is as follows.

IONp Hh ; p: Interrupt program number h: Interrupt sequence number	User macro interrupt valid
IOF ;	User macro interrupt invalid



- If an interrupt signal is input from the machine side during the user macro interrupt valid period (from when ION is commanded to when IOF is commanded or the NC is reset), the user macro interrupt will start, the program commanded with ION_ will interrupt the machining program currently running, and will be executed instead.
- During the user macro interrupt, or in the user macro interrupt invalid state (after IOF command or NC reset), the input interrupt signal will be ignored.

13. PROGRAMMING SUPPORT FUNCTIONS

13.17 Macro Interrupt; ION/IOF

(3) Detailed description

(a) Valid conditions

The user macro interrupt can be executed only when a program is running.

- 1) The automatic operation mode or MDI is selected.
- 2) Automatic startup has been executed.
- 3) A user macro interrupt is not applied.
- 4) The parameter "Macro interrupt valid" is selected.

(Note) Macro interrupt is invalid during manual operation.

(b) Operation outline

- 1) After IONp1; is commanded in the currently running program, if the user macro interrupt signal is input, the interrupt program L(O)p1 will be executed. The original program will be returned to by the G23; command in the interrupt program.
- 2) When G23Hh2; is commanded, the area from the block following the interrupted block to the last block of the program will be searched for. If not found, the area from the block at the head of the program to the block before the interrupted block will be searched for, and the system will return to the block with the sequence number Nh2 that appears first.

(c) Interruption method

The type 1 or type 2 interruption method can be selected with parameters.

[Type 1]

- When the interrupt signal is input, the movement or dwell being executed at that point will be stopped, and the interrupt program will be executed.
- If there is a movement command or miscellaneous function command (MSTB) in the interrupt program, the command in the stopped block will be lost, and the interrupt program will be executed. When the interrupt program is completed, the program will restart from the block following the stopped block.
- If there is no movement command or miscellaneous function command (MSTB) in the interrupt program, after returning from the interrupt program, the program will restart from where the block was interrupted. However, if the interrupt signal is input during execution of a miscellaneous function command (MSTB), the NC will enter the completion signal standby state. Thus, after the completion signal is input, the movement command or miscellaneous function command (MSTB) in the interrupt program will be executed.

[Type 2]

- When the interrupt signal is input, the interrupt program will be executed without stopping the command being executed at that point.
- If there is a movement command or miscellaneous function command (MSTB) in the interrupt program, the movement command or miscellaneous function command (MSTB) in the interrupt program will be executed after the block command being executed is completed. However, if the interrupt program is not completed even when the original block is completed, the machining may temporarily stop.

(d) Calling method

The user macro interrupt includes the following types. The method to be used is selected with the parameters.

- Subprogram type interrupt
- Macro type interrupt

The calling method differs for these interrupt programs.

Both methods are added to the call nesting level. Furthermore, the subprogram and user macro call carried out in the interrupt program are also added to the nesting level.

13. PROGRAMMING SUPPORT FUNCTIONS

13.17 Macro Interrupt; ION/IOF

Subprogram type interrupt	The user macro interrupt program is called as a subprogram. (Same as G22 call) In other words, the local variable level does not change before and after the interruption.
Macro type interrupt	The user macro interrupt program is called as a user macro. (Same as G65 call) In other words, the local variable level changes before and after the interruption. Arguments cannot be transferred from the execution program to the interrupt program.

(e) User macro interrupt signal acceptance method

The acceptance methods for the user macro interrupt signal include the following. The method is selected in the parameters.

- Status trigger method
- Edge trigger method

Status trigger method	The signal is accepted as a valid signal while the user macro interrupt signal is ON. If the interrupt signal is ON when the user macro interrupt is validated with ION, the interrupt program will be executed. If the interrupt signal is continuously turned ON, the interrupt program can be repeated.
Edge trigger method	The signal is accepted as a valid signal at the rising edge where the user macro interrupt signal changes from OFF to ON. This is used when the interrupt program is to be executed only once, etc.

(f) Returning from user macro interrupt

G23 (H_);

To return to the original program from the user macro interrupt program, command G23 in the interrupt program. The sequence number in the program to be returned to can be designated with address H. In this case, the area from the block following the interrupted block to the last block of the program will be searched for. If not found, the area from the block at the head of the program to the block before the interrupted block will be searched for, and the system will return to the block with the sequence number that appears first.
(Same as G23H_ in G22 call)

(g) Modal information in user macro interrupt

If the modal information is changed in the interrupt program, the modal information after returning from the interrupt program will change as follows.

[When returned with G23 ;]

The modal information changed in the interrupt program is invalidated, and returns to the modal information before interruption.

When using interrupt method type 1 and there is a movement command or miscellaneous function command (MSTB) in the interrupt program, the modal information will not return to that before the interruption.

[When returned with G23H_ ;]

If the modal information is changed in the interrupt program, the modal information changed in the interrupt program will be retained even after returning from the interrupt program. This is the same as returning with G23H_ ; from a program called with G22, etc.

13. PROGRAMMING SUPPORT FUNCTIONS

13.17 Macro Interrupt; ION/IOF

(h) Modal information variables (#4401 to #4520)

The modal information for when the control changes to the user macro interrupt program can be recognized by reading the values of #4401 to #4520.

The unit is the commanded unit.

System variable	Modal information
#4401 to #4421	G code (Group 1) to G code (Group 21)
#4507	D code
#4509	F code
#4511	H code
#4513	M code
#4514	Sequence number
#4515	Program number
#4519	S code
#4520	T code

These variables can be used only in the user macro interrupt program.

An error will occur if used in programs other than the user macro interrupt program.

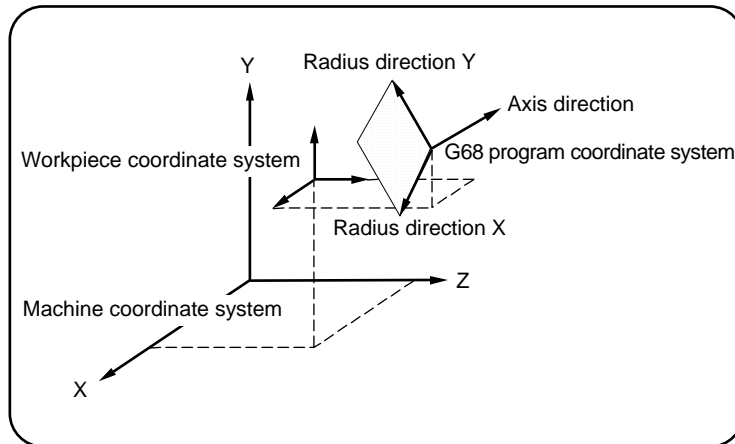
(4) Restrictions

- (a) The user macro interrupt function is part of the user macros, and cannot be used if the user macro specifications are not provided.
- (b) If the system variables #5001 (position information) and following for reading the coordinate values are used in the user macro interrupt program, the read coordinate value will be that read with the pre-read buffer.
- (c) When interrupting during execution of the following function, always designate a sequence number (G23H_ ;) in the return command from the user macro interrupt program. The original program will not be returned to correctly if the sequence number is not designated.
 - Tool radius compensation

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(1) Outline

In respect to the X, Y and Z axes of the currently set workpiece coordinate system, the new coordinate system in which rotation and zero point parallel movement have been carried out can be defined. By using this function, a random plane in the space can be defined, and machining can be carried out by commanding normal programs in respect to that plane.



(2) Command format

The program format is as shown below.

<u>G68.1</u>	<u>Xx</u> <u>Yy</u> <u>Zz</u>	<u>li</u> <u>Jj</u> <u>Kk</u>	<u>Rr</u> ;
Three-dimensional coordinate conversion mode command	Rotation center coordinates	Rotation center axis direction *	Rotation angle

*: If the rotation center axis is the X axis, set "1" in i. If not the rotation center axis, set "0".

x, y, z	Rotation center coordinate	Designated with the absolute position in the local coordinate system.
i, j, k	Rotation center axis direction	1: Designated, 0: Not designated. Note that only one of the three axes is designated.
r	Rotation angle	The counterclockwise direction looking at the rotation center from the rotation center axis direction is the plus (+) direction. Command as a 0.001° unit within the range of -360.000 to 360.000°.

<u>G69.1</u> ;
Three-dimensional coordinate conversion mode cancel command

There are limits to the G codes that can be commanded during the three-dimensional coordinate conversion mode.

(3) Detailed description

(a) Commands

- 1) Command the rotation center coordinates with absolute values.
- 2) If the rotation center coordinates are omitted, the zero point of the currently set coordinate system will be the rotation center coordinates.
- 3) Designate values for I, J and K. If even one is not designated, the program alarm "PR0033 Format error" will occur. If all settings are omitted, the normal program coordinate rotation will be applied. Set 1 in only one of I, J and K, and set 0 for the other two. "PR0033 Format error" will occur if 1 is set in two or more. "PR0033 Format error" will also occur if 0 is set for all.
An error will occur if a G code that cannot be commanded, is commanded in the three-dimensional coordinate conversion mode.
- 4) When addresses I, J and K are not designated, this will be handled as the program coordinate rotation.
- 5) If a number other than 0 (including two-digit numbers) is designated for addresses I, J and K, this will be handled as 1.
- 6) Some of the G codes that can be commanded in the three-dimensional coordinate conversion modal are limited. If a G code that cannot be commanded in the three-dimensional coordinate conversion modal is issued, the program error "PR0921 3DIM coord convert error" will occur.
- 7) If the three-dimensional coordinate conversion specifications are not provided, the program error "PR0920 No 3DIM coord conversion spec." will occur.

13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

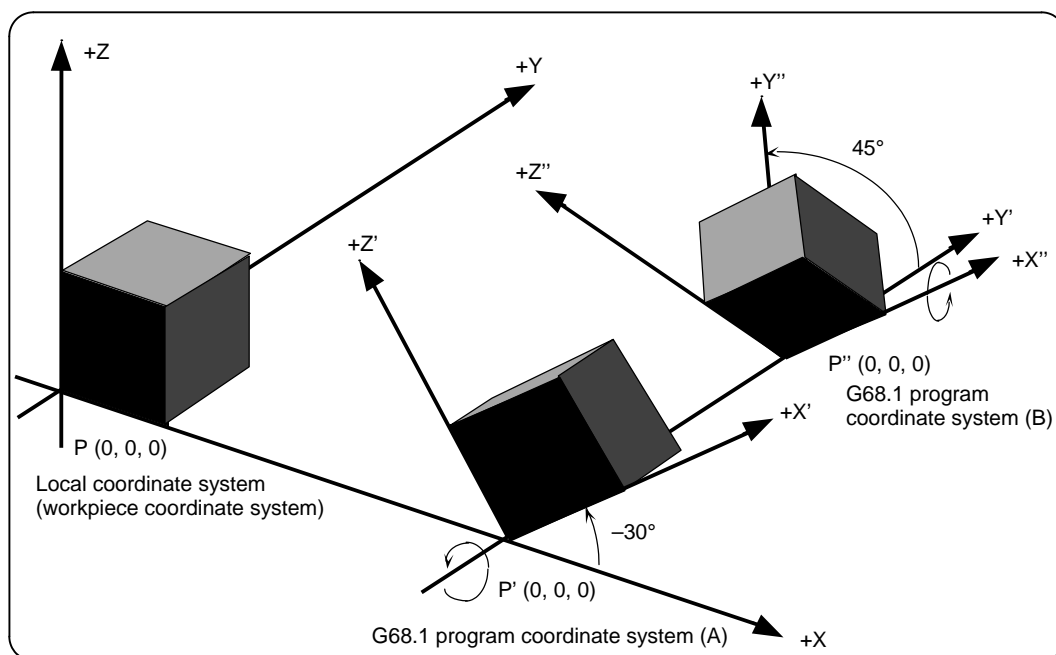
(b) Three-dimensional coordinate conversion and program coordinate system

With the G68.1 command, the rotation center coordinates are shifted by the command value (X, Y, Z) in respect to the current coordinate system. The new program coordinate system rotated by the commanded rotation angle (R) in respect to the command rotation center direction (I, J, K) is set.

(Example)

```
N1 G68.1 X10. Y0 Z0 I0 J1 K0 R-30. ;  
N2 G68.1 X0 Y10. Z0 I1 J0 K0 R45. ;  
N3 G69.1 ;
```

- With N1, the center is shifted by $[x, y, z] = (10., 0, 0)$ in respect to the currently set coordinate system. The new program coordinate system (A in the following figure) rotated -30° in the counterclockwise direction using the Y axis as the center is set.
- With N2, the center is shifted by $[x, y, z] = (0, 10., 0)$ in respect to the newly set program coordinate system (A in the following figure). The new program coordinate system (B in the following figure) rotated $+45^\circ$ in the counterclockwise direction using the X axis as the center is set.
- With N3, the program coordinate systems that have been set are all canceled, and the state where the first G68.1 is commanded is returned to.



(Note) If the local coordinate system offset is not applied, the local coordinate system is the same as the workpiece coordinate system.

13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(c) Conversion method

The coordinate values (Xp, Yp, Zp) in the newly set coordinate system and the coordinate values (Xm, Ym, Zm) in the reference workpiece coordinate system are converted as shown below.

- **First G68.1 command**

$$[X_m, Y_m, Z_m, 1] = [X_p, Y_p, Z_p, 1] R_1 T_1 \dots \dots \dots \text{(Forward row)}$$

$$[X_p, Y_p, Z_p, 1] = [X_m, Y_m, Z_m, 1] (T_1^{-1}) (R_1^{-1}) \dots \dots \dots \text{(Reverse row)}$$

- **Second and following G68.1 command**

$$[X_m, Y_m, Z_m, 1] = [X_p, Y_p, Z_p, 1] R_2 T_2 R_1 T_1$$

$$[X_p, Y_p, Z_p, 1] = [X_m, Y_m, Z_m, 1] (T_1^{-1}) (R_1^{-1}) (T_2^{-1}) (R_2^{-1})$$

R1, R2: Rotation row calculated from first and second G68.1 parameter
 T1, T2: Movement row calculated from first and second G68.1 parameter

The conversion rows Rn and Tn (n = 1, 2) are as follow.

Rn conversion row

1) I designation (rotation around X axis)

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & \cos R & \sin R & 0 \\ 0 & -\sin R & \cos R & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

2) J designation (rotation around Y axis)

$$\begin{bmatrix} \cos R & 0 & -\sin R & 0 \\ 0 & 1 & 0 & 0 \\ \sin R & 0 & \cos R & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

3) K designation (rotation around Z axis)

$$\begin{bmatrix} \cos R & \sin R & 0 & 0 \\ -\sin R & \cos R & 0 & 0 \\ 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

Tn conversion row

$$\begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 \\ 0 & 0 & 1 & 0 \\ x & y & z & 1 \end{bmatrix}$$

x, y, z : Rotation center coordinates (parallel movement amount)

I, J, K : Rotation axis selection

R : Rotation angle

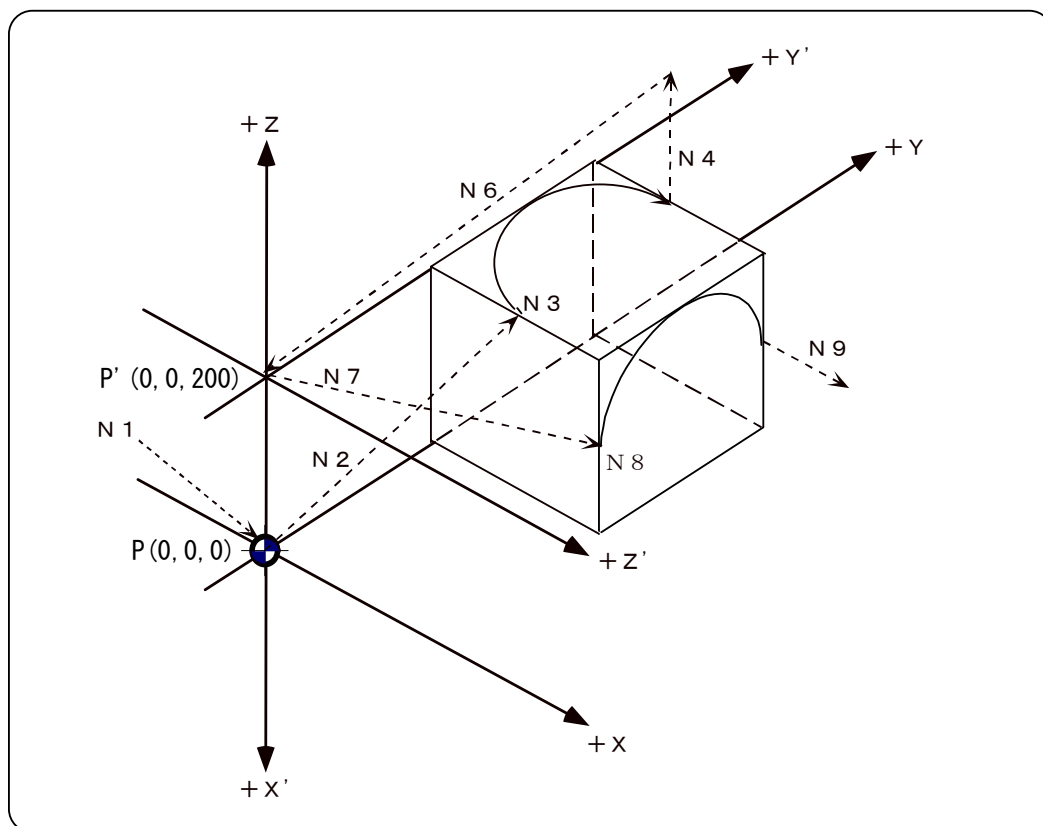
13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(d) Example of program

- 1) Example of machining program using arc cutting
 In the following program example, the arc cutting (N3 block) carried out on the top of the workpiece is also carried out on the side of the workpiece. By using three-dimensional coordinate conversion, the side can be cut with the same process (N8 block).

N01 G17 G90 G00 X0 Y0 Z0;	Position to the workpiece zero point P.
N02 G00 X100. Y200. Z200.;	Move to (100, 200, 200) with rapid traverse.
N03 G02 X100. Y400. J100. F1000;	Carry out arc cutting on workpiece top.
N04 G00 Z300.;	Escape in +Z direction at +100 rapid traverse.
N05 G68.1 X0 Y0 Z200. I0 J1 K0 R90.;	After shifting program coordinate system to (0, 0, 200), rotate coordinate axis +90° in Y axis direction.
N06 G17 G90 G00 X0 Y0 Z0;	Position to new program's zero point P'.
N07 G00 X100. Y200. Z200.;	Move to program coordinate system (100, 200, 200) and workpiece coordinate system (200, 200, 100) with rapid traverse.
N08 G02 X100. Y400. J100. F1000;	Carry out arc cutting on workpiece side.
N09 G00 Z300.;	Move +100 in program coordinate system + Z' direction with rapid traverse.
N10 G69.1;	
N11 M02;	

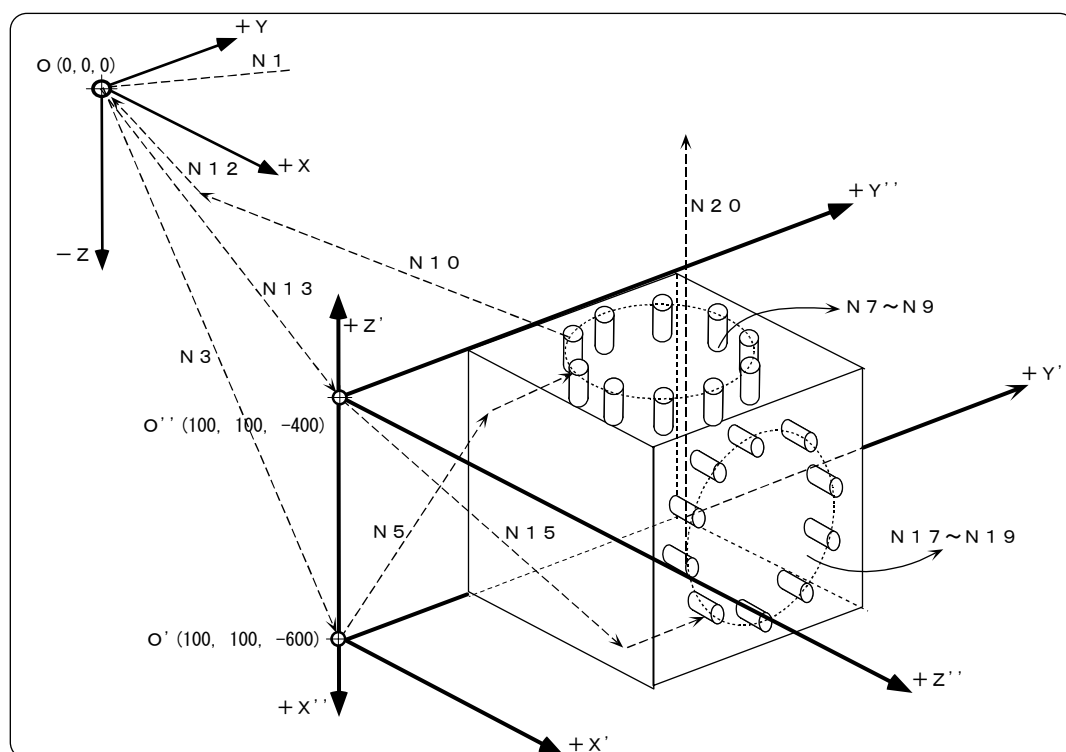


13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

- 2) Example of machining program using fixed cycle
 In the following program, the bolt hole cycle (N08 block) executed on the top of the workpiece is also carried out on the side of the workpiece. By using three-dimensional coordinate conversion, the side can be cut with the same process (N18 block).

N01 G90 G00 X0 Y0 Z0;	Position to the workpiece coordinate system's 1st workpiece zero point.
N02 F2000;	
N03 G00 X100. Y100. Z-600.;	Move to (100, 100, -600) with rapid traverse.
N04 G52 X100. Y100. Z-600.;	Set the local coordinate system to the (100, 100, -600) position.
N05 G00 X100. Y10. Z 200.;	Move to the local coordinate system's (100, 10, 200) position with rapid traverse.
N06 G91;	Incremental value command
N07 G81 Z-10. R5. L0 F2000;	Drilling
N08 G34 X100. Y200. I90. J270. K10.;	Bolt hole cycle
N09 G80;	Drilling cancel
N10 G91 G00 X-200.;	Move -200 from machining end point to X axis direction with rapid traverse.
N11 G90 G52 X0 Y0 Z0;	Cancel local coordinate system.
N12 G90 G00 X0 Y0 Z0;	Position to workpiece zero point.
N13 G00 X100. Y100. Z-400.;	Move to (100, 100, -400) with rapid traverse.
N14 G68.1 X100. Y100. Z-400. I0 J1 K0 R90.;	Set G68 program coordinate system to position rotated +90° in Y axis direction using (100, 100, -400) position as center.
N15 G00 X100.Y10. Z200.;	Move to (100, 10, 200) position in G68 program coordinate system with rapid traverse.
N16 G91;	Incremental value command
N17 G81 Z-10. R5. L0 F200;	Drilling
N18 G34 X100.Y200. I90. J270. K10.;	Bolt hole cycle
N19 G80;	Drilling cancel
N20 G91 G00 X-200.;	Move -200 from machining end point to X axis direction with rapid traverse.
N21 G69.1;	Cancel three-dimensional coordinate conversion modal.
N22 M02;	End program.



13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(e) Coordinate system

- 1) By issuing the three-dimensional coordinate conversion command, the new coordinate system (G68.1 program coordinate system) will be created on the local coordinate system.
- 2) The coordinate system for the three-dimensional coordinate conversion rotation center coordinates is the local coordinate system. Thus, these coordinates are affected by the following coordinate system offset and coordinate system shift amount.
 - Local coordinate system offset issued with G52 command
 - G92 shift amount issued with G92 command
 - Coordinate system offset corresponding to workpiece coordinate system selected with command
 - External workpiece coordinate offset
 - Extended workpiece offset
- 3) During the three-dimensional coordinate conversion modal, the G68.1 program coordinate system is created on the current G68.1 program coordinate system, and is used as the new G68.1 program coordinate system.
- 4) The local coordinate system cannot be created (G52) on the G68.1 program coordinate system. (If G52 is issued, the "PR0921 3DIM coord convert error" will occur.)

(f) Rotation angle

- 1) For the rotation angle, the counterclockwise direction looking at the rotation center from the plus direction of the rotation center axis is the plus (+) direction.
- 2) If a value with no decimal point is designated for the rotation angle, the value obtained by dividing the designated value by 1000 will be used.
When either of the following are designated, the rotation angle will be 45.0°.
G68.1 X0.Y0.Z0. I0J0K1 R45.
G68.1 X0.Y0.Z0. I0J0K1 R45000
- 3) If the rotation angle is omitted, the rotation angle will be handled as 0°.
- 4) The modal display's rotation angle for three-dimensional coordinate conversion overwrites the modal display's rotation angle for the program coordinate conversion.
- 5) The modal display's rotation angle when three-dimensional coordinate conversion is carried out two or more times, will be the rotation angle of the three-dimensional coordinate conversion carried out last.

(g) Rotation center coordinates

- 1) The G68.1 rotation center coordinate system is commanded with the local coordinate system (G68.1 program coordinate system during the three-dimensional coordinate conversion modal).
- 2) The rotation center coordinate designation is handled as an absolute value designation regardless of whether an absolute/incremental modal (G90/G91) is being executed.
- 3) If the rotation center coordinate (address X, Y, Z) is omitted, it will be handled as if the zero point for the current local coordinate (G68.1 program coordinate system during the three-dimensional coordinate conversion modal) is designated for the omitted address's axis. (The same as when 0 is just set.)

13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(h) G68.1 multiple command

By commanding three-dimensional coordinate conversion during the three-dimensional coordinate conversion modal, two or more multiple commands can be issued.

- 1) The three-dimensional coordinate conversion command in the three-dimensional coordinate conversion modal is combined with the conversion in the modal.
- 2) If the three-dimensional coordinate conversion is overlapped during the three-dimensional coordinate conversion modal, the overlapped three-dimensional coordinate conversion will be created on the coordinate system (G68.1 program coordinate system) created with the three-dimensional coordinate conversion in the modal. Thus, the rotary axis and coordinates must be designated with this G68.1 program coordinate system.

For the X axis and Y axis, if each is created on the coordinate system rotated by 90°, these must be designated as shown in Example 2 rather than Example 1.

<Example 1>

G68.1 X0.Y0.Z0. I1J0K0 R90. X axis rotation 90°

G68.1 X0.Y0.Z0. I0J1K0 R90. Y axis rotation 90°

(The Y axis designated here is the same as the Z axis in the original coordinate system.)

<Example 2>

G68.1 X0.Y0.Z0. I1J0K0 R90. X axis rotation 90°

G68.1 X0.Y0.Z0. I0J0K1 R-90. Z axis rotation 90°

(The Z axis -90 rotation designated here is the same as the Y axis +90 rotation in the original coordinate system.)

13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

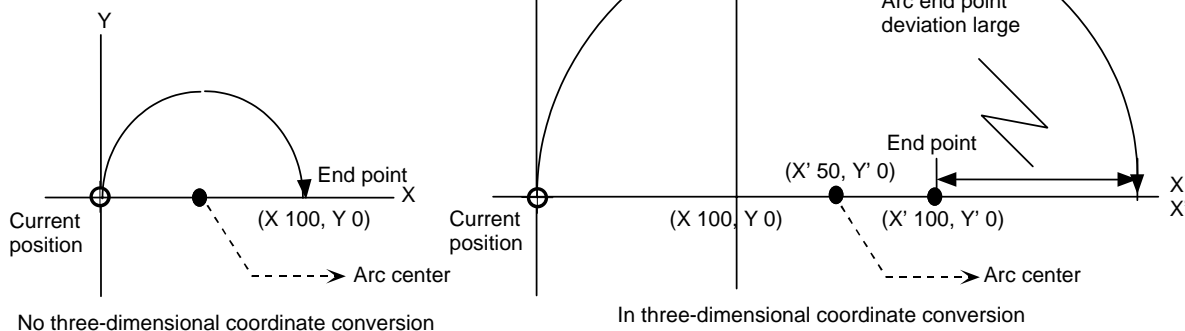
(i) When first movement command after three-dimensional coordinate conversion command is an arc command

When the first movement command after the three-dimensional coordinate conversion command is an arc shape, if the arc center position has not changed before and after the three-dimensional coordinate conversion, the arc will be drawn. However, an error will occur in the following cases.

- 1) If three-dimensional coordinate conversion of an arc center designated (I, J designated) arc causes the center coordinates to deviate, the error "PR0070 Arc end point deviation large" will occur.
- 2) If three-dimensional coordinate conversion of an arc radius designated (R designated) arc causes the center coordinates to deviate, the error "PR0071 Can't calculate arc center" will occur.

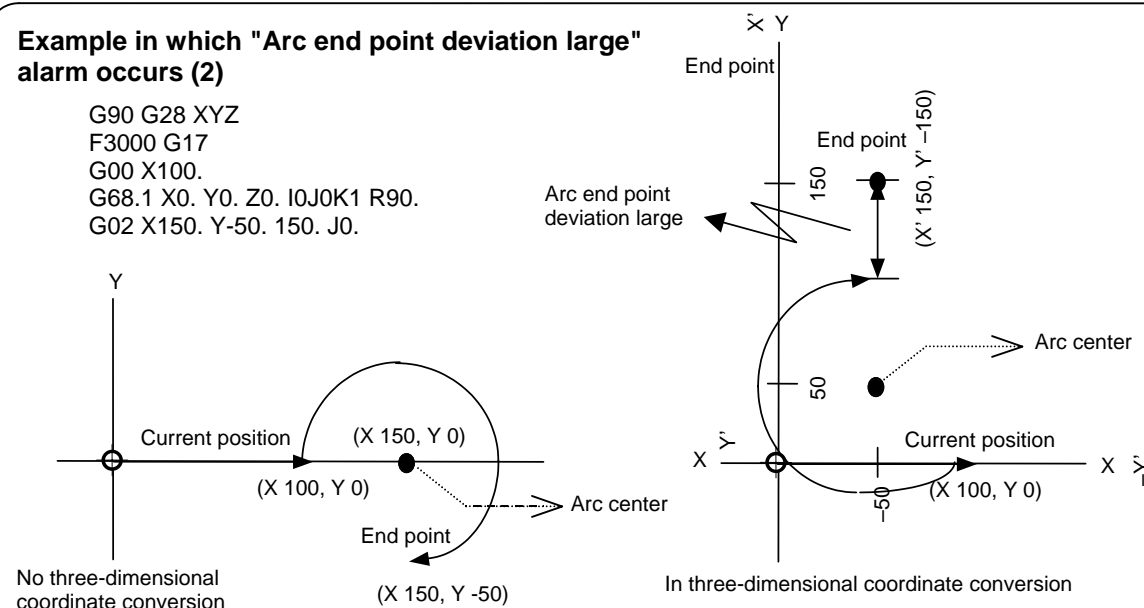
Example in which "Arc end point deviation large" alarm occurs (1)

```
G90 G28 XYZ
F3000 G17
G68.1 X100. Y0. Z0. I0J0K1 R0.
G02 X100. I50.
```



Example in which "Arc end point deviation large" alarm occurs (2)

```
G90 G28 XYZ
F3000 G17
G00 X100.
G68.1 X0. Y0. Z0. I0J0K1 R90.
G02 X150. Y-50. 150. J0.
```

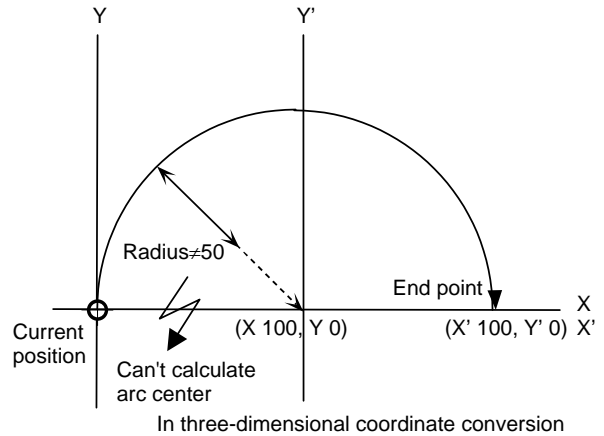
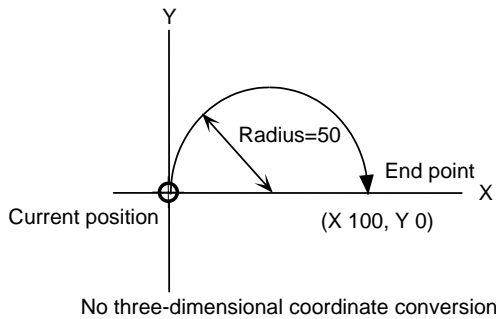


13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

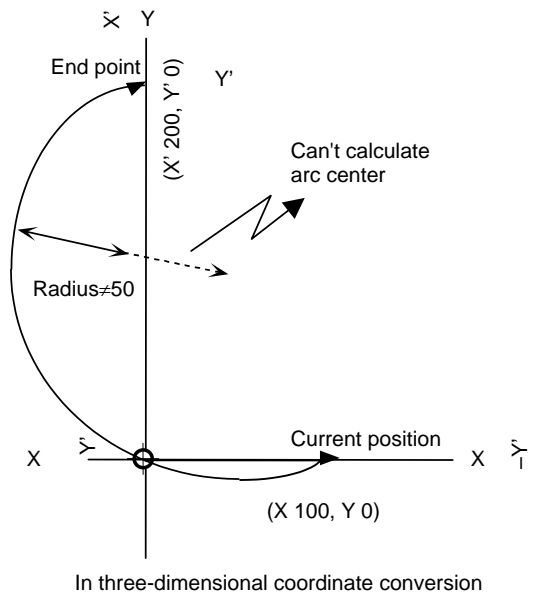
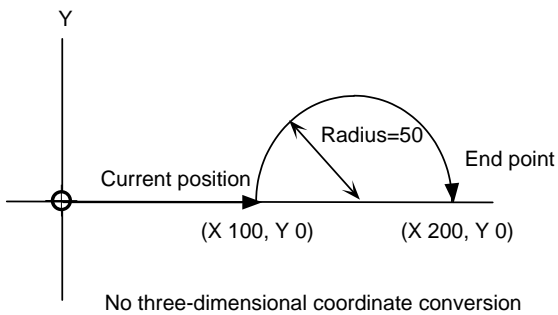
Example in which "Can't calculate arc center" alarm occurs (1)

```
G90 G28 XYZ
F3000 G17
G68.1 X100. Y0. Z0. I0J0K1 R0.
G02 X100. R50.
```



Example in which "Can't calculate arc center" alarm occurs (2)

```
G90 G28 XYZ
F3000 G17
G00 X100.
G68.1 X0. Y0. Z0. I0J0K1 R90.
G02 X200. Y0. R50.
```



13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(j) Three-dimensional coordinate conversion for axis with no movement command

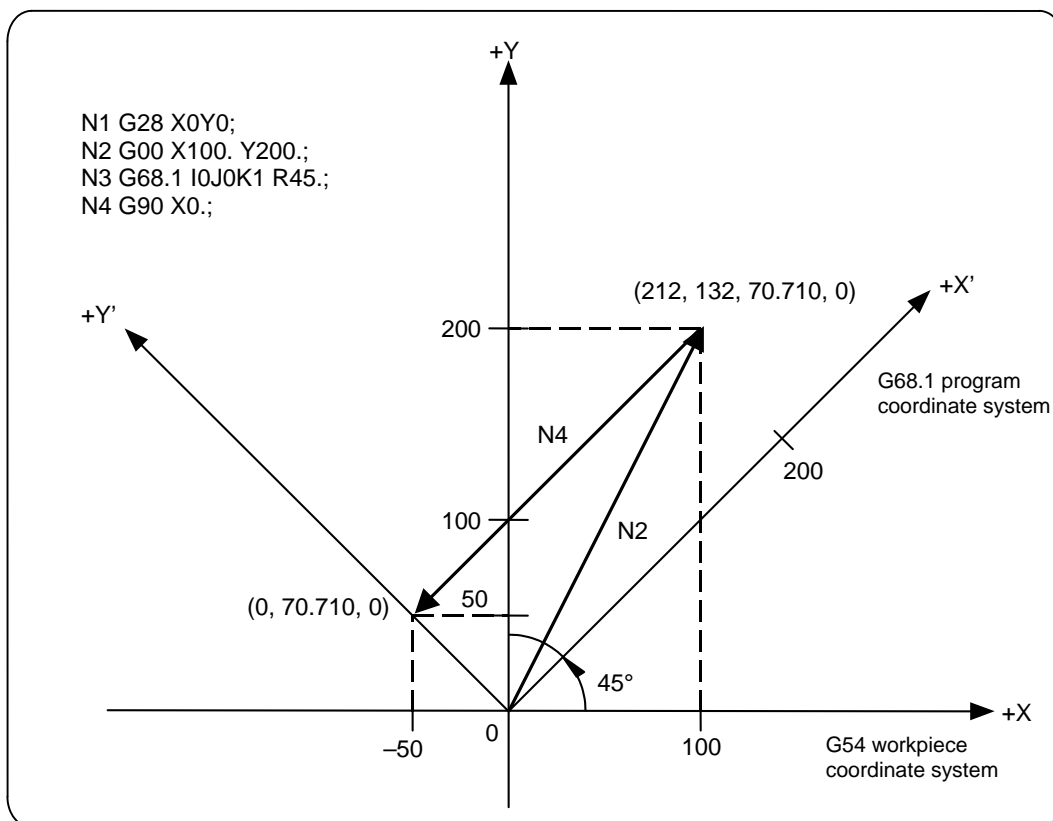
An axis with no movement command will not move.

<Example: When G90 movement is commanded immediately after three-dimensional coordinate conversion>

In the following example, X axis movement is commanded in the N4 block, but movement is not commanded for the other axes.

Here, the coordinates before movement in the N4 block are (X 212.132, Y 70.710, Z 0) in the G68.1 program coordinate system. If X0 movement in the N4 block is attempted on this coordinate system, the end point will be (X0, Y 70.710, Z0).

When viewed on the coordinate system before three-dimensional coordinate conversion, these coordinates are (X -50, Y50, Z0).



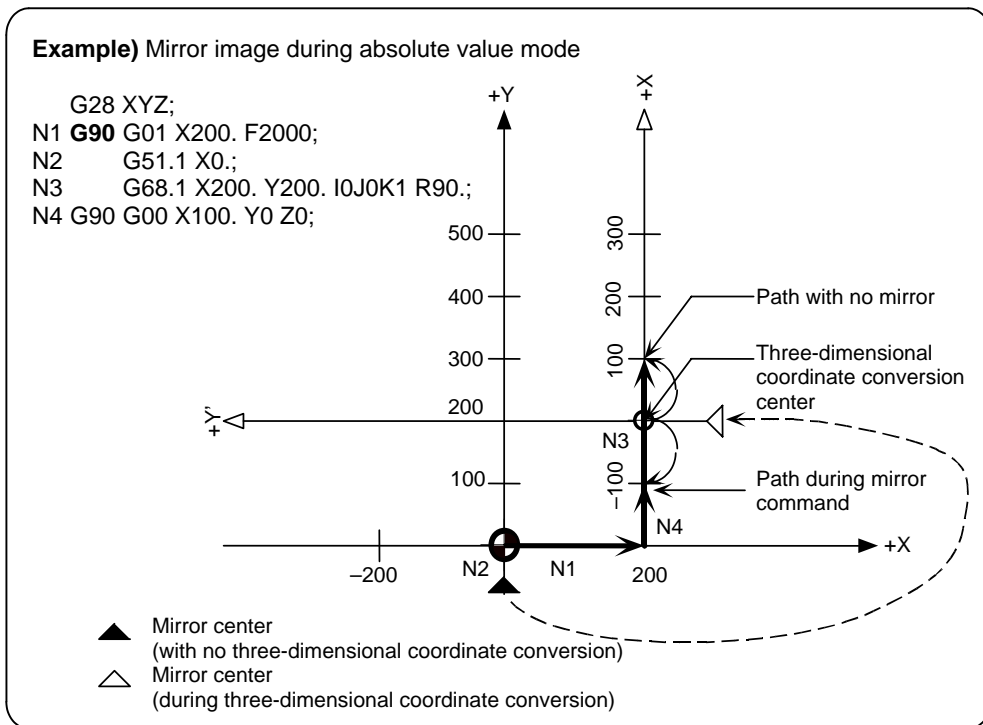
13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

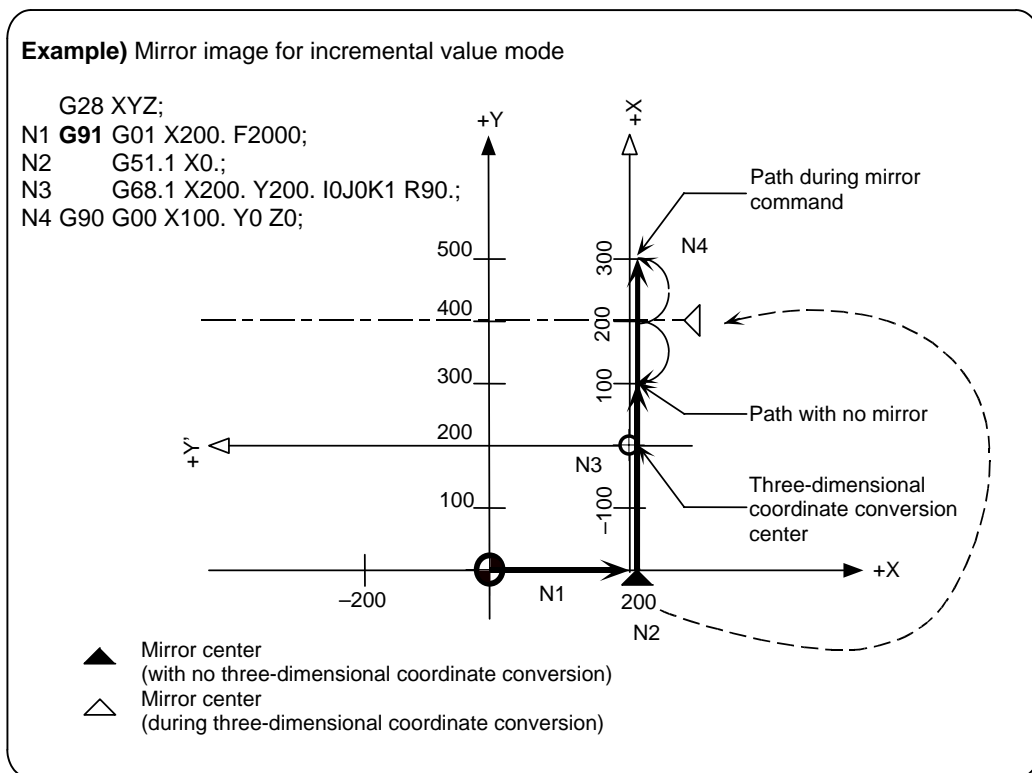
(k) When three-dimensional coordinate conversion is commanded during mirror image

If three-dimensional coordinate conversion is commanded during mirror image, the mirror will not be applied on the rotary conversion center coordinates. Thus, the coordinate value (absolute position) designated in the machining program will be used during coordinate conversion.

The mirror center is converted over the converted coordinate system (G68.1 program coordinate system).



(X, Y: before conversion/X', Y': after conversion)



(X, Y: before conversion/X', Y': after conversion)

13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

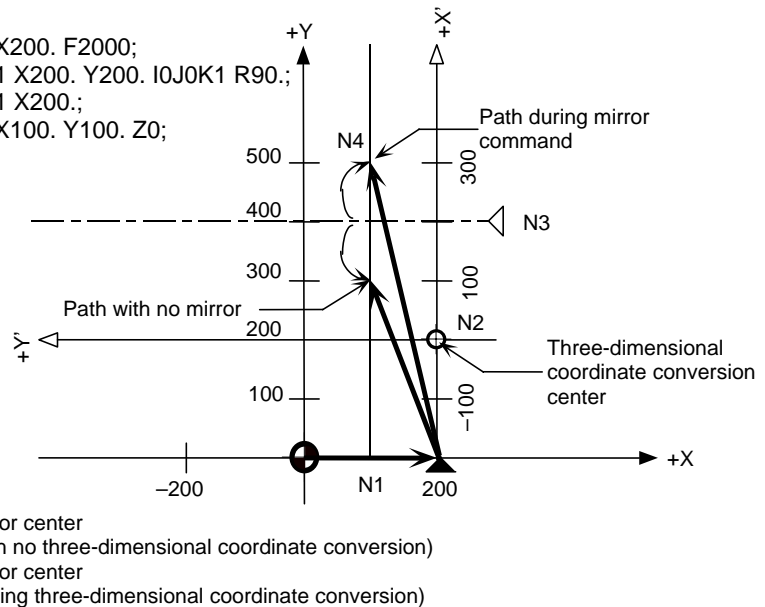
(I) When mirror image is used during three-dimensional coordinate conversion command

If the G command mirror image is applied during the three-dimensional coordinate conversion command, the mirror center will be set with the G68.1 program coordinate system.

If the mode is the absolute value mode, the mirror center will be set with the absolute position in the G68.1 program coordinate system. If the mode is the incremental value mode, the mirror center will be set as a position obtained by adding the incremental value designated in the G68.1 program coordinate system to the current position.

Example) Mirror image for incremental value mode

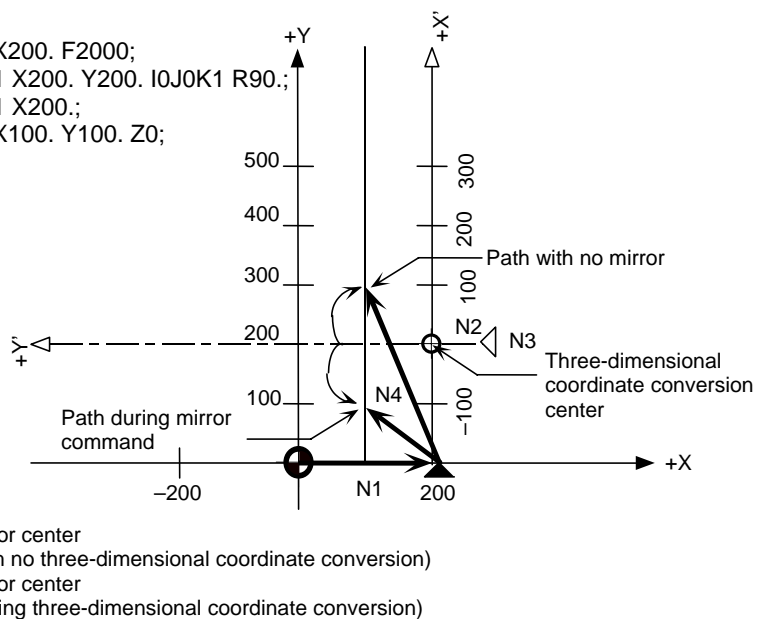
```
G28 XYZ;
N1 G90 G01 X200. F2000;
N2 G68.1 X200. Y200. IOJK1 R90.;
N3 G51.1 X200.;
N4 G90 G01 X100. Y100. Z0;
```



The mirror center is set at the absolute value $X = 200$ in the G68 program coordinate system.
 (X, Y: before conversion/ X' , Y' : after conversion)

Example) Mirror image for incremental value mode

```
G28 XYZ;
N1 G91 G01 X200. F2000;
N2 G68.1 X200. Y200. IOJK1 R90.;
N3 G51.1 X200.;
N4 G90 G01 X100. Y100. Z0;
```



The mirror center is a position $+200$ in the X direction from the current position in the G68 program coordinate system.
 (X, Y: before conversion/ X' , Y' : after conversion)

13. PROGRAMMING SUPPORT FUNCTIONS
13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(4) Relation with other functions

(a) G codes that can be commanded during three-dimensional coordinate conversion mode.
 G00, G01, G02, G03, G04, G05, G6.2, G07, G09, G10, G11, G11.1, G12, G13, G17, G18, G19, G20, G21, G22, G22.1, G23, G23.1, G27, G28, G29, G30, G31, G31.1, G31.2, G31.3, G31.4, G34, G35, G36, G37, G38, G39, G40, G41, G42, G43, G44, G45, G46, G47, G48, G50.1, G51.1, G53, G61, G61.1, G61.2, G61.3, G61.5, G62, G64, G65, G66, G66.1, G67, G68, G69.1, G70, G71, G72, G73, G74, G75, G76, G77, G78, G79, G80, G81, G82, G83, G84, G85, G86, G87, G88, G89, G90, G91, G93, G94, G95, G98, G99

(b) Circular interpolation

- 1) During the three-dimensional coordinate conversion modal, the arc interpolation will operate with the three-dimensional coordinate converted value.
- 2) The arc command during three-dimensional coordinate conversion will operate normally for all planes G17, G18 and G19.
- 3) Even if the three-dimensional coordinate conversion is only rotated on the selected plane (when the same function as program coordinate conversion is executed), the original shape before conversion will be drawn with the graphic check.

(c) Z axis cancel

Z axis cancel during the three-dimensional coordinate conversion modal is movement in respect to the coordinate value after three-dimensional coordinate conversion. The movement in the Z axis direction will be canceled.

In Example 1, there is movement in the Z axis direction in the original figure. However, after three-dimensional coordinate conversion, the movement is converted to movement in the X axis direction, so the axis will move.

<Example 1>

```
G90 G28 XYZ
G68.1 X0. Y0. Z0. I0J1K0 R90. Y axis rotation 90°
G01 Z100.           ( 0, 0, 100) → (100, 0, 0) → (100, 0, 0)
```

Coordinate before conversion	Coordinate after conversion	Movement destination coordinate
---------------------------------	--------------------------------	------------------------------------

In Example 2, there is no movement in the Z axis direction in the original figure. However, movement in the Z axis direction will be generated after three-dimensional coordinate conversion.

If the mode is the Z axis cancel mode, this Z axis direction movement will be ignored and no movement will take place.

<Example 2>

```
G90 G28 XYZ
G68.1 X0. Y0. Z0. I0J1K0 R90. Y axis rotation 90°
G01 X100. Y100.     (100, 100, 0) → ( 0, 100, -100) → ( 0, 100, 0)
```

Coordinate before conversion	Coordinate after conversion	Movement destination coordinate
---------------------------------	--------------------------------	------------------------------------

13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(d) Graphic check

- 1) During graphic check, the original shape before conversion is drawn as the shape being three-dimensional coordinate converted.
- 2) When three-dimensional coordinate conversion is canceled, the current position in the graphic check is aligned with the actual machine position.
- 3) If machining program with incremental value movement is executed after three-dimensional coordinate conversion is canceled, the position of the shape after three-dimensional coordinate conversion is canceled may be deviated from the original shape.

This occurs because movement is executed with an incremental value command while the position of the shape after three-dimensional coordinate conversion is canceled is deviated from the original shape.

<Example>	Tool path	Graphic check path	Original path
N1 G90 G28 XYZ	0, 0, 0	0, 0, 0	0, 0, 0
N2 F3000			
N3 G01 X100.	100, 0, 0	100, 0, 0	100, 0, 0
N4 G68.1 X0Y0Z0 I1J0K0 R90.	<Three-dimensional coordinate conversion start>		
N5 G01 Y100.	100, 0, 100	100, 100, 0	100, 100, 0
N6 G01 X0.	0, 0, 100	0, 100, 0	0, 100, 0
N7 G69.1	<Three-dimensional coordinate conversion cancel>		
N8 G91 Y-100.	0, -100, 100	0, -100, 100	0, 0, 0
N9 G90 X0.Y0.Z50.	0, 0, 50	0 0, 50	0, 0, 50
N10 M02			

Normally, the path drawn with graphic check is the original shape onto which three-dimensional coordinate conversion has not been applied. However, when three-dimensional coordinate conversion is canceled, the graphic check position is aligned with the machine position, so if the canceled position is different from the original shape, it will be deviated from the original shape.

In the above examples, the graphic check's current position (original shape's current position) in N6 is (0,100,0), but when three-dimensional coordinate conversion is canceled in N7, the position is aligned to the current machine position (0,0,100).

Thus, if the next movement command is an incremental value command (N8 block), the axis will move to a position deviated from the original shape.

This deviation can be corrected by moving with an absolute value command. (N9 block)

(e) High-accuracy control/initial high-accuracy

Three-dimensional coordinate conversion and high-accuracy control can be used simultaneously.

(f) NURBS interpolation

Three-dimensional coordinate conversion and NURBS interpolation can be used simultaneously. If the three-dimensional coordinate conversion command (G68.1) and NURBS command (G6.2) are issued in the same block, the error "PR0033 Format error" will occur.

13. PROGRAMMING SUPPORT FUNCTIONS

13.18 Three-dimensional Coordinate Conversion Function; G68.1/G69.1

(g) Synchronous tap

The synchronous tap command can be issued in the three-dimensional coordinate conversion modal.

The synchronous tap feed direction will be the direction commanded on the G68.1 program coordinate system by the three-dimensional coordinate conversion modal.

Even if manual synchronous tap is executed during the three-dimensional coordinate conversion modal, the feed direction and the spindle will not be synchronized.

(h) High-speed skip/multi-step skip

High-speed skip and multi-step skip can be executed in the three-dimensional coordinate conversion modal.

Command the target coordinate value as a coordinate value on the G68.1 program coordinate system.

The coordinate value in the G68.1 program coordinate system will be output as soon as the skip signal is input in the skip coordinates (macro variables). A coordinate value in the local coordinate system before G68.1 is commanded can be output by setting the parameters.

If manual skip is used during the three-dimensional coordinate conversion, the local coordinate value will be output.

If the skip command (G31, G31.1, G31.2, G31.3, G31.4) is commanded in the same block as G68.1, only the G68.1 command will be valid, and the skip command will be ignored.

13.19 Cutting Mode; G64

(1) Outline

When the cutting mode command is issued, the NC will enter the cutting mode that allows a smooth cutting surface to be achieved. As opposed to the exact stop check mode (G61), the operation will not decelerate and stop between cutting feed blocks. Instead, the next block will be continuously executed. The cutting mode state is entered when the power is turned ON.

(2) Command format

G code	Function
G64	Cutting mode ON

The cutting mode can be canceled with the exact stop check mode (G61), automatic corner override (G62), tapping mode (G63), or high-accuracy control mode (G61.1 to G61.5).

13.20 Automatic Corner Override; G62

(1) Outline

This function automatically applies an override on the cutting feedrate so that the cutting amount is not increased per constant time at the corner. This prevents strain on the machining surface due to an increase in the cutting load when cutting corners.

The automatic corner override is valid only during tool radius compensation.

(2) Command format

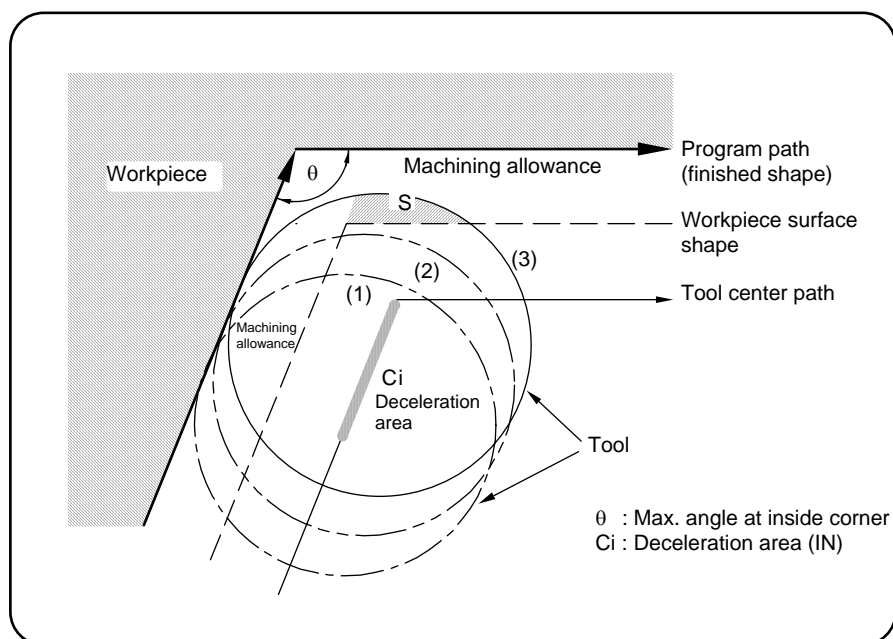
G62 ;

The automatic corner override turns ON with the G62 command, and is canceled with the following G commands.

G40 Tool radius compensation cancel
 G61 Exact stop check mode
 G63 Tapping mode
 G64 Cutting mode
 G61.1~G61.5 . . . High-accuracy control mode

(3) Detailed description

(a) Description of operation



- 1) When automatic corner override is not to be applied
 When the tool moves in the order of (1) → (2) → (3) shown above, the machining allowance at (3) increases by an amount equivalent to the area of shaded section S and so the tool load increases.
- 2) When automatic corner override is to be applied
 When the inside corner angle θ shown above is less than the angle set in the parameter, the override set into the parameter is automatically applied in the deceleration range C_i .

13.21 Tapping Mode; G63

(1) Outline

With this function, the NC state can be set the following type of control mode suitable for tapping. This is effective for creating a tapping program without using a fixed cycle program.

- Cutting override is fixed at 100%.
- Deceleration commands at joints between blocks are invalid.
- Feed hold is invalid.
- Single block is invalid.
- In-tapping mode signal is output.

(2) Command format

G63 ;

G63 is released by the exact stop mode (G61), automatic corner override (G62), or cutting mode (G64), or high-accuracy control mode (G61.1, G61.2, G61.3, G61.5) command.

(3) Detailed description

- If the exact stop mode (G61), automatic corner override (G62), cutting mode (G64) and high-accuracy control mode (G61.1, G61.2, G61.3, G61.5) are commanded in the same block, the G code commanded last will have the priority.
- The dry run signal is also invalid during the tapping mode.
- The error detect signal is also invalid during the tapping mode.
- Block stop will not be applied during the tapping mode, so even compare stop is designated for a tapping mode block, the operation will not stop.

(4) Notes

- Even if the cutting feed override is set to 0%, the movement will not stop during the tapping mode.
- If feed hold is input during the tapping mode, the feed hold will be invalid. However, block stop will be applied at the end point of the first block after the tapping mode.

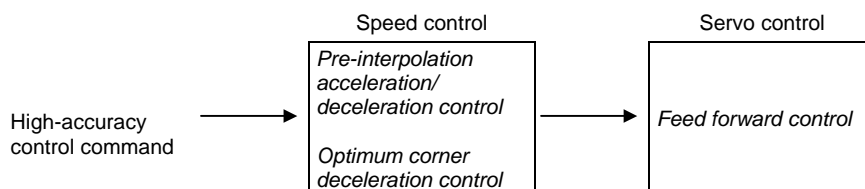
13. PROGRAMMING SUPPORT FUNCTIONS
13.22 High-accuracy Control; G61.1

13.22 High-accuracy Control; G61.1

(1) Outline

This function controls the operation so the lag is eliminated in control systems and servo systems. With this function, improved machining accuracy can be realized, especially during high-speed machining, and machining time can be reduced.

The high-accuracy control function is configured of the following functions.



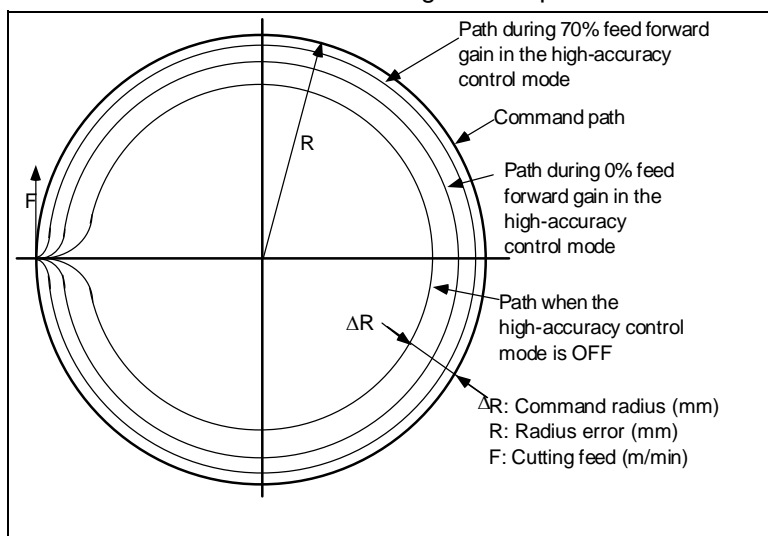
Function	Cutting feed	Rapid traverse positioning
Pre-interpolation acceleration/deceleration	Acceleration/deceleration time is reduced using constant inclination acceleration/deceleration and a S-character filter, and surface accuracy is improved by suppressing machine vibration.	Positioning time for micro-distances is reduced.
Optimum corner deceleration	Sharp corners are accurately machined without waste.	
Feed forward control	Machining path accuracy is improved by reducing the tracking lag error.	Positioning time is reduced by reducing the tracking lag error.

(2) Command format

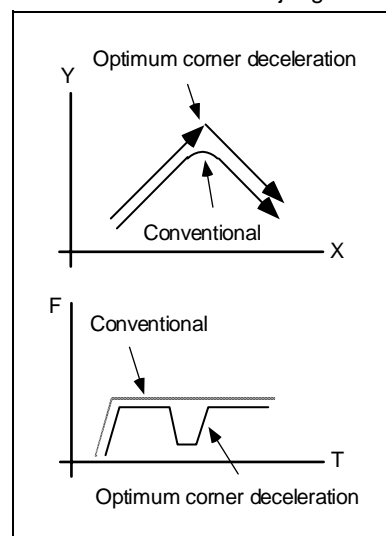
```
G61.1 ;
```

High-speed control is commanded with G61.1.

G02/G03 Effect during arc interpolation



Sharp corners are accurately machined without waste by optimum linear acceleration/deceleration and corner judgment.



13. PROGRAMMING SUPPORT FUNCTIONS

13.22 High-accuracy Control; G61.1

(3) Detailed description

(a) Pre-interpolation acceleration/deceleration

The machining shape error due to smoothing is eliminated by accelerating/decelerating before interpolation, and a highly accurate path can be obtained.

Decreased radius errors during arc commands can be kept extremely low.

Constant inclination acceleration/deceleration is also carried out, so the positioning time for micro-distances during G00 commands is reduced.

(Note) Constant pre-interpolation acceleration/deceleration for rapid traverse (G00) can be validated or invalidated in the parameters. This can be done independently from the high-accuracy control designation.

Refer to the Section on "High-accuracy Control for Part Machining" for details on improving the arc radius error.

(b) Optimum corner deceleration

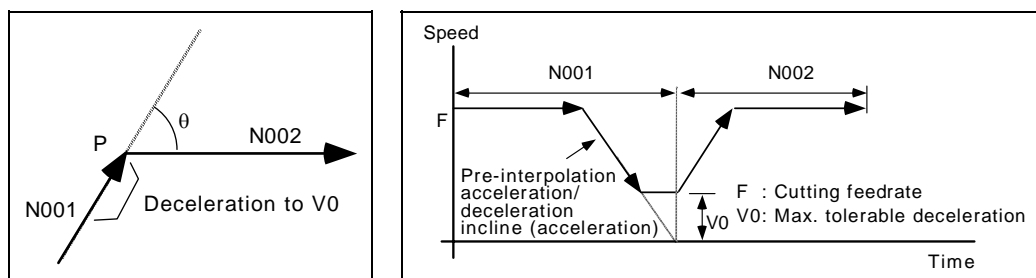
Highly accurate edge machining is enabled by judging the machining program command vectors and carrying out corner deceleration.

The following drawing shows the deceleration pattern at the corner.

(Optimum corner deceleration is a function of the high-accuracy control mode.)

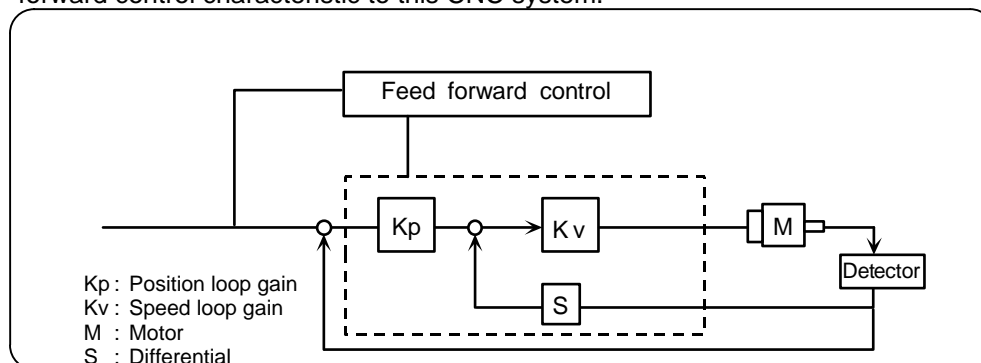
By using an S-character filter, the speed change becomes smoother, machine vibration is suppressed, and surface accuracy can be improved.

At the corners, the command vectors of the machining program are automatically judged, and corner deceleration is carried out. Highly accurate edge machining is enabled by corner deceleration.



(c) Feed forward control function

A stable servo control with an extremely small servo error can be realized using the feed forward control characteristic to this CNC system.

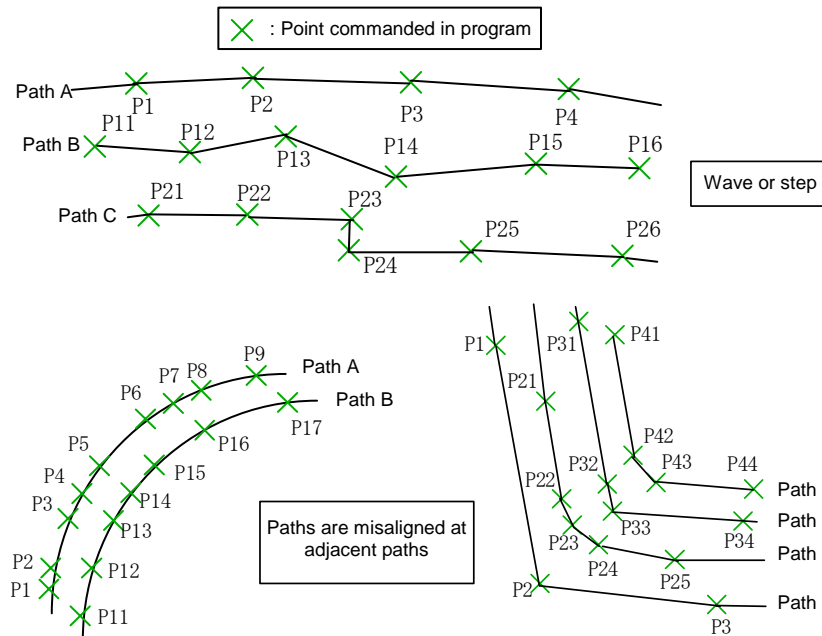


13. PROGRAMMING SUPPORT FUNCTIONS

13.22 High-accuracy Control; G61.1

(d) High-accuracy SS control

A fine segment program for die machining, etc., may contain fine steps or paths waved in a S-pattern caused by CAM operation errors, etc. This may cause the paths to be misaligned in adjacent paths.



If strict speed control, based on the angle of two blocks, is used on a program with this type of step or error, sensitive acceleration/deceleration may be repeated at local steps or errors. As a result, marks like scratch may be made or streaks may appear on the cutting surface.

With SS (Smooth Surface) control, by using the angle of two blocks and the path information of a wide area, optimum speed control not affected excessively by the fine steps or waves can be carried out.

The features of SS control are listed below.

- 1) Effective for machining a smooth shape die with a fine segment program.
- 2) Speed control not easily affected by errors in the path.
- 3) Similar acceleration/deceleration patterns used if shape and accuracy of adjacent paths are similar.
- 4) Even at sections not requiring corner deceleration, speed is clamped if predicted acceleration rate is high.

(Clamp rate is set in the machining parameter (user parameter) "SS ctrl clamp coef".)

The length of the path direction recognized with SS control can be adjusted with the machining parameter (user parameter) "SS ctrl std length". When a large value is set, a wider range will be applied, and the effect of errors will be lessened.

(Note) SS control executes speed control using path information for a wide range. Thus, the number of pre-read blocks will increase. The performance in the high-speed machining mode I, II cannot be guaranteed when SS control is valid.

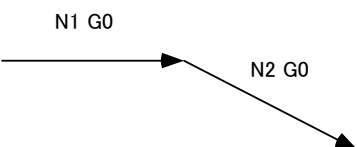
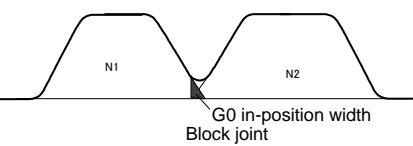

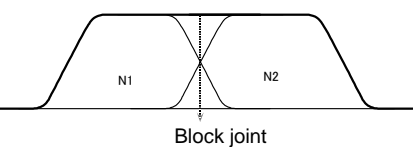
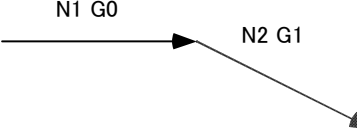
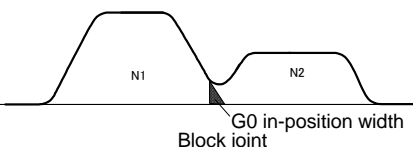

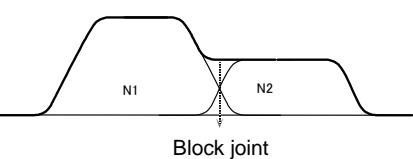
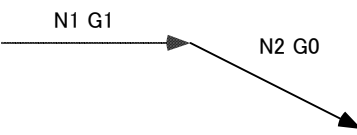
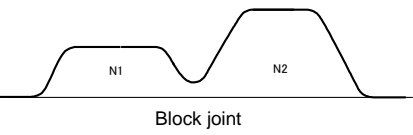
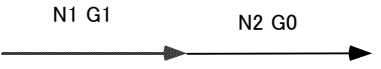
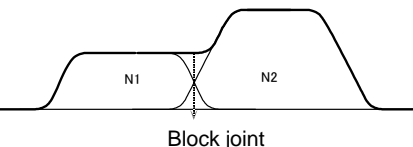
(e) High-accuracy control deceleration check

During the high-accuracy control mode (G61.1, G61.2, G61.3, G61.5), deceleration check is not carried out at the joint of the positioning command (G0) and cutting feed command (G1) blocks. Instead, the corner is judged and when found to be a corner, in-position check is carried out. This allows the machining time to be shortened in a program with many hole drilling steps, etc.

When the control parameter 2 "Prec ctrl dec chk2" is turned ON, if the positioning command (G0) and cutting feed command (G1) movement directions do not change, movement with the cutting feed command (G1) will start before the movement speed with the positioning command (G0) reaches 0. Deceleration check is not carried out at the joint of the cutting feed command (G1) and positioning command (G0) blocks, and instead optimum corner deceleration is carried out.

13. PROGRAMMING SUPPORT FUNCTIONS
13.22 High-accuracy Control; G61.1

1) Operation

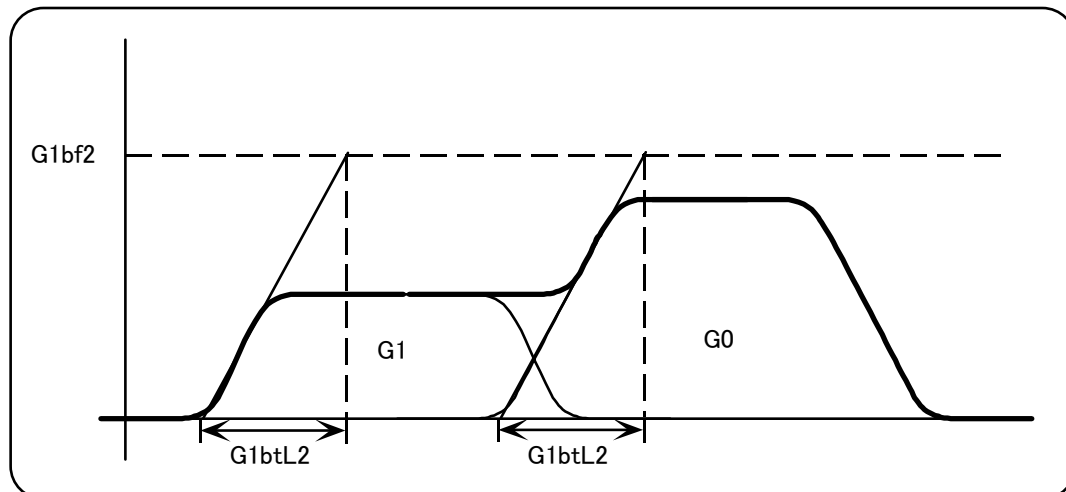
Previous block (N1)	Next block (N2)	Changes in movement direction	Deceleration check at block joint
G0	G0	When movement direction changes 	G0 in-position width is checked. 
G0	G0	When movement direction does not change 	The G0 movement in the next block is started without decelerating. 
G0	G1	When movement direction changes 	G0 in-position width is checked. 
G0	G1	When movement direction does not change 	G0 block decelerates, and after reaching the G1 speed, the next block's G1 movement is started. 
G1	G0	When movement direction changes 	Axis decelerates to corner deceleration speed. 
G1	G0	When movement direction does not change 	Axis does not decelerate at G1 block, and instead next block's G0 movement is started. 

13. PROGRAMMING SUPPORT FUNCTIONS

13.22 High-accuracy Control; G61.1

2) Acceleration/deceleration

When this function is validated (control parameter 2 "Prec ctrl dec chk2" is ON), the inclination of the acceleration/deceleration during the positioning command (G0) and cutting feed command (G1) will be determined by the base system parameter #120090 "G1bf2" and #120091 "G1btL2".



3) G0 feed forward gain

When this function is validated (control parameter 2 "Prec ctrl dec chk2" is ON), the feed forward gain for the positioning command (G0) will be the same as the feed forward gain for the cutting feed command (G1), and will be determined by the axis specification parameter #130021 "Fwdg".

4) Restrictions

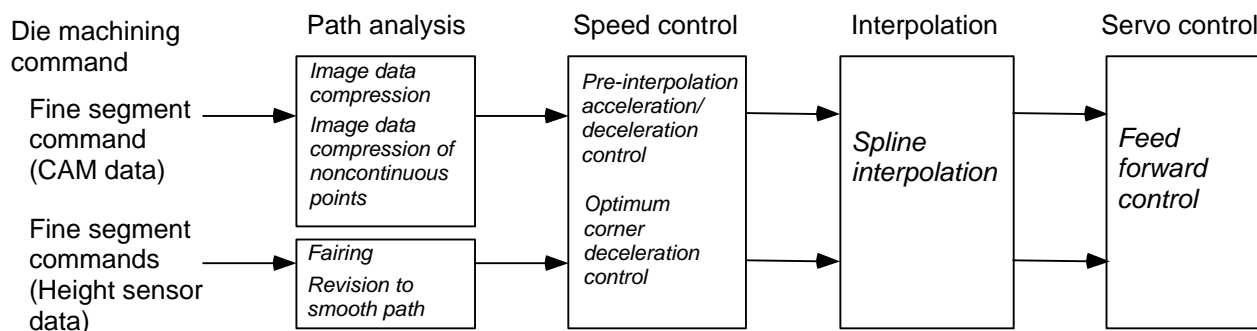
- Deceleration check is carried out when the control parameter 2 #320010 "G0 interpolation OFF" is ON.
- Deceleration check is carried out when error detect is ON.
- Deceleration check is carried out in the exact stop check (G09) command block.
- Deceleration check is carried out during the synchronous tapping mode (G74/G84).
- When this function is valid and the machining time is calculated, the calculated time will be the same as when the function is invalid.
- If there is a block with no movement, such as the M code, during the movement commands, the tool will decelerate by the process time for the block with no movement.

13.23 High-accuracy Control for Die Machining; G61.2/G61.3

(1) Outline

This function allows free curves to be machined at a high-accuracy with fine segment commands. By using this in combination with the high-speed machining function, high-speed and high-accuracy machining can be carried out.

The function for controlling the machining program commanded with fine segments at a high accuracy is configured of the following functions.



Function	Cutting feed	Rapid traverse positioning
Image data compression	Deletion of unnecessary fine noncontinuous data	
Fairing	Revision on noncontinuous command points for smooth machining	
Pre-interpolation acceleration/deceleration	Acceleration/deceleration time is reduced using constant inclination acceleration/deceleration and an S-character filter, and surface accuracy is improved by suppressing machine vibration.	Positioning time for micro-distances is reduced.
Optimum corner deceleration	Neat machining of sharp corners without waste	
Spline interpolation	Smooth, high-speed machining of free curves	
Feed forward control	Machining path accuracy is improved by reducing the tracking lag error.	Positioning time is reduced by reducing the tracking lag error.

(2) Command format

G61.2 ;

The high-accuracy control for die machining (CAM data) is commanded with G61.2.

G61.3 ;

The high-accuracy control for die machining (height sensor data) is commanded with G61.3.

(3) Detailed description

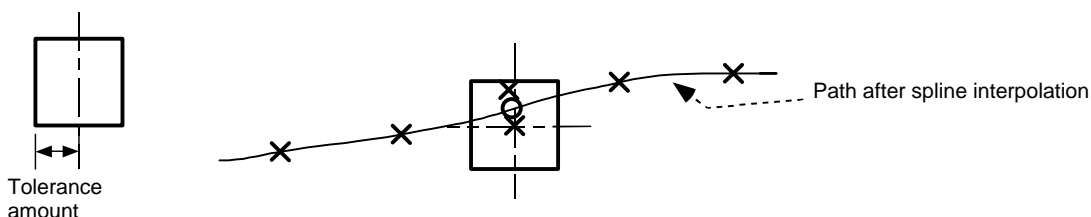
- Cutting feed commands are all spline interpolated during G61.2 and G61.3 modal.
- The following shows function combinations during G61.2 and G61.3 modal.
 - G61.2: Image data compression + Pre-interpolation acceleration/deceleration + Optimum corner deceleration + Spline interpolation + Feed forward
 - G61.3: Image data compression + Fairing + Pre-interpolation acceleration/deceleration + Optimum corner deceleration + Spline interpolation + Feed forward

(a) Fairing function, image data compression function

This function carries out image data compression of the commanded point and revision of the commanded point within the tolerance amount (parameter setting) range. By carrying out spline interpolation on the command point after revision, a smooth machining surface can be achieved with high-speed machining. The corner is smoothly connected within the tolerance range.

For CAD/M data

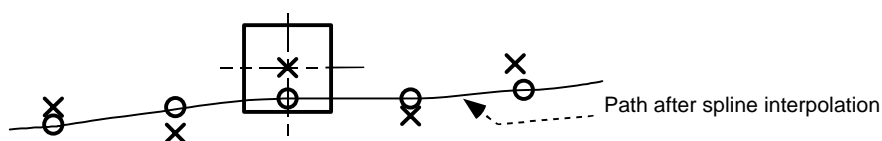
The noncontinuous points caused by CAD/M operation errors, etc., are compressed.



For height sensor data

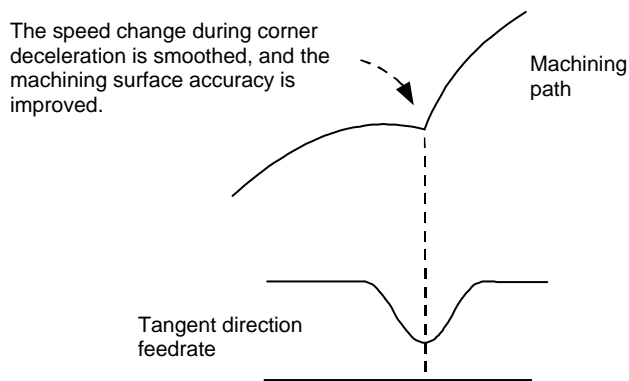
The noncontinuous points caused by the height sensor system's measurement errors, etc., are revised to make a smooth path.

- X Commanded point
- O Revised point



(Note) The block will not stop at the image data compression command point. The block will stop at the revised point.

(b) Acceleration/deceleration before interpolation and optimum corner deceleration

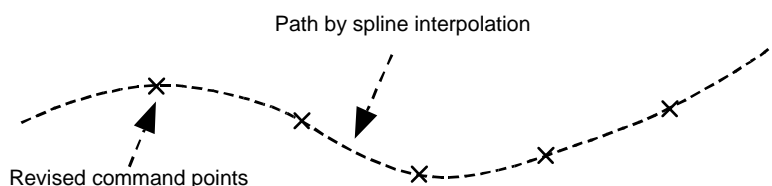


By accelerating/decelerating before interpolation, the machining shape error can be eliminated with smoothing, and a highly accurate path can be achieved. The speed change can be smoothed by the S-character filter, the machine vibration can be suppressed, and the surface accuracy improved.

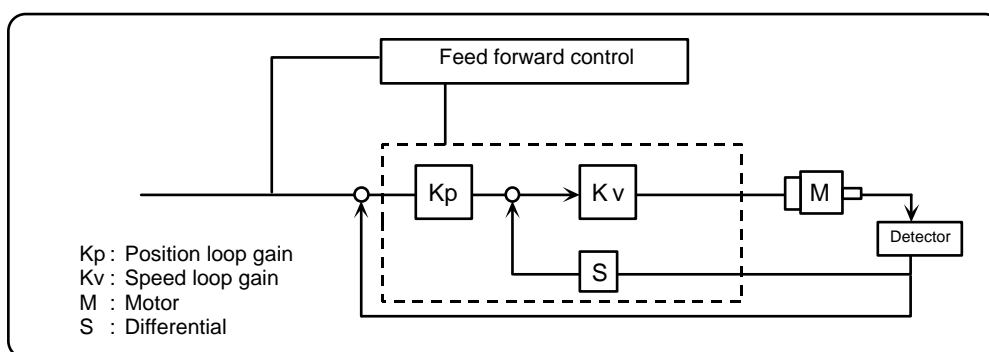
At the corner, the vector commanded in the machining program is automatically determined, and the speed is decelerated at the corner. A highly accurate edge can be machined by decelerating at the corner.

(c) High-accuracy spline interpolation

The command point train smoothed by fairing is spline interpolated. With the spline interpolation, the next level's feed forward control can be applied more stably and effectively.



(d) Feed forward control function



A stable servo control with an extremely small servo error can be realized using the feed forward control characteristic to this CNC system.

13.24 High-accuracy Control for Part Machining; G61.5

(1) Outline

This function carries out the machining commanded with linear interpolation or arc interpolation at a high-accuracy. This function is effective for high accuracy machining of shapes configured of linear lines and arcs instead of free curves commanded with fine segments.

Function	Cutting feed	Rapid traverse positioning
ACCS control	The block seams is controlled to be smooth and to improve the machining surface quality.	
Pre-interpolation acceleration/deceleration	Acceleration/deceleration time is reduced using constant inclination acceleration/deceleration and an S-character filter, and surface accuracy is improved by suppressing machine vibration.	Positioning time for micro-distances is reduced.
Optimum corner deceleration	Neat machining of sharp corners without waste	
Feed forward control	Machining path accuracy is improved by reducing the tracking lag error.	Positioning time is reduced by reducing the tracking lag error.

(2) Command format

G61.5 ;

The high-accuracy control for the part machining is commanded with G61.5.

(3) Detailed description

The functions in the G61.5 modal are as follow.

(a) ACCS control

The block seams are controlled to be smooth, allowing a favorable surface to be obtained even during high-speed machining. The operation is controlled so that the shape error caused by ACCS control is within the tolerance amount (parameter setting range).

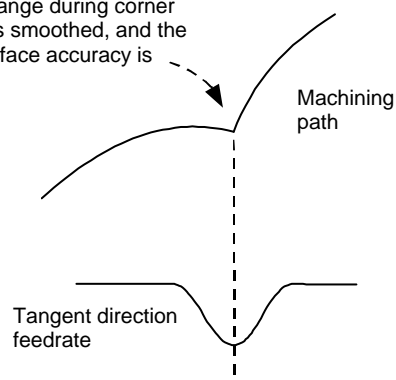
- 1) Optimum deceleration is carried out at the corner.
- 2) At block stop, the axis may not be over the commanded end point. However, the difference will be within the tolerance amount range.

(b) Pre-interpolation acceleration/deceleration and optimum corner deceleration

By accelerating/decelerating before interpolation, the machining shape error can be eliminated with smoothing, and a highly accurate path can be achieved. The speed change can be smoothed by the S-character filter, the machine vibration can be suppressed, and the surface accuracy improved.

At the corner, the vector commanded in the machining program is automatically determined, and the speed is decelerated at the corner. A highly accurate edge can be machined by decelerating at the corner.

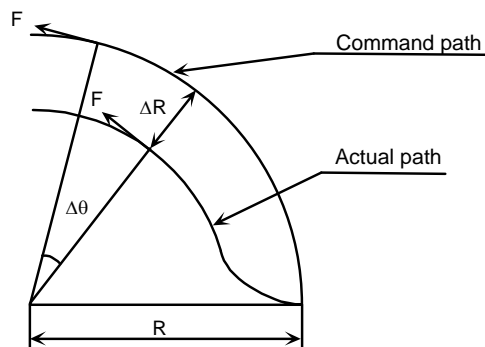
The speed change during corner deceleration is smoothed, and the machining surface accuracy is improved.



13. PROGRAMMING SUPPORT FUNCTIONS

13.24 High-accuracy Control for Part Machining; G61.5

When using arc interpolation, the arc radius decrease error is improved as follows.



High-accuracy control mode OFF (acceleration/deceleration after interpolation)	Exponential acceleration/deceleration	$\Delta R = \frac{1}{R} \cdot \left(-\frac{1}{2} T_s^2 + \frac{1}{2} T_p^2 \right) \cdot \left(\frac{F \times 10^3}{60} \right)^2 \text{ (mm)}$
	Linear acceleration/deceleration	$\Delta R = \frac{1}{R} \cdot \left(-\frac{1}{24} T_s^2 + \frac{1}{2} T_p^2 \right) \cdot \left(\frac{F \times 10^3}{60} \right)^2 \text{ (mm)}$
High-accuracy control mode ON (acceleration/deceleration before interpolation)		$\Delta R \doteq \frac{1}{R} \cdot \left(0 + \frac{1}{2} T_p^2 (1 - K_f^2) \right) \cdot \left(\frac{F \times 10^3}{60} \right)^2 \text{ (mm)}$

R : Command radius (mm) Ts : Time constant of set acceleration/deceleration circuit (s)
 ΔR : Radius error (mm) Tp : Position loop time constant (s)
 Δθ : Angle error (rad) Kf : Feed forward gain (fwd_g)
 F : Command cutting speed (m/min) F : Real cutting speed (m/min)

(c) Feed forward control

This control is the same as for high-accuracy control of die machining. Stable servo control with extremely small servo error can be realized.

13.25 Program Parameter Input; G10L70/G11.1

(1) Outline

The parameters designated from the setting and display unit can be changed from the machining program. The parameter input command is carried out by the parameter setting command after the data setting command.

(2) Command format

<u>G10 L70</u> ; Parameter input command
<u>Pp Ss Nn Aa Dd</u> ; Parameter data designation
. . . ; (Parameter data designation)

Address	Meaning	Command details
P	Data group No. (Section No.)	The parameter data's section No. is commanded with a two-digit value (positive integer) following P. This is equivalent to the first two digits of the parameter number.
S	System No.	Set "0" when parameters are common for all systems. Set the system No. (1 or 2) when parameters must be set for each system.
N	Data number (Sub-section No.)	The parameter data's data number (characteristic value for each data) is commanded with a four-digit value (positive integer) following N. This is equivalent to the first four digits of the parameter number.
A	Data sub-No. (Axis, spindle number)	If the input data is independent for the axis, the independent data for each control axis (1: 1st axis, 2: 2nd axis, ...) can be set with the axis number following A. (The number of control axes is characteristic to each model.) An alarm will occur if this address is used for axis common data.
D	Data	The parameter value is commanded with the value following D.

<u>G11.1</u> ; Parameter input cancel command
--

(3) Detailed description

- (a) The program parameter input function option is required to use this function.
- (b) If the same address is commanded two or more times in one block, the latter address will be valid.

(4) Notes

- (a) A program error will occur if this function is commanded when the option is not provided.
- (b) A program error will occur if the parameter No. is illegal. A program error will also occur if a numerical value is set that exceeds the setting range of that parameter.
- (c) A program error will occur if an address other than P, A, N, D or S is commanded.
- (d) A program error will occur if the program parameter input command is issued during a fixed cycle modal or radius compensation modal.
- (e) Only parameters that are set with decimal numerical values can be set with program parameter input.
(The value following D is always a decimal, and is always handled as having a decimal point. Hexadecimal data and character strings cannot be commanded.)
- (f) Issue the backlash compensation amount and machine error compensation amount program parameter input command at the zero point.

13.26 Program Compensation Input; G10

(1) Outline

The tool offset and workpiece offset can be set and changed from tape.

(2) Command format

(a) Workpiece coordinate system compensation input

The workpiece coordinate system value selected with G54 to G59 can be set or changed with program commands.

The workpiece coordinate setting command format is as follows.

<u>G10</u>	<u>L2</u>	<u>Pp1</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Zz1</u>	;
Parameter change command	Workpiece coordinate number		Setting value			

(Note) L2 can be omitted. A program error will occur if Pp1 is omitted.

G code	Function
G10 L2 P0	External workpiece coordinate system setting
G10 L2 P1	Workpiece coordinate system 1 setting (G54)
G10 L2 P2	Workpiece coordinate system 2 setting (G55)
G10 L2 P3	Workpiece coordinate system 3 setting (G56)
G10 L2 P4	Workpiece coordinate system 4 setting (G57)
G10 L2 P5	Workpiece coordinate system 5 setting (G58)
G10 L2 P6	Workpiece coordinate system 6 setting (G59)

(b) Tool compensation input

The tool compensation amount set from the setting and display unit can be input with program commands.

The tool compensation input format is as follows.

<u>G10</u>	<u>Ll1</u>	<u>Pp1</u>	<u>Rr1</u>	;
Compensation amount setting command	Compensation number	Compensation amount		

(Note) If Ll1 is omitted, the tool shape compensation amount will be set. A program error will occur if Pp1 is omitted.

G code	Function
G10 L10	Tool length shape compensation amount setting
G10 L11	Tool length wear compensation amount setting
G10 L12	Tool radius shape compensation amount setting
G10 L13	Tool radius wear compensation amount setting

(c) Expanded workpiece coordinate system compensation input

The values in each workpiece coordinate system selected with G54.1 P1 to P96 can be set or changed with program commands.

The format for expanded workpiece coordinate system compensation input is as follows.

G10	L20	Pn	Xx	Yy	Zz ;
		n = 1 to 96			

13. PROGRAMMING SUPPORT FUNCTIONS

13.26 Program Compensation Input; G10

(3) Detailed description

- (a) The program tool compensation input function option is necessary to use this function.
- (b) G10 is an unmodal command, and is valid only in the commanded block.
- (c) The compensation amount which is commanded is overwritten or added according to the modal state of the absolute value (G90) / the incremental value (G91).
- (d) G10 does not incur movement, but it must not be used with G commands other than G54 to G59, G90 and G91.
- (e) Particularly, if G10 is programmed in the block within which G22 or G23 is programmed, workpiece coordinate system compensation input is executed, but tool offset input is not executed because P becomes the number of subprogram repetitions.
- (f) If G10 compensation number P is programmed in the G23 block, it is assumed to be the number of subprogram repetitions and the subprogram will be repeated the number of times that is specified in P.
- (g) Decimal point inputs are valid for the compensation amount.
- (h) The external workpiece coordinate system and workpiece coordinate system compensation amount are set as the distance from the basic machine coordinate system zero point.
- (i) The workpiece coordinate system and workpiece coordinate system offset amount are set as the distance from the basic machine coordinate system zero point.
- (j) L2 can be omitted when inputting the workpiece offset.

(4) Notes

- (a) A program error will occur if this function is commanded when the option is not provided.
- (b) A program error will occur if an illegal L number or offset number is commanded.
- (c) A program error will occur if the offset amount exceeds the maximum command value.

13.27 N Number Single Command Skip

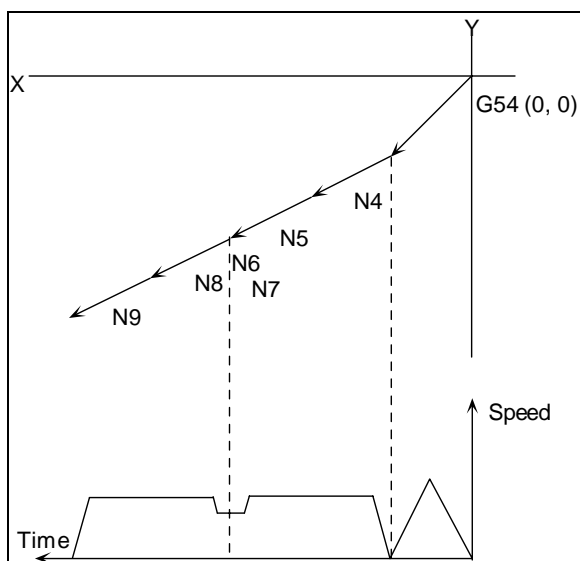
(1) Outline

If there is a single sequence number (N number) during cutting feed (G01), or if the sequence No. is commanded in the same block as the macro control statement, the tool may decelerate at the block joint, and may cause cutter marks on the machined surface. By validating this function, the single N number can be skipped to prevent deceleration at the block joint. This realizes a smooth machining surface.

(2) Example of program

(a) For single N number

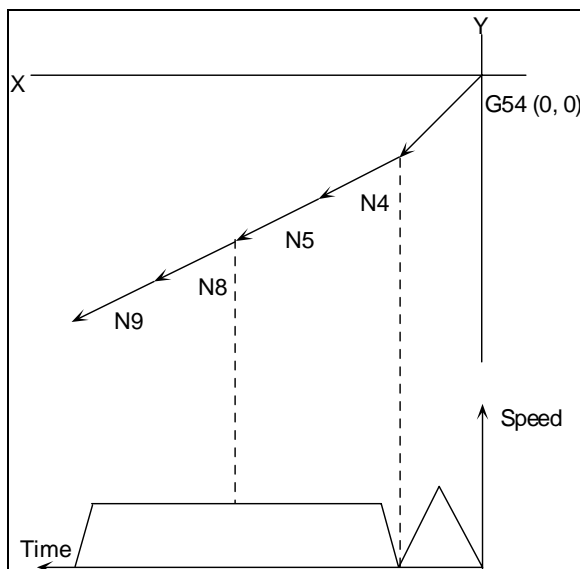
```
N1G90G54X0Y0 ;
N2G00X-10.Y-10. ;
N3G91G43H1Z-50. ;
N4 X-10.Y-5. ;
N5 X-10.Y-5.;
N6 ;
N7;
N8X-10.Y-5. ;
N9X-10.Y-5 ;
N10G01Z-20.F500 ;
.
.
```



The tool will decelerate at the N6 and N7 blocks.

The N6 and N7 blocks will be skipped when parameter (100132 SEQ NUM single skip) is turned ON.

```
N1G90G54X0Y0 ;
N2G00X-10.Y-10. ;
N3G91G43H1Z-50. ;
N4 X-10.Y-5. ;
N5 X-10.Y-5. ;
N6 ; ← Skip
N7 ; ← Skip
N8X-10.Y-5. ;
N9X-10.Y-5 ;
N10G01Z-20.F500 ;
.
.
```



13. PROGRAMMING SUPPORT FUNCTIONS

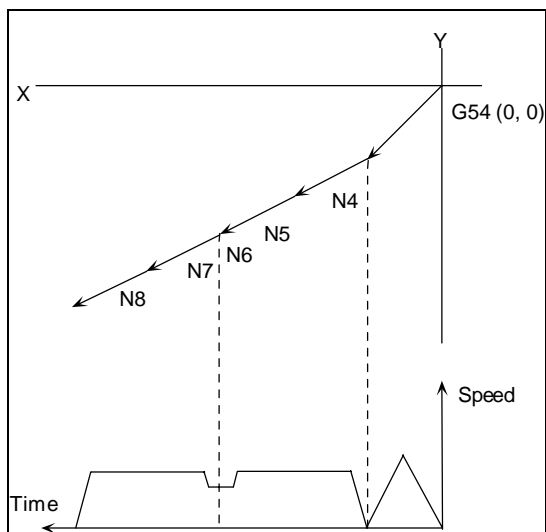
13.27 N Number Single Command Skip

(b) For N number + macro control statement

```

N1G90G54X0Y0 ;
N2G00X-10.Y-10. ;
N3G91G43H1Z-50. ;
N4 X-10.Y-5. ;
N5 X-10.Y-5.;
N6
IF[#5023EQ-50.]GOTO9;
N7 X-10.Y-5. ;
N8 X-10.Y-5 ;
N9 G01Z-20.F500 ;
.
.

```

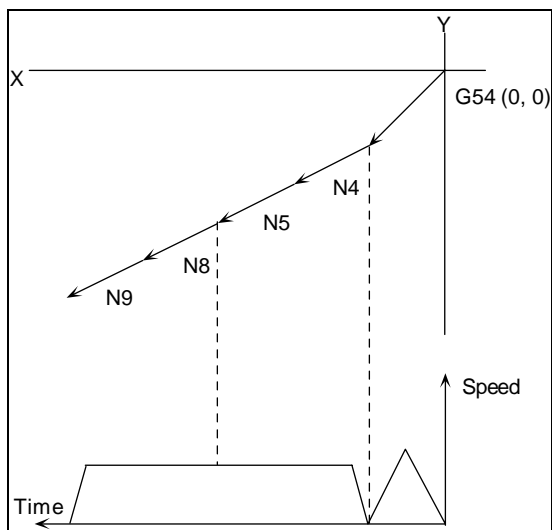


The axis will decelerate the at the N6 block.
 By turning the parameter ON, the program can be executed without decelerating at the N6 block.

```

N1G90G54X0Y0 ;
N2G00X-10.Y-10. ;
N3G91G43H1Z-50. ;
N4 X-10.Y-5. ;
N5 X-10.Y-5. ;
N6
IF[#5023EQ-50.]GOTO9;
N7 X-10.Y-5. ;
N8 X-10.Y-5 ;
N9 G01Z-20.F500 ;
.
.

```



(3) Notes

- (a) The sequence number on the screen is not updated for a block with a single N number. Instead, it will be updated at the block containing the next axis address or G code.
- (b) Even if the N single block is commanded for compare stop, the operation will not stop, and instead the program will end.
- (c) During graphic check, the N single block will not be skipped regardless of the parameter settings.
- (d) During the high-speed machining modes 1 and 2, the N single block will not be skipped regardless of the parameter settings.
- (e) The sequence number for an N single block can be searched.
- (f) The jump destination can be designated with a macro control statement.
- (g) When macro signal is ON, single block stop will be executed with the Nn + macro control statement and Nn + variable command.

13.28 End Point Error Check Clear; G69

(1) Outline

If an invalid program is given to NC, a program error occurs.

However, the error check clear command G69 can be used to suppress a program error only in the following conditions.

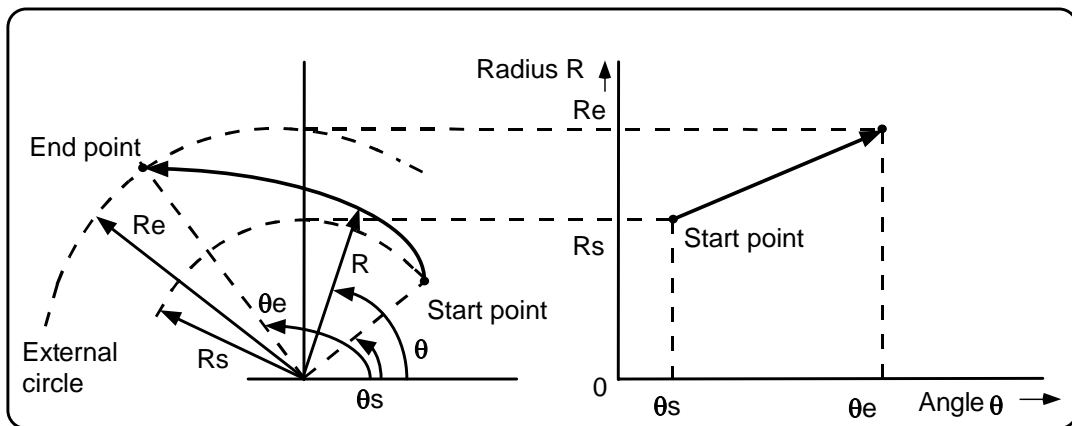
- (a) The end point does not exist on the circumference of a circle in circular interpolation (containing helical cutting).
- (b) Compensation amount is changed during circular arc modal in cutter compensation.

(2) Detailed description

The G69 command is unmodal.

(a) Circular interpolation (containing helical cutting)

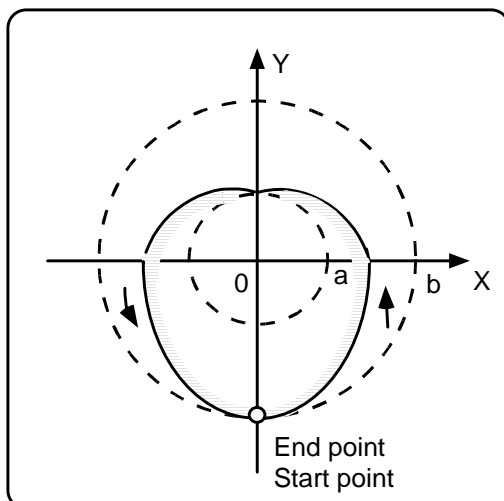
If the end point does not exist on the circumference of a circle and G69 command is given in circular interpolation, the arc radius together with the move angle changes linearly as shown below:



Although the circumference speed of the external circle is slightly faster than the command speed, the angular speed is constant.

Example 1: Heart cum cutting

Displacement is the radius difference at the start and end points ((b - a) in the illustration below). The example program is separated into two blocks of the right and left sides.

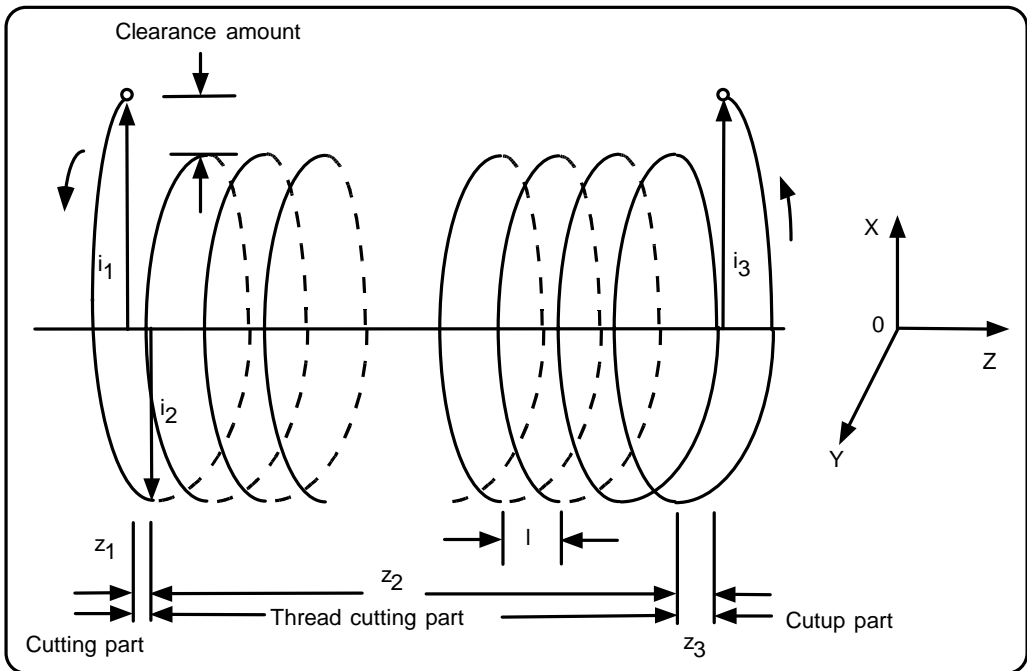


```
G69 G03 Y a+b J b F f1;
(right side)
G69 G03 Y -a-b J -a;
(left side)
```

13. PROGRAMMING SUPPORT FUNCTIONS
13.28 End Point Error Check Clear; G69

Example 2: Large-caliber thread cutting

For large-caliber thread cutting, separate the program into the cutting, thread cutting, and cutup parts (helical cutting command in each part). Use G69 to specify the radius clearance amount of the cutting and cutup parts. (The arc end point is displaced from the circumference by clearance.)

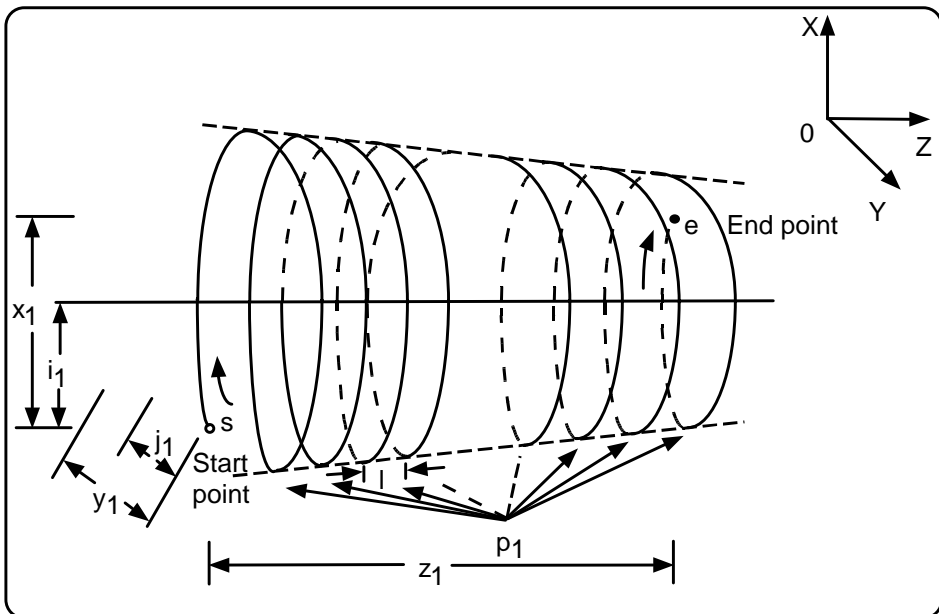


```
G69 G03 X-i1-i2 Y0 Zz1 I-i1 J0 Ff1; (Cutting part, half circle)
G03 X0 Y0 Zz2 Ii2 J0 Pp2; (Thread cutting part, true circle)
G69 G03 Xi2+i3 Y0 Zz3 Ii2 J0; (Cutup part, half circle)
```

Note: The number of thread cutting part pitches, p_2 is found by dividing stroke z_2 by pitch l . Program so that the remainder is set to 0.

Example 3: Taper, thread cutting

Tapered helical cutting starting at any desired angle can also be performed as shown below:



13. PROGRAMMING SUPPORT FUNCTIONS
13.28 End Point Error Check Clear; G69

Program incremental values from the start point s to end point e, x_1 , y_1 , and z_1 for the X, Y and Z values; incremental values from the start point s to the center i_1 and j_1 for the I and J values; and the number of pitches p_1 for the P value.

G69 G03 $Xx_1 Yy_1 Zz_1 Ii_1 Jj_1 Pp_1 Ff_1$;

Taper t and pitch l can be found by the expression:

$$t = \frac{2(re - rs)}{x_1}$$

where $rs = \sqrt{i_1^2 + j_1^2}$ $re = \sqrt{(x_1 - i_1)^2 + (y_1 - j_1)^2}$

$$l = \frac{Z_1}{(2\pi \cdot p_1 + \theta)/2\pi}$$

where $\theta = \theta_e - \theta_s = \tan^{-1} \frac{j_1 - y_1}{i_2 - x_1} - \tan^{-1} \frac{-j_2}{-i_1}$

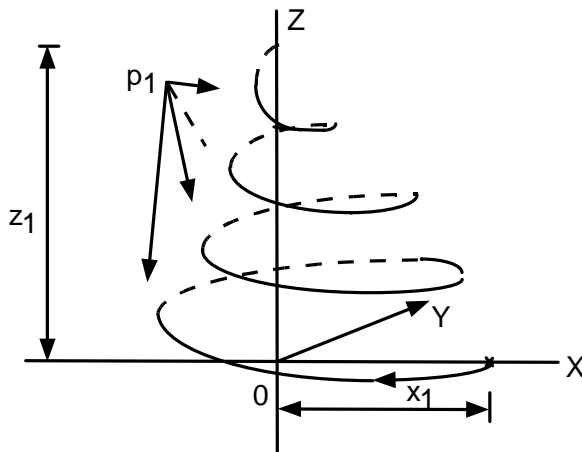
Where rs and re are the radiuses at the start and end points respectively;
 θ_s and θ_e are the angles at the start and end points respectively.

Example 4: Cone cutting

Cone cutting is taper thread cutting application.

The start or end point exists on the center line.

Taper can be prepared by incrementing or decrementing the arc diameter. The pitch is determined by z_1/p_1 .



G69 G02 $X-x_1 Y0 Zz_1 I-x_1 Pp_1 Ff_1$;
 x_1 : Bottom surface radius
 z_1 : Height
 p_1 : Number of pitches
 f_1 : Feedrate

(b) Tool compensation

When the compensation amount changes during circular arc modal, specify the G69 command.

Example : Tool compensation during circular arc mode

G69 G41 G02 $Xx_1 Yy_1 Ii_1 Jj_1 Ff_1 Hh_1$; Tool compensation amount change

G69 G40 G02 $Xx_2 Yy_2 Ii_2 Jj_2$; Tool compensation cancel

13.29 Coordinate Read Function;G14

(1) Outline

The G14 command can be used to read the end point coordinates of the immediately preceding block, the machine coordinates, the work coordinates, the TLM coordinates, or the skip coordinates.

(2) Command format

G14 Xx Yy Zz Aa Pp;

- x: Variable number where X axis coordinate is stored.
- y: Variable number where Y axis coordinate is stored.
- z: Variable number where Z axis coordinate is stored.
- a: Variable number where the additional axis coordinate is stored.
- p: Type of coordinates to be read

(3) Detailed description

The type of coordinates to be read is specified in p. The coordinates that can be read are listed below:

Types of coordinates that can be read

P command value	No p command is given or a value other than 1 - 4 is specified	1	2	3	4
Meaning	End point position of the immediately preceding block	Machine coordinates	Work coordinates	TLM coordinates	Position where skip signal is turned on in G31 block
Coordinate system	Work coordinates	Basic machine coordinates	Work coordinates	Coordinate system whose origin is the TLM switch on point when the machine stops	Work coordinates
Compensation amount	Not contained	Contained	Contained	Contained	Contained
Read during moving	Enabled	Disabled	Disabled	Disabled	Enabled

- (a) The skip coordinates become the end point position when the skip signal is not turned on in a G31 block.
- (b) The G14 command which is unmodal is effective only in the block within which it is programmed.
- (c) The end point and skip coordinates can also be read during moving. Check move stop before reading the machine, work, and TLM coordinates.

13. PROGRAMMING SUPPORT FUNCTIONS
13.29 Coordinate Read Function;G14

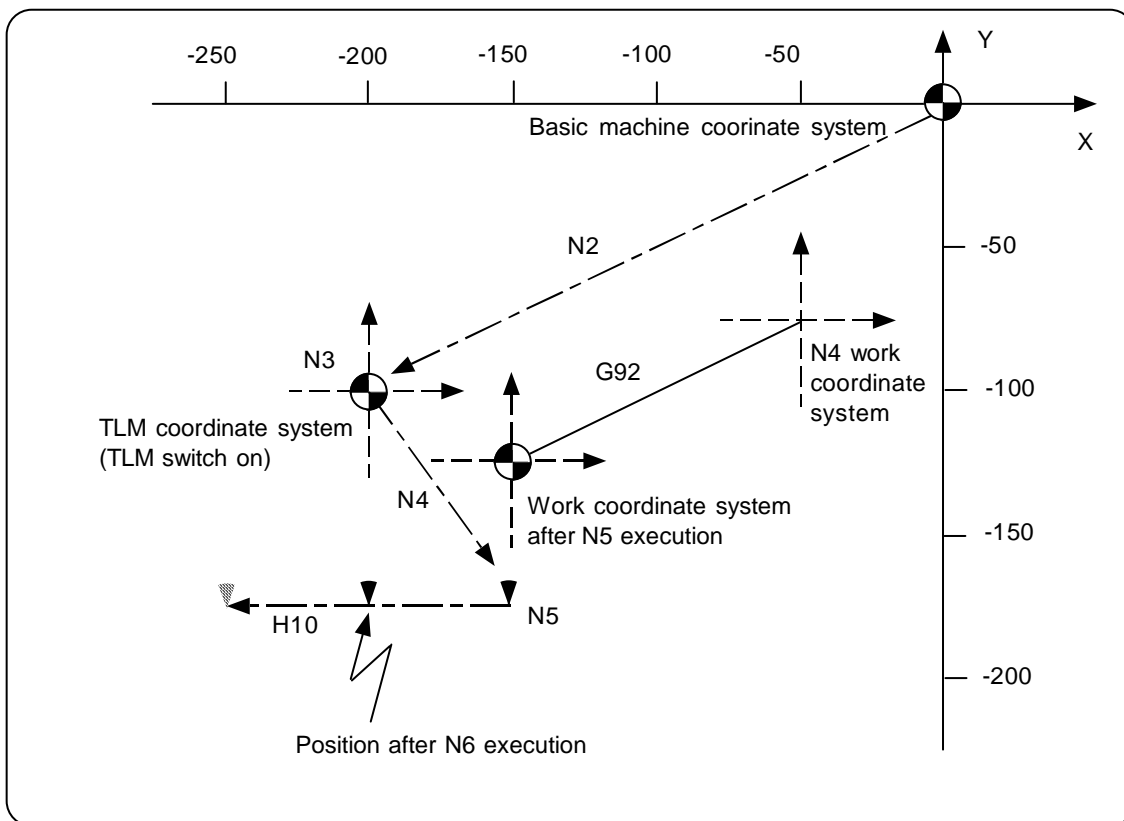
(4) Program examples

Example 1:

```

N1 G28 X0 Y0 Z0 ; An example of p1 command value and read coordinates is given.
N2 G90 G00 X-200. Y-100. G53;
N3 M60; (TLM switch is turned from off to on.)
N4 G00 G54 X-100. Y-100. ;
N5 G92 X0 Y0 Y-50. ;
N6 G00 G43 X-100. H10;
N7 G14 X100 Y101 Pp1;
( G54 : X-50. H10 : 50. )
  Y-75.

```



Read of basic machine, work, and TLM coordinate systems

13. PROGRAMMING SUPPORT FUNCTIONS
13.29 Coordinate Read Function;G14

- (a) End point position of the immediately preceding block when no P command is given or p1 is not 1 to 4
 #100 = -100.
 #101 = -50.
 Compensation amount is not contained.

- (b) Machine coordinates
 Specify p1 = 1.
 #100 = -200.
 #101 = -175.
 Compensation amount is contained.

- (c) Work coordinates
 Specify p1 = 2.
 #100 = -50.
 #101 = -50.
 Compensation amount is contained.

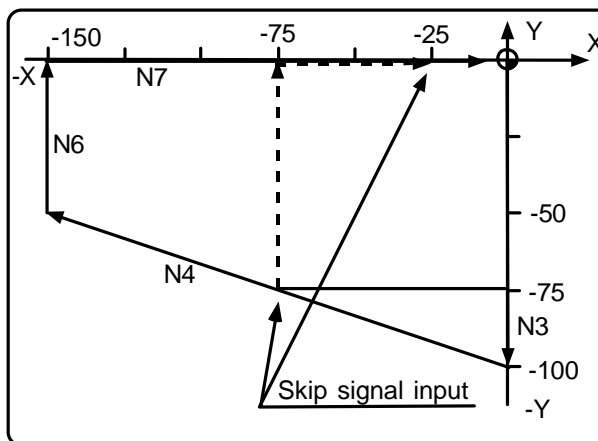
- (d) TLM coordinates
 Specify p1 = 3.
 #100 = 0
 #101 = -75.
 Compensation amount is contained.

- (e) Skip coordinates
 Specify p1 = 4.

Example 2: Skip coordinate read

```

N1 G91 G28 X0 Y0 Z0;
N2 G90 G00 X0 Y0;
N3 X0 Y-100.;
N4 G31 X-150. Y-50. F80;
N5 G14 X100 Y101 P4;
N6 G00 Y0;
N7 G31 X0;
N8 G14 X110 Y111 P4;
N9 M02;
  
```



$$\begin{aligned} \#100 &= -75. + \varepsilon & \#101 &= -75. + \varepsilon \\ \#110 &= -25. + \varepsilon & \#111 &= -75. + \varepsilon \end{aligned}$$

(Note 1) ε is an error caused by response delay. (For details, see “14.1 High-speed Skip; G31”.)
(Note 2) #111 becomes the N4 skip signal input coordinates because Y command is not included in N7.

(5) Restrictions

- (a) When an invalid variable number is specified, program error P241 occurs.

14. AUTOMATED SUPPORT FUNCTIONS

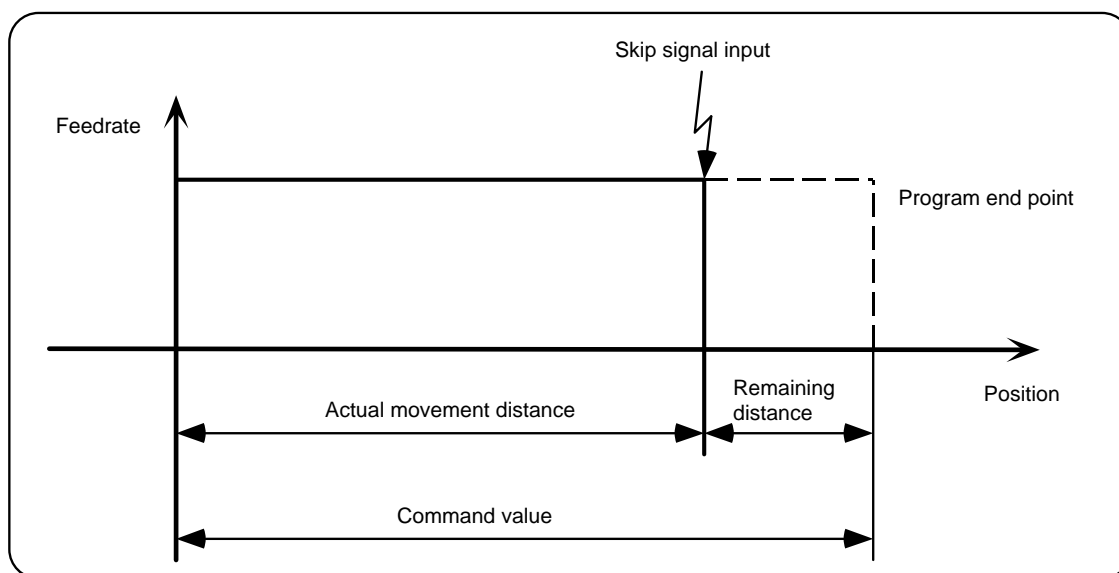
14.1 High-speed Skip; G31

(1) Outline

If a skip signal is input from an external source during linear interpolation by the G31 command, the machine feed will stop immediately, the remaining distance will be discarded, and the next block command will be executed. The feed by the G31 is normally time constant 0ms.

(2) Command format

<u>G31</u>	<u>Xx1</u>	<u>Yy1</u>	<u>Zz1</u>	<u>Ff1</u> ;
Skip command	Command value			Feedrate



(3) Detailed description

(a) Function

- 1) The following two types of skip feedrates can be used.
 - a. If there is an F command in the program, feedrate following program commands.
 - b. When there is no F command in the program, feedrate following parameter setting.
 In either case, the F modal is not updated.
- 2) Automatic acceleration and deceleration do not apply to the G31 block. The maximum speed of G31 is determined by the machine specifications.
- 3) When G31 is commanded, the override is invalid and fixed at 100% if the machine lock is OFF.
- 4) The dry run is also invalid. Note that the stop conditions (feed hold, interlock, override zero, stroke end) are valid. The external speed is also valid.
- 5) The G31 command is unmodal and so it needs to be assigned each time.
- 6) If any of the skip singles are input during G31 command start, the G31 command will be completed immediately. If the skip signal is not input before the G31 block ends, the G31 command will also end at the completion of the movement command.
- 7) When the G31 command is assigned during radius compensation, program error will occur.
- 8) When there is no F command in the G31 command and the parameter speed is also zero, program error will occur.
- 9) When the automatic machine lock is applied to all axes, the skip signal will be ignored and execution will continue as far as the end of the block.

14. AUTOMATED SUPPORT FUNCTIONS

14.1 High-speed Skip; G31

(b) Skip coordinate readout

The coordinate positions at which the skip signals are entered are stored in system variables #5061 (1st axis) through #5064 (4th axis). They can therefore be used with the user macros.

```

:
G90 G00 X-100.;
G31 X-200. F60;      Skip command
#101 = #5061         The skip signal input coordinates (workpiece coordinate system)
:                   are read out in variables #101.
    
```

(c) G31 coasting

The amount of coasting from the time when the skip signal is input during the G31 command until the time when the machine stops differs according to the parameter "112210 skip (G31 skip speed)" and F command in G31.

It takes only a short time to start responding to the skip signal until the machine decelerates and stops. As a result, this enables the machine to stop with a high degree of precision and with minimal coasting.

The coasting distance to the feed stop is approximately as follows because of the delay in detecting the skip signal input.

$$\delta \doteq \frac{F}{60} \times (T_p + t)$$

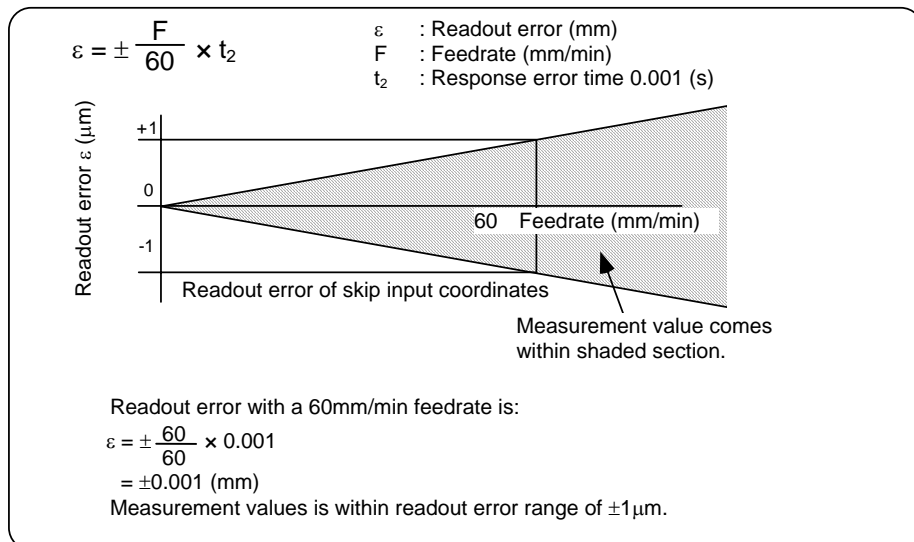
δ : Coasting amount (mm)
 F : G31 speed (mm/min)
 T_p : Position loop time constant (s) = (position loop gain)⁻¹
 t : Response delay time = 0.0035 (s)

(d) Skip coordinate readout error

1) Skip signal input coordinate readout

The coasting amount based on the position loop time constant T_p and cutting feed time constant T_s is not included in the skip signal input coordinate values.

Therefore, these workpiece coordinate values applying when the skip signal is input can be read out across the error range in the following formula as the skip signal input coordinate values. However, coasting amount based on response delay time t_1 results in a measurement error and so compensation must be provided.



14. AUTOMATED SUPPORT FUNCTIONS

14.1 High-speed Skip; G31

2) Readout of other coordinates

The readout coordinate values include the coasting amount. When coordinate values are required with skip signal input, refer to "(C) G31 coasting" included in (3) of this section and compensate. As in the case of 1), the coasting amount based on the delay error time t_2 cannot be calculated, and this generates a measuring error.

3) Examples of compensating for coasting amount

a. Compensating for skip signal input coordinates

#110 = Skip feedrate;
#111 = Response delay
time t_1 ;

```

}
G31 X100. F100 ;—————Skip command
G04 ;—————Machine stop check
#101 = #5061 ;—————Skip signal input
coordinate readout
#102 = #110 * #111/60 ;—————Coasting amount based
on response delay time
#105 = #101 - #102 ;—————Skip signal input
coordinates
}
```

b. Compensating for workpiece coordinates

#110 = Skip feedrate;
#111 = Response delay
time t_1 ;
#112 = Position loop time
constant T_p ;

```

}
G31 X100. F100 ;—————Skip command
G04 ;—————Machine stop check
#101 = #5061 ;—————Skip signal input coordinate
readout
#102 = #110 * #111/60 ;—————Coasting amount based on
response delay time
#103 = #110 * #112/60 ;—————Coasting amount based on
position loop time constant
#105 = #101 - #102 - #103;—————Skip signal input
coordinates
}
```

14. AUTOMATED SUPPORT FUNCTIONS
14.2 Multi-step Skip; G31.1 to G31.4

14.2 Multi-step Skip; G31.1 to G31.4

(1) Outline

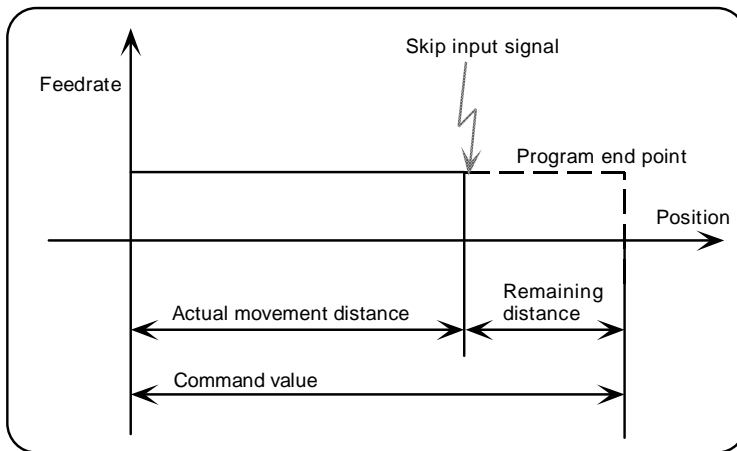
By designating the combination of skip signals input from the external source, the program can be skipped at those conditions. The skip signals that can be used are 1 to 8.

When measuring, multi-step skip can be used to shorten the measurement time by changing the speed in stages.

Command the skip speed with the command speed F or a parameter. Note that the F modal is not updated.

There are two multi-skip command formats differing by the method that the skip signal combination is designating.

When the skip signal is input with the preset conditions during dwell, the remaining dwell time will be canceled.



(2) Command format

(a) Method 1

Designate the skip signal combination with parameters for each skip command (G31.1, G31.2, G31.3, G31.4). When one of the signals in the combination is input, the skip operation takes place.

<u>G31.n</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u>	<u>Ff1</u> ;
Skip command (n=1,2,3,4)	Command value and target coordinate value *2	Feedrate *1

*1: Follows G94/G95.
 *2: Follows G90/G91.

(b) Method 2

The skip signals that can be used are 1 to 8. Which of these to be used is designated with P in the program. When one of the combination signals is completely input, the skip operation takes place.

When there is no P command, the normal skip command will be applied.

<u>G31</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u>	<u>Pp</u>	<u>Ff1</u> ;
Skip command	Command value and target coordinate value *2	Skip signal command	Feedrate *1

*1: Follows G94/G95.
 *2: Follows G90/G91.

(c) Dwell skip

Skip can be applied in respect to dwell. The remaining dwell time is canceled by the skip conditions set with parameters, and the dwell command and the next block will be executed. Even during dwell (speed designation), the remaining rotation speed can be canceled by the skip conditions, and the next block will be executed in the same manner.

<u>G04</u>	<u>Xx</u> ;
Dwell command	Dwell time *1

*1: Follows G94/G95.

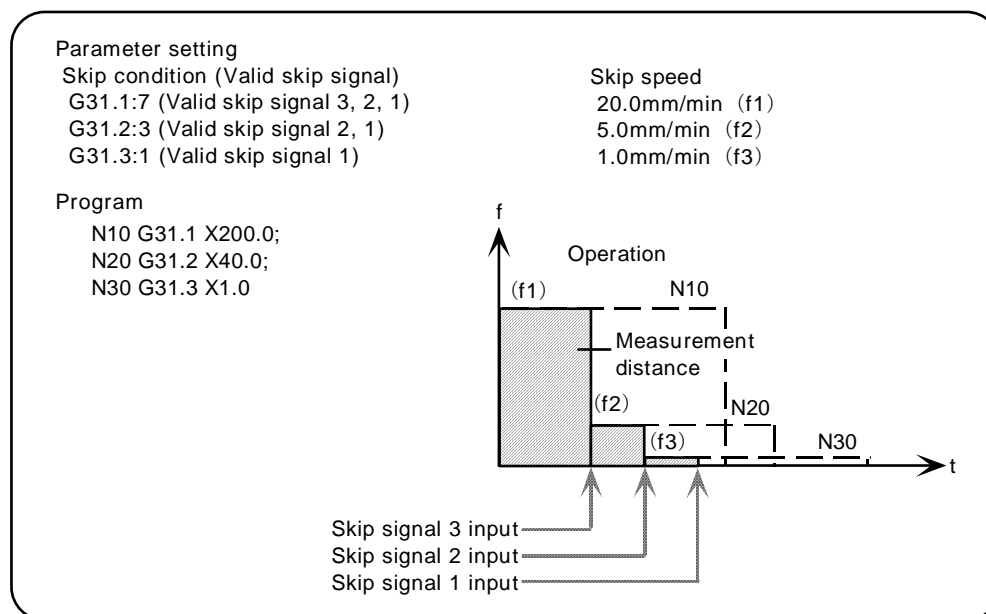
14. AUTOMATED SUPPORT FUNCTIONS

14.2 Multi-step Skip; G31.1 to G31.4

(3) Operation example

(a) Method 1

By using multi-step skip, the measurement can be carried out while reducing the speed in stages, instead of feeding at a set speed, to the measurement point. This allows the measurement time to be shortened.



(Note) If the skip signal 1 is input before the skip signal 2 during execution of N20 above, N20 will be skipped at that point, and M30 will end immediately.

(4) Detailed description

(a) Skip feedrate

The skip feedrate is commanded with the program command or with parameters.

- 1) When there is an F command in the program, the feedrate will follow the program command.
- 2) When there is no F command in the program, the feedrate will follow the parameter settings.

Refer to the Parameter Manual for details on the parameters.

Note that in either case, the F modal will not be updated.

If there is no F command in the G31.n command and the parameter speed is "0", the program error "PR0603 Skip speed 0" will occur.

(b) Skip conditions

The skip conditions are commanded with the program command or parameters.

1) Method 1 or dwell skip

The skip signal combinations are designated with the parameters, and when even one of the combination signals is input, the skip operation will take place.

Refer to the Parameter Manual for details on the parameters.

Refer to the following table for the relation of the skip conditions and valid signals.

2) Method 2

The skip signal combination is designated with the P following G31. When even one of the combination signals is input, the skip operation will take place.

Command P in the range of 1 to 255. If the command range is exceeded, the program error "PR0035 Address range over" will occur. When commanded with a decimal, the value after the decimal point will be ignored.

Refer to the following table for the relation of the P value and valid signals.

14. AUTOMATED SUPPORT FUNCTIONS
14.2 Multi-step Skip; G31.1 to G31.4

Skip signal command P	Valid skip signal							
	8	7	6	5	4	3	2	1
1								○
2							○	
3							○	○
4						○		
5						○		○
6						○	○	
7						○	○	○
8					○			
:	:	:	:	:	:	:	:	:
253	○	○	○	○	○	○		○
254	○	○	○	○	○	○	○	
255	○	○	○	○	○	○	○	○

14. AUTOMATED SUPPORT FUNCTIONS
14.3 Automatic Tool Length Measurement; G37.1

14.3 Automatic Tool Length Measurement; G37.1

(1) Outline

By using G37.1, the tool length can be automatically measured with program commands. When this command is issued, the tool end is moved to the tool measurement position. When the sensor is contacted, the tool immediately stops, and the remaining commands are canceled. At the same time, the difference of the coordinate value when measurement point is reached and the G37.1 start position is calculated, and registered as the tool length compensation amount.

(2) Command format

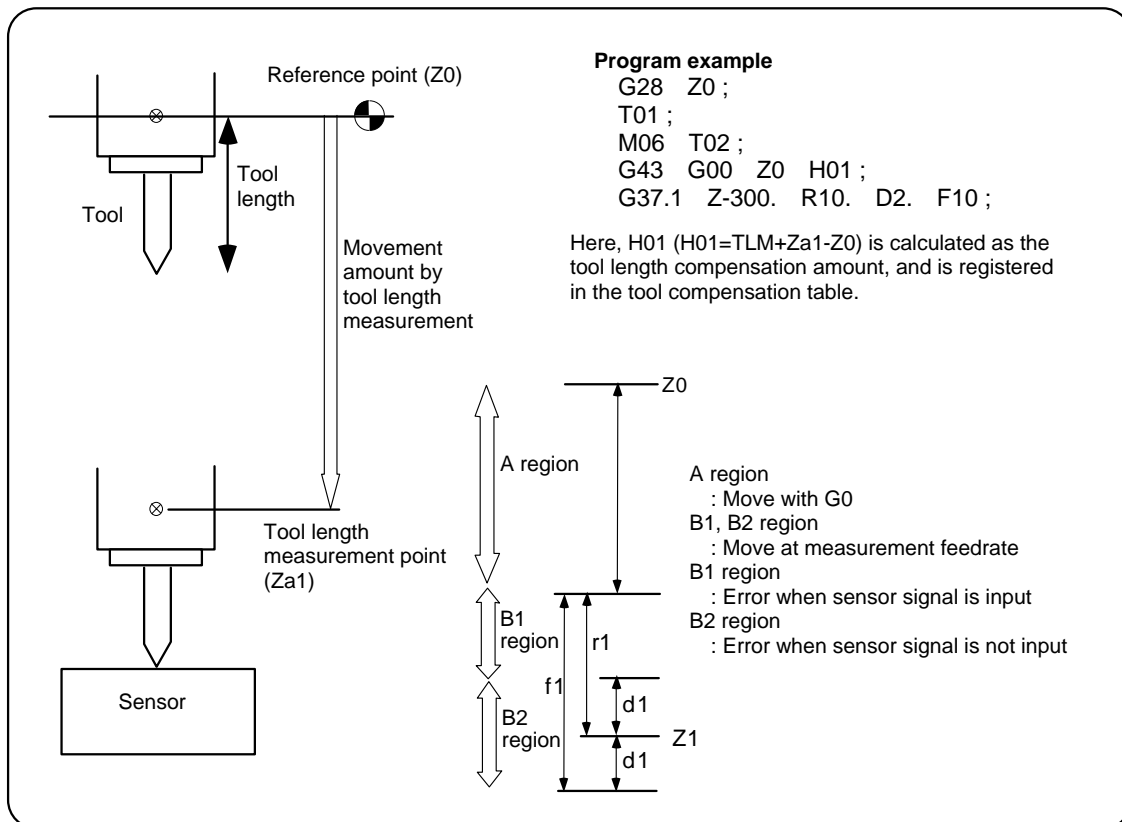
<u>G37.1</u>	<u>Zz1</u>	<u>Rr1</u>	<u>Dd1</u>	<u>Ff1</u> ;
Measurement command	Command value	Deceleration region	Measurement region	Feedrate

- Z : Measurement axis address and measurement position coordinate value
- R : Distance between point for starting movement at measurement speed and measurement position
- D : Range for tool to stop
- F : Measurement speed

If R_, D_, and F_ are omitted, the values set in the parameters will be used.

(3) Detailed description

(a) Example of execution and operation



14. AUTOMATED SUPPORT FUNCTIONS

14.3 Automatic Tool Length Measurement; G37.1

- 1) The sensor signal (measuring position arrival signal) is used in common with the skip signal.
- 2) The feedrate will be 1mm/min if the F command and parameter measurement speed are 0.
- 3) An updated offset amount is valid from the Z axis (measurement axis) command following the G37.1 command.
- 4) Excluding the values applying at the PLC side, the delay or fluctuations in the sensor signal processing range from 0 to 0.2ms at the NC system side only.
Therefore, a measuring error will arise as follows:
$$\text{Maximum measuring error [mm]} = \text{Measuring speed [mm/min]} \times 1/60 \times 0.2 \text{ [ms]}/1000$$
- 5) The machine position coordinates are read by sensor signal detection, and the machine will overtravel and stop at a position equivalent to the servo droop.
$$\text{Maximum overtravel [mm]} = \text{Measuring speed [mm/min]} \times 1/60 \times 1/\text{Position loop gain [1/s]}.$$
- 6) The standard position loop gain is 33 [1/s].
- 7) When the current tool length offset amount is zero, it will be interpreted as a new measurement, and the length dimension will be compensated.
- 8) When the tool length offset amount is not zero and there is a distinction between the length dimensions and length wear, the length wear will be compensated. If there is no distinction, the length dimensions will be compensated.

(4) Notes

- (a) A program error will occur when any axis is not commanded in the G37.1 block or when two or more axes have been commanded.
- (b) A program error will occur if the H code is not commanded in the G37.1 block.
- (c) Command G37.1 as a pair with the G43H_ command that designates the compensation number. A program error will occur if G43H_ is not commanded before the G37.1 block.
- (d) A program error will occur when the sensor signal was input outside the allowable measuring range or when the sensor signal was not detected even when the tool arrived at the end point.
- (e) When manual interrupt has been applied during movement at the measuring speed, return to the position prior to interrupt and then operation must be resumed.
- (f) The data commanded in G37.1 or the parameter setting data must meet the following conditions:
$$|\text{Measurement point} - \text{start point}| > R \text{ command or parameter } r > D \text{ command or parameter } d$$
- (g) When the D command and parameter d in (f) above are zero, operation will be completed normally only when the commanded measurement point and sensor signal detection point coincide. Otherwise, program error will result.
- (h) When the R and D commands as well as parameters r and d in (f) above are all zero, a program error will occur regardless of whether the sensor signal is present or not after positioning at the commanded measurement point.

15. PROTECTION FUNCTIONS

15.1 Pre-movement Stroke Check; G22.1/G23.1

(1) Outline

By commanding the boundaries from the program with coordinate values on the machine coordinate system, machine entry into that boundary can be prohibited. This can be set only for the three basic axes.

While the normal stored stroke limit stops entry before the prohibited area, this function causes a program error before movement to the block if a command exceeding the valid movement area is issued.

G code	Function
G22.1	Pre-movement stroke check ON
G23.1	Pre-movement stroke check OFF

(2) Command format

The G22.1 command format is as follows.

(Note) In the following command format, the basic axes are X, Y and Z. If the basic axis name differs, command the point 1 coordinate command address with the basic axis name.

The inner side of the boundary commanded with the point 1 coordinate and point 2 coordinate is the prohibited area.

If the command is omitted, "0" will be set for the address.

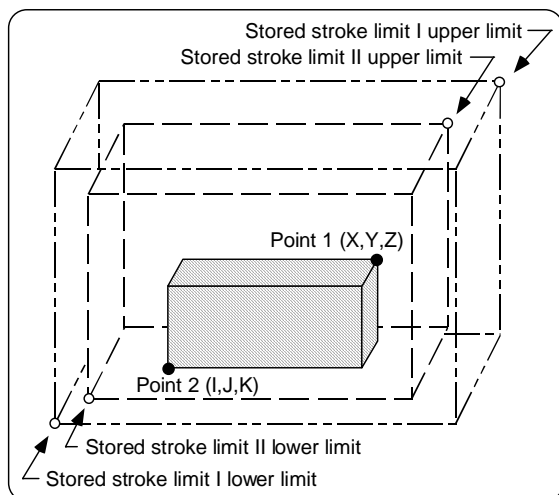
<u>G22.1</u>	<u>Xx1</u> <u>Yy1</u> <u>Zz1</u>	<u>li1</u> <u>Jj1</u> <u>Kk1</u> ;
Pre-movement stroke check ON	Point 1 coordinate	Point 2 coordinate

The G23.1 command format is as follows.

<u>G23.1</u> ;
Pre-movement stroke cancel

(3) Detailed description

The area designated with this function is different from the area designated with the stored stroke limit. However, the area enabled by both functions will be the actual valid movement range.



(Note) The upper limit value and lower limit value (points 1 and 2 in the above drawing) are commanded with coordinate values on the machine coordinate system.

15. PROTECTION FUNCTIONS

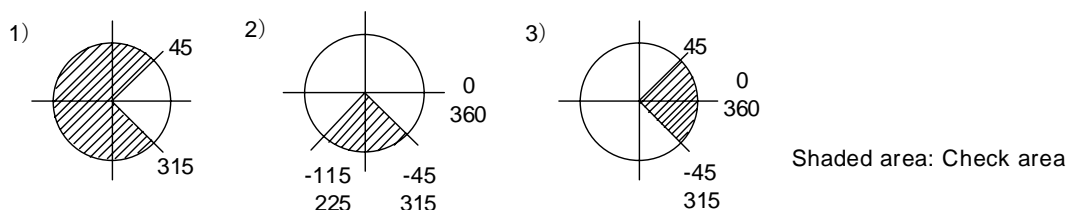
15.1 Pre-movement Stroke Check; G22.1/G23.1

(4) Restrictions

- (a) This function is valid only when starting the automatic operation. When interrupted with manual absolute OFF, the prohibited area will also be shifted by the interrupted amount.
- (b) An error will occur during stroke check if the start point or end point is in the prohibited area.
- (c) Stroke check will not be carried out for axes having the same value set for point 1 and point 2.
- (d) The stroke check is carried out with the tool center coordinate values.
- (e) If G23.1X_Y_Z_; etc., is commanded, the command will be interpreted as G23.1;X_Y_Z;. Thus, after the pre-movement stroke check is canceled, movement will take place with the previous movement modal.
- (f) During automatic reference point return, the check will not be carried out between the intermediate point and zero point. With G29, when moving between the start point and intermediate point, the check will not be carried out.
- (g) If there is an address not used in one block, the warning "PRW0004 Unused address exists" will occur.
- (h) When the basic axis is set as a rotary-type rotation axis, the prohibited area will change from 0° to 360° in the same manner as the movement command.
If the setting extends over "0°", the side containing "0°" will be the check area. (Refer to the example below.)

Example

- 1) G22 Z45. K315. : Stroke check area $45 \leq Z \leq 315$.
- 2) G22 Z-115. K-45. : Stroke check area $225 \leq Z \leq 315$.
- 3) G22 Z45. K-45. : Stroke check area $0 \leq Z \leq 45$, $315 \leq Z \leq 360$.



APPENDIX 1. LIST OF VARIABLE NUMBERS

APPENDIX 1. LIST OF VARIABLE NUMBERS

○ : Reading/writing possible
 × : Writing not possible
 — : Unrelated to reading/writing

Variable No. (#)	Details	Read	Write																																																																		
0	Blank	○	×																																																																		
1 to 33	Local variable Argument designation I <table border="1" style="width:100%; border-collapse: collapse;"> <thead> <tr> <th>#</th><th>1</th><th>2</th><th>3</th><th>4</th><th>5</th><th>6</th><th>7</th><th>8</th><th>9</th><th>10</th><th>11</th><th>12</th><th>13</th><th>14 to 26</th><th>27 to 33</th></tr> </thead> <tbody> <tr> <td>Argument address</td><td>A</td><td>B</td><td>C</td><td>I</td><td>J</td><td>K</td><td>D</td><td>E</td><td>F</td><td>G</td><td>H</td><td>L</td><td>M</td><td>N to Z</td><td></td></tr> </tbody> </table> Argument designation II <table border="1" style="width:100%; border-collapse: collapse;"> <thead> <tr> <th>#</th><th>1</th><th>2</th><th>3</th><th>4</th><th>5</th><th>6</th><th>7</th><th>8</th><th>9</th><th>10 to 27</th><th>28</th><th>29</th><th>30</th><th>31</th><th>32</th><th>33</th></tr> </thead> <tbody> <tr> <td>Argument address</td><td>A</td><td>B</td><td>C</td><td>I1</td><td>J1</td><td>K1</td><td>I2</td><td>J2</td><td>K2</td><td>I3 to K8</td><td>I9</td><td>J9</td><td>K9</td><td>I10</td><td>J10</td><td>K10</td></tr> </tbody> </table>	#	1	2	3	4	5	6	7	8	9	10	11	12	13	14 to 26	27 to 33	Argument address	A	B	C	I	J	K	D	E	F	G	H	L	M	N to Z		#	1	2	3	4	5	6	7	8	9	10 to 27	28	29	30	31	32	33	Argument address	A	B	C	I1	J1	K1	I2	J2	K2	I3 to K8	I9	J9	K9	I10	J10	K10	○	○
#	1	2	3	4	5	6	7	8	9	10	11	12	13	14 to 26	27 to 33																																																						
Argument address	A	B	C	I	J	K	D	E	F	G	H	L	M	N to Z																																																							
#	1	2	3	4	5	6	7	8	9	10 to 27	28	29	30	31	32	33																																																					
Argument address	A	B	C	I1	J1	K1	I2	J2	K2	I3 to K8	I9	J9	K9	I10	J10	K10																																																					
100 to 199 500 to 999	Common variable 100 sets: 100 to 149, 500 to 549 200 sets: 100 to 199, 500 to 599 300 sets: 100 to 199, 500 to 699 600 sets: 100 to 199, 500 to 999	○	○																																																																		
450 to 499	Machine maker macro variables (Can be used only in machine maker macro program.)	○	○																																																																		
1000 to 1031 1032 to 1041	Macro I/F input <table border="1" style="width:100%; border-collapse: collapse;"> <thead> <tr> <th>#</th><th>Details</th><th>#</th><th>Details</th></tr> </thead> <tbody> <tr> <td>1000</td><td>Register R 124 bit0 to 1015</td><td>1034</td><td>Register R 128, R 129</td></tr> <tr> <td>1015</td><td>Register R 124 bit15</td><td>1035</td><td>Register R 130, R 131</td></tr> <tr> <td>1016</td><td>Register R 125 bit0 to 1031</td><td>1036</td><td>Register R 140, R 141</td></tr> <tr> <td>1031</td><td>Register R 125 bit15</td><td>1037</td><td>Register R 142, R 143</td></tr> <tr> <td>1032</td><td>Register R 124, R 125</td><td>1038</td><td>Register R 144, R 145</td></tr> <tr> <td>1033</td><td>Register R 126, R 127</td><td>1039</td><td>Register R 146, R 147</td></tr> <tr> <td></td><td></td><td>1040</td><td>Register R 148, R 149</td></tr> <tr> <td></td><td></td><td>1041</td><td>Register R 150, R 151</td></tr> </tbody> </table>	#	Details	#	Details	1000	Register R 124 bit0 to 1015	1034	Register R 128, R 129	1015	Register R 124 bit15	1035	Register R 130, R 131	1016	Register R 125 bit0 to 1031	1036	Register R 140, R 141	1031	Register R 125 bit15	1037	Register R 142, R 143	1032	Register R 124, R 125	1038	Register R 144, R 145	1033	Register R 126, R 127	1039	Register R 146, R 147			1040	Register R 148, R 149			1041	Register R 150, R 151	○	×																														
#	Details	#	Details																																																																		
1000	Register R 124 bit0 to 1015	1034	Register R 128, R 129																																																																		
1015	Register R 124 bit15	1035	Register R 130, R 131																																																																		
1016	Register R 125 bit0 to 1031	1036	Register R 140, R 141																																																																		
1031	Register R 125 bit15	1037	Register R 142, R 143																																																																		
1032	Register R 124, R 125	1038	Register R 144, R 145																																																																		
1033	Register R 126, R 127	1039	Register R 146, R 147																																																																		
		1040	Register R 148, R 149																																																																		
		1041	Register R 150, R 151																																																																		
1100 to 1131 1132 to 1141	Macro I/F output <table border="1" style="width:100%; border-collapse: collapse;"> <thead> <tr> <th>#</th><th>Details</th><th>#</th><th>Details</th></tr> </thead> <tbody> <tr> <td>1100</td><td>Register R 24 bit0 to 1115</td><td>1134</td><td>Register R 28, R 29</td></tr> <tr> <td>1115</td><td>Register R 24 bit15</td><td>1135</td><td>Register R 30, R 31</td></tr> <tr> <td>1116</td><td>Register R 25 bit0 to 1131</td><td>1136</td><td>Register R 40, R 41</td></tr> <tr> <td>1131</td><td>Register R 25 bit15</td><td>1137</td><td>Register R 42, R 43</td></tr> <tr> <td>1132</td><td>Register R 24, R 25</td><td>1138</td><td>Register R 44, R 45</td></tr> <tr> <td>1133</td><td>Register R 26, R 27</td><td>1139</td><td>Register R 46, R 47</td></tr> <tr> <td></td><td></td><td>1140</td><td>Register R 48, R 49</td></tr> <tr> <td></td><td></td><td>1141</td><td>Register R 50, R 51</td></tr> </tbody> </table>	#	Details	#	Details	1100	Register R 24 bit0 to 1115	1134	Register R 28, R 29	1115	Register R 24 bit15	1135	Register R 30, R 31	1116	Register R 25 bit0 to 1131	1136	Register R 40, R 41	1131	Register R 25 bit15	1137	Register R 42, R 43	1132	Register R 24, R 25	1138	Register R 44, R 45	1133	Register R 26, R 27	1139	Register R 46, R 47			1140	Register R 48, R 49			1141	Register R 50, R 51	○	○																														
#	Details	#	Details																																																																		
1100	Register R 24 bit0 to 1115	1134	Register R 28, R 29																																																																		
1115	Register R 24 bit15	1135	Register R 30, R 31																																																																		
1116	Register R 25 bit0 to 1131	1136	Register R 40, R 41																																																																		
1131	Register R 25 bit15	1137	Register R 42, R 43																																																																		
1132	Register R 24, R 25	1138	Register R 44, R 45																																																																		
1133	Register R 26, R 27	1139	Register R 46, R 47																																																																		
		1140	Register R 48, R 49																																																																		
		1141	Register R 50, R 51																																																																		

(Note) Variables compatible with additional specifications can be used when those additional specifications have been added.

APPENDIX 1. LIST OF VARIABLE NUMBERS

○ : Reading/writing possible
 × : Writing not possible
 — : Unrelated to reading/writing

Variable No. (#)	Details	Read	Write																																																								
1900	Machining parameter C axis rotation radius	○	○																																																								
1901	Machining parameter C axis insertion radius	○	○																																																								
3000	NC alarm (alarm display in the macro program)	—	—																																																								
3001	Cumulative time 1	○	○																																																								
3002	Cumulative time 2	○	○																																																								
3003	Control of single block stop and miscellaneous function finish signal wait	○	○																																																								
3004	Feed hold, feedrate override, G09 valid/invalid	○	○																																																								
3006	Message display/stop (message display in the macro program, block stop)	—	—																																																								
3007	Mirror image	○	×																																																								
3008	Miscellaneous function lock state	○	×																																																								
3009	Miscellaneous function lock invalid	○	○																																																								
3011	Clock (year/month/day)	○	○																																																								
3012	Clock (hour/min./sec.)	○	○																																																								
4001 to 4520	Modal information <table border="1" style="width: 100%; border-collapse: collapse; margin-top: 5px;"> <thead> <tr> <th style="text-align: center;">Modal information</th> <th style="text-align: center;">Pre-read</th> <th style="text-align: center;">Execut-ing</th> <th style="text-align: center;">Interrupt</th> <th style="text-align: center;">Modal information</th> <th style="text-align: center;">Pre-re ad</th> <th style="text-align: center;">Execut-ing</th> <th style="text-align: center;">Inter-rupt</th> </tr> </thead> <tbody> <tr> <td>G code (group 1) to G code (group 19)</td> <td>4001 to 4019</td> <td>4201 to 4219</td> <td>4401 to 4419</td> <td>M code</td> <td>4113</td> <td>4313</td> <td>4513</td> </tr> <tr> <td></td> <td></td> <td></td> <td></td> <td>N code</td> <td>4114</td> <td>4314</td> <td>4514</td> </tr> <tr> <td></td> <td></td> <td></td> <td></td> <td>O code</td> <td>4115</td> <td>4315</td> <td>4515</td> </tr> <tr> <td>Tool nose radius compensation No. D</td> <td>4107</td> <td>4307</td> <td>4507</td> <td>S code</td> <td>4119</td> <td>4319</td> <td>4519</td> </tr> <tr> <td>Feedrate F</td> <td>4109</td> <td>4309</td> <td>4509</td> <td>T code</td> <td>4120</td> <td>4320</td> <td>4520</td> </tr> <tr> <td>Tool length offset No. H</td> <td>4111</td> <td>4311</td> <td>4511</td> <td></td> <td></td> <td></td> <td></td> </tr> </tbody> </table>	Modal information	Pre-read	Execut-ing	Interrupt	Modal information	Pre-re ad	Execut-ing	Inter-rupt	G code (group 1) to G code (group 19)	4001 to 4019	4201 to 4219	4401 to 4419	M code	4113	4313	4513					N code	4114	4314	4514					O code	4115	4315	4515	Tool nose radius compensation No. D	4107	4307	4507	S code	4119	4319	4519	Feedrate F	4109	4309	4509	T code	4120	4320	4520	Tool length offset No. H	4111	4311	4511					○	×
Modal information	Pre-read	Execut-ing	Interrupt	Modal information	Pre-re ad	Execut-ing	Inter-rupt																																																				
G code (group 1) to G code (group 19)	4001 to 4019	4201 to 4219	4401 to 4419	M code	4113	4313	4513																																																				
				N code	4114	4314	4514																																																				
				O code	4115	4315	4515																																																				
Tool nose radius compensation No. D	4107	4307	4507	S code	4119	4319	4519																																																				
Feedrate F	4109	4309	4509	T code	4120	4320	4520																																																				
Tool length offset No. H	4111	4311	4511																																																								
5001 to 5114	Position information <table border="1" style="width: 100%; border-collapse: collapse; margin-top: 5px;"> <thead> <tr> <th style="text-align: center;">Position information \ Axis</th> <th style="text-align: center;">1st axis</th> <th style="text-align: center;">2nd axis</th> <th style="text-align: center;">3rd axis</th> <th style="text-align: center;">4th axis</th> <th style="text-align: center;">5th axis to 14th axis</th> </tr> </thead> <tbody> <tr> <td>Block end point</td> <td>5001</td> <td>5002</td> <td>5003</td> <td>5004</td> <td>5005 to 5014</td> </tr> <tr> <td>Machine coordinates</td> <td>5021</td> <td>5022</td> <td>5023</td> <td>5024</td> <td>5025 to 5034</td> </tr> <tr> <td>Workpiece coordinates</td> <td>5041</td> <td>5042</td> <td>5043</td> <td>5044</td> <td>5045 to 5054</td> </tr> <tr> <td>Skip coordinates</td> <td>5061</td> <td>5062</td> <td>5063</td> <td>5064</td> <td>5065 to 5074</td> </tr> <tr> <td>Tool position offset amount</td> <td>5081</td> <td>5082</td> <td>5083</td> <td>5084</td> <td>5085 to 5094</td> </tr> <tr> <td>Servo deviation amount</td> <td>5101</td> <td>5102</td> <td>5103</td> <td>5104</td> <td>5105 to 5114</td> </tr> </tbody> </table>	Position information \ Axis	1st axis	2nd axis	3rd axis	4th axis	5th axis to 14th axis	Block end point	5001	5002	5003	5004	5005 to 5014	Machine coordinates	5021	5022	5023	5024	5025 to 5034	Workpiece coordinates	5041	5042	5043	5044	5045 to 5054	Skip coordinates	5061	5062	5063	5064	5065 to 5074	Tool position offset amount	5081	5082	5083	5084	5085 to 5094	Servo deviation amount	5101	5102	5103	5104	5105 to 5114	○	×														
Position information \ Axis	1st axis	2nd axis	3rd axis	4th axis	5th axis to 14th axis																																																						
Block end point	5001	5002	5003	5004	5005 to 5014																																																						
Machine coordinates	5021	5022	5023	5024	5025 to 5034																																																						
Workpiece coordinates	5041	5042	5043	5044	5045 to 5054																																																						
Skip coordinates	5061	5062	5063	5064	5065 to 5074																																																						
Tool position offset amount	5081	5082	5083	5084	5085 to 5094																																																						
Servo deviation amount	5101	5102	5103	5104	5105 to 5114																																																						

(Note) Variables compatible with additional specifications can be used when those additional specifications have been added.

APPENDIX 1. LIST OF VARIABLE NUMBERS

○ : Reading/writing possible
 ✕ : Writing not possible
 — : Unrelated to reading/writing

Variable No. (#)	Details	Read	Write																																																																		
5201 to 5334 7001 to 8914	<p>Workpiece coordinate offset</p> <table border="1" style="width: 100%; border-collapse: collapse; margin-top: 10px;"> <thead> <tr> <th style="text-align: center;">Workpiece \ Axis</th> <th style="text-align: center;">1st axis</th> <th style="text-align: center;">2nd axis</th> <th style="text-align: center;">3rd axis</th> <th style="text-align: center;">4th axis</th> <th style="text-align: center;">5th axis to 14th axis</th> </tr> </thead> <tbody> <tr> <td>EXT</td> <td style="text-align: center;">5201</td> <td style="text-align: center;">5202</td> <td style="text-align: center;">5203</td> <td style="text-align: center;">5204</td> <td style="text-align: center;">5205 to 5214</td> </tr> <tr> <td>G54</td> <td style="text-align: center;">5221</td> <td style="text-align: center;">5222</td> <td style="text-align: center;">5223</td> <td style="text-align: center;">5224</td> <td style="text-align: center;">5225 to 5234</td> </tr> <tr> <td>G55</td> <td style="text-align: center;">5241</td> <td style="text-align: center;">5242</td> <td style="text-align: center;">5243</td> <td style="text-align: center;">5244</td> <td style="text-align: center;">5245 to 5254</td> </tr> <tr> <td>G56</td> <td style="text-align: center;">5261</td> <td style="text-align: center;">5262</td> <td style="text-align: center;">5263</td> <td style="text-align: center;">5264</td> <td style="text-align: center;">5265 to 5274</td> </tr> <tr> <td>G57</td> <td style="text-align: center;">5281</td> <td style="text-align: center;">5282</td> <td style="text-align: center;">5283</td> <td style="text-align: center;">5284</td> <td style="text-align: center;">5285 to 5294</td> </tr> <tr> <td>G58</td> <td style="text-align: center;">5301</td> <td style="text-align: center;">5302</td> <td style="text-align: center;">5303</td> <td style="text-align: center;">5304</td> <td style="text-align: center;">5305 to 5314</td> </tr> <tr> <td>G59</td> <td style="text-align: center;">5321</td> <td style="text-align: center;">5322</td> <td style="text-align: center;">5323</td> <td style="text-align: center;">5324</td> <td style="text-align: center;">5325 to 5334</td> </tr> <tr> <td>G54.1 P1</td> <td style="text-align: center;">7001</td> <td style="text-align: center;">7002</td> <td style="text-align: center;">7003</td> <td style="text-align: center;">7004</td> <td style="text-align: center;">7005 to 7014</td> </tr> <tr> <td style="text-align: center;">to</td> <td style="text-align: center;">:</td> <td style="text-align: center;">:</td> <td style="text-align: center;">:</td> <td style="text-align: center;">:</td> <td style="text-align: center;">:</td> </tr> <tr> <td>G54.1 P96</td> <td style="text-align: center;">8901</td> <td style="text-align: center;">8902</td> <td style="text-align: center;">8903</td> <td style="text-align: center;">8904</td> <td style="text-align: center;">8905 to 8914</td> </tr> </tbody> </table>	Workpiece \ Axis	1st axis	2nd axis	3rd axis	4th axis	5th axis to 14th axis	EXT	5201	5202	5203	5204	5205 to 5214	G54	5221	5222	5223	5224	5225 to 5234	G55	5241	5242	5243	5244	5245 to 5254	G56	5261	5262	5263	5264	5265 to 5274	G57	5281	5282	5283	5284	5285 to 5294	G58	5301	5302	5303	5304	5305 to 5314	G59	5321	5322	5323	5324	5325 to 5334	G54.1 P1	7001	7002	7003	7004	7005 to 7014	to	:	:	:	:	:	G54.1 P96	8901	8902	8903	8904	8905 to 8914	○	○
Workpiece \ Axis	1st axis	2nd axis	3rd axis	4th axis	5th axis to 14th axis																																																																
EXT	5201	5202	5203	5204	5205 to 5214																																																																
G54	5221	5222	5223	5224	5225 to 5234																																																																
G55	5241	5242	5243	5244	5245 to 5254																																																																
G56	5261	5262	5263	5264	5265 to 5274																																																																
G57	5281	5282	5283	5284	5285 to 5294																																																																
G58	5301	5302	5303	5304	5305 to 5314																																																																
G59	5321	5322	5323	5324	5325 to 5334																																																																
G54.1 P1	7001	7002	7003	7004	7005 to 7014																																																																
to	:	:	:	:	:																																																																
G54.1 P96	8901	8902	8903	8904	8905 to 8914																																																																
10001 to 18040 2001 to 2940	<p>Tool offset</p> <p>The tool data can be read and a value can be substituted using the variable number. Either the #10000 address or #2000 address variables can be used. The last three digits of the variable number correspond to the tool offset number.</p> <table border="1" style="width: 100%; border-collapse: collapse; margin-top: 10px;"> <thead> <tr> <th colspan="2" style="text-align: center;">Variable number range</th> <th style="text-align: center;">Type I</th> <th style="text-align: center;">Type II</th> </tr> </thead> <tbody> <tr> <td style="text-align: center;">#10001 to #10000 + n</td> <td style="text-align: center;">#2001 to #2000 + n</td> <td style="text-align: center;">○</td> <td style="text-align: center;">○</td> </tr> <tr> <td style="text-align: center;">#11001 to #11000 + n</td> <td style="text-align: center;">#2201 to #2200 + n</td> <td style="text-align: center;">—</td> <td style="text-align: center;">○</td> </tr> <tr> <td style="text-align: center;">#16001 to #16000 + n</td> <td style="text-align: center;">#2401 to #2400 + n</td> <td style="text-align: center;">—</td> <td style="text-align: center;">○</td> </tr> <tr> <td style="text-align: center;">#17001 to #17000 + n</td> <td style="text-align: center;">#2601 to #2600 + n</td> <td style="text-align: center;">—</td> <td style="text-align: center;">○</td> </tr> </tbody> </table> <p style="margin-top: 10px;">n: Number of compensation sets Standard 40 sets Additional specifications 800 sets</p> <p>The tool offset data is the same as other variables, and is data with decimal points. Thus, when substituting data less than the decimal point, command the decimal point.</p>	Variable number range		Type I	Type II	#10001 to #10000 + n	#2001 to #2000 + n	○	○	#11001 to #11000 + n	#2201 to #2200 + n	—	○	#16001 to #16000 + n	#2401 to #2400 + n	—	○	#17001 to #17000 + n	#2601 to #2600 + n	—	○	○	○																																														
Variable number range		Type I	Type II																																																																		
#10001 to #10000 + n	#2001 to #2000 + n	○	○																																																																		
#11001 to #11000 + n	#2201 to #2200 + n	—	○																																																																		
#16001 to #16000 + n	#2401 to #2400 + n	—	○																																																																		
#17001 to #17000 + n	#2601 to #2600 + n	—	○																																																																		

(Note) Variables compatible with additional specifications can be used when those additional specifications have been added.


APPENDIX 2. FIXED CYCLE PROGRAM

The subprograms in the standard fixed cycles G73, G74, G76 and G81 to G89 can be input/output and edited.

It is also possible to create separate subprograms, and use G70, G71, G72, G75, G77, G78 and G79 as user fixed cycles.

These subprograms can also be input/output and edited.

Caution

 Do not change fixed cycle programs without the prior approval of the machine manufacturer.

2.1 Parameters for Fixed Cycle Program Operation

The "INPUT/OUTPUT SCREEN" and "EDIT" screens are used to input/output and edit subprogram data for each G70 to G89 fixed cycle, the same as the operation for normal user-created machining programs. However, the parameter (basic common parameter 110007 Fix_P) must be set beforehand.

(1) Turn ON the parameters for fixed cycle operation.

(1)

Display the BASIC COMMON PARAMETER SCREEN of the machine parameters.
--

(2)

Set "1" to the parameter "110007 Fix_P".		
<table border="1" style="display: inline-table; border-collapse: collapse;"><tr><td style="padding: 2px 10px;">1</td></tr></table> <table border="1" style="display: inline-table; border-collapse: collapse;"><tr><td style="padding: 2px 10px;">INPUT</td></tr></table>	1	INPUT
1		
INPUT		

➔ The "INPUT/OUTPUT SCREEN" and "EDIT" screens are dedicated for fixed cycle program operation while this parameter is valid.
(The machining programs handled here become fixed cycle control programs.)
Only fixed cycle programs are displayed in the program list.

(Note 1) Set "0" to the parameter "110007 Fix_P" after the operations related to fixed cycle programs are finished. (This enables the normal user created machining programs to be operated.) This parameter is set to "0" when the power is turned ON.

2.2 Input/Output of Fixed Cycle Programs

The fixed cycle programs are transferred (input/output with external devices) on the INPUT/OUTPUT SCREEN. Confirm that the fixed cycle operation parameter "110007 Fix_P" on the BASIC COMMON PARAMETER SCREEN is set to "1".

The operation method is the same as that for transferring user machining programs.

(1) Input (external device → memory)

The operation method is the same as that for user machining programs.

Generally, 12 groups of fixed cycle subprograms configure one set. Thus, when inputting, it is convenient to create types that can be continuously input.

When the data transfer starts, the message "Transferring" is displayed. When the EOR code is read, the data transfer finishes and the message "Transfer complete" is displayed.

After inputting, use the program list and EDIT screen to confirm that the data was correctly registered.

(2) Output (memory → external device)

The operation method is the same as that for user machining programs.

The registered fixed cycle programs can be output one at a time or as a batch of programs.

When the data transfer starts, the message "Transferring" is displayed. When the EOR code is read, the data transfer finishes and the message "Transfer complete" is displayed.

After outputting, verify the data to confirm that there are no output errors.

(3) Erasure

The operation method is the same as that for user machining programs.

The registered fixed cycle programs can be erased one at a time or as a batch of continuous programs.

Set the device, directory and No. of the program to be erased in setting column A. When erasing a batch of continuous programs, set the range of program Nos. to be erased.

The message "Erase complete" is displayed if the data erasure process finishes normally.

If some files that could not be erased remain after a batch of programs is erased, the message "Erase ended. Some file not erased" is displayed.

APPENDIX 2. FIXED CYCLE PROGRAM
2.3 Standard Fixed Cycle Programs

2.3 Standard Fixed Cycle Programs

G81 (O100000810)	Drill, spot drill
------------------	-------------------

<p>G. 1 ; IF [#30] GOTO1 ; G0 Z#2 G#6 H#7 ; #2 = ##5 #3003 = #8 OR 1 ; G1 Z#3 ; #3003 = #8 ; G0 Z - #3 - #2 ; N1 G23%</p>	<p>Fixed cycle block 1 movement command. Check for fixed cycle invalidity.</p> <p>Inhibit single block stop.</p> <p>Retract.</p>
---	---

G82 (O100000820)	Drill, counter boring
------------------	-----------------------

<p>G. 1 ; IF [#30] GOTO1 ; G0 Z#2 G#6 H#7 ; #2 = ##5 #3003 = #8 OR 1 ; G1 Z#3 ; IF [#4 EQ #0] GOTO5 ; G4 P#4 ; N5 #3003 = #8 ; G0 Z - #3 - #2 ; N1 G23%</p>	<p>Fixed cycle block 1 movement command. Check for fixed cycle invalidity.</p> <p>Inhibit single block stop.</p> <p>Dwell.</p> <p>Retract.</p>
---	---

G83 (O100000830)	Deep hole drill cycle
------------------	-----------------------

<p>G. 1 ; IF [#30] GOTO2 ; #29 = #11 #28 = 0 ; G0 Z#2 G#6 H#7 ; #2 = ##5 #3003 = #8 OR 1 ; DO 1 ; #28 = #28 - #11 #26 = - #28 - #29 Z#26 ; IF [ABS [#28] GE [ABS [#3]]] GOTO 1 ; G1 Z#29 ;</p>	<p>Fixed cycle block 1 movement command. Check for fixed cycle invalidity. Define the cutting amount. Initialize the return amount (total cutting amount).</p> <p>Inhibit single block stop.</p> <p>Define the return amount of the next block. Calculate the feed amount. Feed. Does the total cutting amount (return amount) exceed the cut amount? Cutting feed.</p>
--	--

(Continued on the next page)

APPENDIX 2. FIXED CYCLE PROGRAM
2.3 Standard Fixed Cycle Programs

(Continued from the previous page)

<p>G0 Z#28 ; #29 = #11 + #14 ; END1 ; N1 G1 Z#3 – #26 ; #3003 = #8 ; G0 Z – #3 – #2 ; N2 G23%</p>	<p>Retract. Define the cutting amount for block 2 and after. Cutting feed Retract.</p>
---	---

G84 (O100000840)	Tap cycle
------------------	-----------

<p>G. 1 ; IF [#30] GOTO1 ; G0 Z#2 G#6 H#7 ; #2 = ##5 #3003 = #8 OR 1 ; #3004 = #9 OR 3 ; G1 Z#3 ; IF [#4 EQ #0] GOTO5 ; G4 P#4 ; N5 M4 ; #3900 = 1 ; G1 Z – #3 ; #3004 = #9 ; IF [#4 EQ #0] GOTO6 ; G4 P#4 ; N6 M3 ; #3003 = #8 ; G0 Z – #2 ; N1 G23%</p>	<p>Fixed cycle block 1 movement command. Check for fixed cycle invalidity. Inhibit single block stop. Invalidate feed hold/override. Dwell. Rotate the spindle reversely. Dwell. Rotate the spindle forward. Retract.</p>
---	---

G85 (O100000850)	Boring 1
------------------	----------

<p>G. 1 ; IF [#30] GOTO1 ; G0 Z#2 G#6 H#7 ; #2 = ##5 #3003 = #8 OR 1 ; G1 Z#3 ; #3003 = #8 ; Z – #3 ; G0 Z – #2 ; N1 G23%</p>	<p>Fixed cycle block 1 movement command. Check for fixed cycle invalidity. Inhibit single block stop. Retract.</p>
--	---

APPENDIX 2. FIXED CYCLE PROGRAM
2.3 Standard Fixed Cycle Programs

G86 (O100000860)	Boring 2
------------------	----------

G. 1 ;	Fixed cycle block 1 movement command.
IF [#30] GOTO1 ;	Check for fixed cycle invalidity.
G0 Z#2 G#6 H#7 ;	
#2 = ##5	
#3003 = #8 OR 1 ;	Inhibit single block stop.
G1 Z#3 ;	
IF [#4 EQ #0] GOTO5 ;	Dwell.
G4 P#4 ;	Stop the spindle.
N5 M5 ;	Retract.
G0 Z - #3 - #2 ;	
#3003 = #8 ;	
M3 ;	Rotate the spindle forward.
N1 G23%	

G87 (O100000870)	Back boring
------------------	-------------

G. 1 ;	Fixed cycle block 1 movement command.
IF [#30] GOTO1 ;	Check for fixed cycle invalidity.
#3003 = #8 OR 1 ;	Inhibit single block stop.
#26 = #30064 + #30074 ;	
#28 = #12 * COS [#26] + #13 * SIN [#26] ;	
#29 = #13 * COS [#26] - #12 * SIN [#26] ;	
M19 ;	Orient the spindle.
X#28 Y#29 ;	
#3003 = #8 ;	Cancel single block stop inhibition.
G0 Z#2 G#6 H#7 ;	
#3003 = #8 OR 1 ;	Inhibit single block stop.
G1 X - #28 Y - #29 ;	
#3003 = #8 ;	Cancel single block stop inhibition.
M3 ;	Rotate the spindle forward.
#3003 = #8 OR 1 ;	Inhibit single block stop.
Z#3 ;	
M19 ;	Orient the spindle.
G0 X#28 Y#29 ;	Shift.
Z - #2 - #3 ;	G87 is not affected by the G98 or G99 modal.
#3003 = #8 ;	Cancel single block stop inhibition.
X - #28 Y - #29 ;	Shift.
M3 ;	Rotate the spindle forward.
N1 G23%	

APPENDIX 2. FIXED CYCLE PROGRAM
2.3 Standard Fixed Cycle Programs

G88 (O100000880)	Boring 3
------------------	----------

<pre> G. 1 ; IF [#30] GOTO1 ; G0 Z#2 G#6 H#7 ; #2 = ##5 #3003 = #8 OR 1 ; G1 Z#3 ; IF [#4 EQ #0] GOTO5 ; G4 P#4 ; N5 #3003 = #8 ; M5 ; #3003 = #8 OR 1 ; G0 Z - #3 - #2 ; #3003 = #8 ; M3 ; N1 G23%</pre>	<pre> Fixed cycle block 1 movement command. Check for fixed cycle invalidity. Inhibit single block stop. Dwell. Cancel single block stop inhibition. Rotate the spindle forward Inhibit single block stop. Retract. Cancel single block stop inhibition. Rotate the spindle forward.</pre>
---	--

G89 (O100000890)	Boring 4
------------------	----------

<pre> G. 1 ; IF [#30] GOTO1 ; G0 Z#2 G#6 H#7 ; #2 = ##5 #3003 = #8 OR 1 ; G1 Z#3 ; IF [#4 EQ #0] GOTO5 ; G4 P#4 ; N5 #3003 = #8 ; Z - #3 ; G0 Z - #2 ; N1 G23%</pre>	<pre> Fixed cycle block 1 movement command. Check for fixed cycle invalidity. Inhibit single block stop. Dwell. Retract.</pre>
--	---

APPENDIX 2. FIXED CYCLE PROGRAM
2.3 Standard Fixed Cycle Programs

G73 (O100000730)	Step cycle
------------------	------------

G. 1 ;	Fixed cycle block 1 movement command.
IF [#30] GOTO2 ;	Check for fixed cycle invalidity.
#29 = 0	Initialize the total cutting amount.
#28 = #11	Define the cutting amount.
G0 Z#2 G#6 H#7 ;	
#2 = ##5	
#3003 = #8 OR 1 ;	Inhibit single block stop.
DO 1 ;	
#29 = #29 + #11 ;	Increment the total cutting amount counter.
IF [ABS [#29] GE [ABS [#3]]] GOTO1 ;	Does the total cutting amount exceed the cut amount Z?
G1 Z#28 ;	Cutting feed
IF [#4 EQ #0] GOTO5 ;	
G4 P#4 ;	Dwell.
N5 G0 Z – #14 ;	Retract.
#28 = #11 + #14 ;	Define the cutting amount for block 2 and after.
END1 ;	
N1 G1 Z#3 – #29 + #28 ;	Cutting feed
IF [#4 EQ #0] GOTO6 ;	
G4 P#4 ;	Dwell.
N6 #3003 = #8 ;	
G0 Z – #3 – #2 ;	Retract.
N2 G23%	

G74 (O100000740)	Reverse tap cycle
------------------	-------------------

G. 1 ;	Fixed cycle block 1 movement command.
IF [#30] GOTO1 ;	Check for fixed cycle invalidity.
G0 Z#2 G#6 H#7 ;	
#2 = ##5	
#3003 = #8 OR 1 ;	Inhibit single block stop.
#3004 = #9 OR 3 ;	Invalidate feed hold/override.
G1 Z#3 ;	
IF [#4 EQ #0] GOTO5 ;	
G4 P#4 ;	Dwell.
N5 M3 ;	Rotate the spindle forward.
#3900 = 1 ;	
Z – #3 ;	
#3004 = #9 ;	
IF [#4 EQ #0] GOTO6 ;	
G4 P#4 ;	Dwell.
N6 M4 ;	Rotate the spindle reversely.
#3003 = #8 ;	
G0 Z – #2 ;	Retract.
N1 G23%	

APPENDIX 2. FIXED CYCLE PROGRAM
2.3 Standard Fixed Cycle Programs

G76 (O100000760)	Fine boring
------------------	-------------

<pre> IF [#30] GOTO1 ; G. 1 ; IF [#30] GOTO1 ; G0 Z#2 G#6 H#7 ; #2 = ##5 #26 = #30064 + #30074 ; #28 = #12 * COS [#26] + #13 * SIN [#26] ; #29 = #13 * COS [#26] - #12 * SIN [#26] ; #3003 = #8 OR 1 ; G1 Z#3 ; M19 ; X#28 Y#29 ; G0 Z - #3 - #2 ; #3003 = #8 ; X - #28 Y - #29 ; M3 ; N1 G23%</pre>	<pre> Fixed cycle block 1 movement command. Check for fixed cycle invalidity. Inhibit single block stop. Orient the spindle. Shift. Retract. Shift. Rotate the spindle forward.</pre>
--	--

APPENDIX 3. LIST OF ALARMS
3.1 Program Errors

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

These alarms occur during automatic operation, and the causes of these alarm are mainly program errors which occur, for instance, when mistakes have been made in the preparation of the machining programs or when programs conform to the NC specification have not been prepared. (The error No. and message appear on the screen.)

Error No.	Message	Details	Remedy
PR0001	Mode mismatch	Automatic start was attempted in a mode different from that of the device for which operation search was performed.	<ul style="list-style-type: none"> Match the operation mode with that of the searched device, then retry automatic start.
PR0002	Search during auto operation	A search was attempted during program operation.	<ul style="list-style-type: none"> Execute the search after the program operation has finished.
PR0003	Device invalid	A nonexistent or inoperable device was designated.	<ul style="list-style-type: none"> Check the device availability and whether it is usable.
PR0004	No file designation	A program file has not been designated.	<ul style="list-style-type: none"> Reconsider the program.
PR0005	No file	The designated program file does not exist.	<ul style="list-style-type: none"> Reconsider the program.
PR0006	No N/B number	The designated sequence No. (or program No.) does not exist, or a block having the designated block No. does not exist.	<ul style="list-style-type: none"> Reconsider the program.
PR0010	No. of simultaneous axes over	The number of axis addresses commanded in the same block exceeds the specifications.	<ul style="list-style-type: none"> Divide the alarm block command into two. Check the specifications.
PR0011	Illegal axis address	The axis address commanded by the program and the axis address set by the parameter do not match.	<ul style="list-style-type: none"> Revise the axis names in the program.
PR0020	Division error	An axis command that cannot be divided by the command unit was issued.	<ul style="list-style-type: none"> Reconsider the program.
PR0032	Illegal address	An address not listed in the specifications has been designated.	<ul style="list-style-type: none"> Check and revise the program address. Check the specifications.
PR0033	Format error	The command format in the program is not correct.	<ul style="list-style-type: none"> Reconsider the program.
PR0034	Illegal G code	A G code not listed in the specifications has been used.	<ul style="list-style-type: none"> Check and correct the G code address in the program.
PR0035	Address range over	The setting range for the addresses has been exceeded.	<ul style="list-style-type: none"> Reconsider the program.
PR0036	Program end error	<ul style="list-style-type: none"> "EOR" has been read during tape and memory operation. M02 and M30 are not entered at the end of the program. M99 is not entered at the end of the subprogram. (Or, G23 or G68 is not entered at the end of the subprogram when it is compliant with an M2/M0 format.) 	<ul style="list-style-type: none"> Reconsider the program.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0037	Label and N number zero	Zero has been designated for a program number or sequence number.	<ul style="list-style-type: none"> • The program numbers are designated across a range from 1 to 99999999. • The sequence numbers are designated across a range from 1 to 99999.
PR0038	No additional Op block skip	"/n" has been issued even though there are no optional block skip addition specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0039	No EOB	There was no EOB code at the end of the program block, and an EOR (%) code resulted.	<ul style="list-style-type: none"> • Use an EOB code at the end of the program block.
PR0040	Unused address exists	More address than those that can be simultaneously commanded in one block were commanded. Number of simultaneous commands in one block: M = maximum 4 commands, other than M = 1	<ul style="list-style-type: none"> • Reconsider the program.
PR0041	Overlap G code exists	Multiple G codes were commanded in the same group.	<ul style="list-style-type: none"> • Reconsider the program.
PR0042	Meaningless H code	An H command other than H0 was commanded independently during tool length offset.	<ul style="list-style-type: none"> • Reconsider the program.
PR0043	No end point operation in arc	The end point coordinate command has not been issued for the R-designated arc.	<ul style="list-style-type: none"> • Reconsider the program.
PR0050	No inch/mm spec.	A command has been issued with a G code for inch/mm conversion even though there is no inch/mm conversion G code specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0060	Compensation length over	The commanded movement distance is excessive. (It exceeds 2^{31})	<ul style="list-style-type: none"> • Reconsider the program.
PR0061	No 1-direction positioning spec.	The G60 command has been issued even though there is no uni-direction positioning specifications.	<ul style="list-style-type: none"> • Check the specifications. • Change the G60 program command to G00.
PR0062	No F command	<p>The feedrate command has not been entered.</p> <p>This alarm occurs because the movement modal command is set to G01 when the power is turned ON, and so movement will be based on the G1 command, provided that a movement command is issued, even without G01 having been designated in the program.</p>	<ul style="list-style-type: none"> • Designate an F command in the program.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0063	No high speed mode spec.	The high speed mode command has been issued even though there are no specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0070	Arc end point deviation large	There is an error in the arc start and end points as well as in the arc center.	<ul style="list-style-type: none"> • Check the numerical values of the addresses that specify the start and end points as well as the arc center in the program. • Check the "+" and "-" directions of the address numerical values.
PR0071	Can't calculate arc center	The arc center is not calculated during R-specified circular interpolation.	<ul style="list-style-type: none"> • Check the numerical values of the addresses in the program.
PR0072	No helical spec.	A helical command was issued even though there are no helical specifications. Three axes were commanded for the arc command.	<p>Check the specifications.</p> <ul style="list-style-type: none"> • If this is not a helical command, move the linear axes command to the next block.
PR0073	No scroll spec.	A spiral command was issued despite the fact that such a command does not exist in the specifications.	<ul style="list-style-type: none"> • The G02.1 and G03.1 commands are issued for circular interpolation. • Check the spiral specifications.
PR0080	No hypothetical axis spec.	A hypothetical axis command (G07) was issued despite the fact that such a command does not exist in the specifications.	<ul style="list-style-type: none"> • Check the hypothetical axis specifications. • Change the G07 command.
PR0090	No thread cutting spec.	A thread cutting command was issued despite the fact that such a command does not exist in the specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0093	Screw pitch err	The thread lead (thread pitch) is incorrect.	<ul style="list-style-type: none"> • Correctly set the thread lead.
PR0100	No cylinder interpolation spec.	Cylindrical interpolation was commanded even though there are no cylindrical interpolation specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0101	Cylinder mode error	A command not permitted with the cylindrical mode was issued.	<ul style="list-style-type: none"> • Reconsider the machining program.
PR0110	Plane selected during pattn rot	A plane selection command (G17, G18, G19) has been issued in a pattern rotation command.	<ul style="list-style-type: none"> • Delete the plane selection command (G17, G18, G19) from the pattern rotation subprogram.
PR0111	Coord rotation plane selection	A plane selection command (G17, G18, G19) has been issued in a coordinate rotation command (G68).	<ul style="list-style-type: none"> • Upon completion of the G68 command, the plane selection command must be issued following the G69 command (coordinate rotation cancel). (For the program based on the M2/M0 format, the coordinate rotation command is G68.1 and the coordinate rotation cancel command is G69.1.)

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0112	Plane selected during R compen	A plane selection command (G17, G18, G19) has been issued when the tool radius compensation command (G41, G42) is issued.	<ul style="list-style-type: none"> • Issue the plane selection command after the tool radius compensation command has been completed (issue the G40 cancel command.)
PR0120	No synchronous feed spec.	Synchronous feed was commanded even though there are no synchronous feed specifications.	<ul style="list-style-type: none"> • Check the specifications. • Change the synchronous feed command (G95) to a feed per minute command (G94). (Also change the F command value.)
PR0121	In F0 arc modal mode	The F0 command (F1-digit rapid traverse command) has issued when the circular interpolation command (G02, G03) was issued during the F1-digit feed command.	<ul style="list-style-type: none"> • Rapid traverse cannot be commanded and so any F1-digit command except F0 should be designated for the circular interpolation command. • Issue the G0 or G1 command unless the block which caused the alarm is the circular command block.
PR0122	No auto corner override spec.	An automatic corner override command (G62) was issued despite the fact that such a command does not exist in the specifications.	<ul style="list-style-type: none"> • Check the specifications. Delete the G62 command from the program.
PR0124	No G93 spec.	The inverse time feed option is not provided.	<ul style="list-style-type: none"> • Check the specifications.
PR0125	G93 mode error	A G command that cannot be commanded in the G93 mode was issued. Or, the G93 was commanded in a mode incapable of inverse time feed.	<ul style="list-style-type: none"> • Reconsider the program.
PR0140	No t-pos offset	The tool position compensation option is not provided.	<ul style="list-style-type: none"> • Check the specifications.
PR0141	T-pos during pat/cood rot	Tool position compensation was attempted during figure rotation or program coordinate rotation.	<ul style="list-style-type: none"> • Reconsider the program.
PR0142	T-pos offset G3 error	An arc command with which position compensation is not possible was issued.	<ul style="list-style-type: none"> • Reconsider the program.
PR0150	No nose radius compensation	A tool radius compensation command (G41, G42) has been issued even though such a command does not exist in the specifications.	<ul style="list-style-type: none"> • Check the tool radius compensation specification.
PR0151	Radius compen in arc modal	A compensation command (G40, G41, G42, G43, G44) has been issued in the arc modal (G02, G03).	<ul style="list-style-type: none"> • Issue the linear command (G01) or rapid traverse command (G00) in the compensation command block or cancel block. (Set the modal to linear interpolation.)

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0152	No intersection	The intersection is not obtained when one block is skipped during the processing of an interference block with the execution of a tool radius compensation command (G41, G42).	<ul style="list-style-type: none"> Reconsider the program.
PR0153	Compensation interference	An interference error has arisen while the tool radius compensation command (G41, G42) was being executed.	<ul style="list-style-type: none"> Reconsider the program.
PR0154	No 3D-compensation spec.	A three-dimensional compensation command was issued even though there are no three-dimensional compensation specifications.	<ul style="list-style-type: none"> Check the specifications.
PR0155	Fix cyc executed during compen	A fixed cycle command has been issued in the radius compensation mode.	<ul style="list-style-type: none"> The radius compensation mode is established when a fixed cycle command is executed and so the radius compensation cancel command (G40) should be issued.
PR0164	No optional backward spec.	Backward movement was commanded even though there are no backward movement specifications.	<ul style="list-style-type: none"> Check the specifications.
PR0170	No compensation number	The compensation number (D $\circ\circ$) command was not given when the radius compensation command (G41, G42, G43) was issued. Alternatively, the compensation number is larger than the number of sets in the specifications.	<ul style="list-style-type: none"> Add the compensation number command to the compensation command block.
PR0171	No G10 spec.	G10 command has been issued even though such a command does not exist in the specifications.	<ul style="list-style-type: none"> Check the specifications.
PR0172	G10 L-number error	The address command is not correct when the G10 command is issued.	<ul style="list-style-type: none"> Check the address L-number of the G10 command and correct the number.
PR0174	No G11 spec.	The G11 command has been issued even though there are no program tool compensation input (G11) specifications.	<ul style="list-style-type: none"> Check the specifications.
PR0180	No fixed cycle	A fixed cycle command was issued even though there are not fixed cycle (G72 to G89) specifications.	<ul style="list-style-type: none"> Check the specifications. Correct the program.
PR0181	No spindle command (Tap cycle)	The spindle speed command has not been issued when the hole drilling fixed cycle command is given.	<ul style="list-style-type: none"> Issue the spindle speed command (S) when the hole drilling fixed cycle command G84 or G74 is given.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0182	Synchronous tap error	Connection with the spindle unit was not established.	<ul style="list-style-type: none"> • Check the connection with the spindle. • Check that the spindle encoder exists.
PR0183	No pitch/thread number	The pitch or thread number command has not been issued in the tap cycle of a hole drilling fixed cycle command.	<ul style="list-style-type: none"> • Specifies the pitch data and the number of threads via F or E command.
PR0184	Pitch/thread number error	The pitch or thread number command is not correct in the tap cycle of a hole drilling fixed cycle command.	<ul style="list-style-type: none"> • Check the pitch data or the number of threads.
PR0220	No special fixed cycle spec.	There are no special fixed cycle specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0221	No. of special fixed holes = 0	0 has been specified as the number of holes in special fixed cycle.	<ul style="list-style-type: none"> • Reconsider the program.
PR0222	G36 angle interval error	0 has been specified as the angle internal with G36.	<ul style="list-style-type: none"> • Reconsider the program.
PR0223	G12/G13 radius error	The radius value is less than the compensation amount specified with G12 or G13.	<ul style="list-style-type: none"> • Reconsider the program.
PR0224	No G12/G13 spec.	There are no circular cutting specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0230	Sub-program nesting over	A subprogram has been called 8 or more times in succession from the subprogram.	<ul style="list-style-type: none"> • Check the number of subprogram calls and correct the program so that it does not exceed 8 times.
PR0231	No N-Number	The sequence number set at sub-program call time, at return from the subprogram or specified by GOTO, was not set.	<ul style="list-style-type: none"> • Specify the sequence numbers in the call block of the subprogram.
PR0232	No program No.	When the subprogram is called, it is not registered.	<ul style="list-style-type: none"> • Register the subprogram.
PR0240	No variable command	A variable command was issued even though there are no variable command (#○○) specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0241	No variable number	The variable number commanded is higher than the variable numbers in the specifications.	<ul style="list-style-type: none"> • Check the specifications. • Check the program variable number.
PR0242	Variable definition = none	The "=" sign has not been commanded when a variable is defined.	<ul style="list-style-type: none"> • Designate the "=" sign in the variable definition of the program.
PR0243	Can't use variables	An unavailable variable has been commanded in the left side or right side of a formula.	<ul style="list-style-type: none"> • Correct the program.
PR0244	No. of var definitions over	The No. of variables that can be defined in one block was exceeded.	<ul style="list-style-type: none"> • Correct the program.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0250	No pattern rotation	A pattern rotation command was issued even though there are no such command specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0251	Pattern rotation duplicated	A pattern rotation command was issued during another pattern rotation command.	<ul style="list-style-type: none"> • Check the program.
PR0252	Pattern and coord rotation	A pattern rotation command and coordinate rotation command have been designated simultaneously.	<ul style="list-style-type: none"> • Check the program.
PR0260	No coordinates rotation	A coordinate rotation command was issued even though there are no such command specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0270	No user macro	A macro specification was commanded even though there are no such command specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0271	No macro interrupt	A macro interrupt command was issued even though there are no macro interrupt specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0272	NC text/macro text simultaneous	An NC statement and a macro statement exist together in the same block.	<ul style="list-style-type: none"> • Reconsider the program and place the NC statement and macro statement in separate blocks.
PR0273	Macro call nesting over	The frequency of the macro call has exceeded the quantity determined in the specifications.	<ul style="list-style-type: none"> • Reconsider the program and correct it so that the macro calls do not exceed the limit imposed by the specification.
PR0275	Variable number illegal	The number of macro call argument type II sets has exceeded the limit.	<ul style="list-style-type: none"> • Reconsider the program.
PR0276	Macro call cancel independent	A G67 command was issued even though it was not during the G66 command modal.	<ul style="list-style-type: none"> • Reconsider the program. The G67 command is the call cancel command and so the G66 command must be designated first before G67 is issued.
PR0277	Macro alarm message	An alarm command has been issued in #3000.	<ul style="list-style-type: none"> • Refer to OPERATOR MESSAGE on DIAGN screen. • Refer to instruction manual issued by machine manufacturer.
PR0280	'[' and ']' nesting over	The number of parentheses [,] which can be commanded in a single block has exceeded five.	<ul style="list-style-type: none"> • Reconsider the program and correct it so that the number of "[" or "]" does not exceed five.
PR0281	Number of '[' and ']' differ	The number of "[" and "]" parentheses commanded in a single block do not match.	<ul style="list-style-type: none"> • Reconsider the program and correct it so that the "[" and "]" parentheses are paired up properly.
PR0282	Can't calculate	The arithmetic formula is incorrect.	<ul style="list-style-type: none"> • Reconsider the program to correct the arithmetic formula.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0283	Zero division	The denominator of the division is zero.	<ul style="list-style-type: none"> Reconsider the program and correct it so that the denominator for division in the formula is not zero.
PR0284	Overflow value	The integer value exceeded -2^{31} (2^{31-1}) during the operation process.	<ul style="list-style-type: none"> Review the program's operation expression, and correct so that the operated integer value does not exceed -2^{31} (2^{31-1}).
PR0285	Float value overflow	The variable data has overflowed.	<ul style="list-style-type: none"> Reconsider the variable data in the program.
PR0290	IF statement error	There is an error in the IF [<conditional>] GOTO statement.	<ul style="list-style-type: none"> Reconsider the program.
PR0291	WHILE statement error	There is an error in the WHILE [<conditional>] DO - END statement.	<ul style="list-style-type: none"> Reconsider the program.
PR0292	SETVN statement error	There is an error in the SETVN statement when the variable name was set.	<ul style="list-style-type: none"> Reconsider the program. The number of characters in the variable name of the SETVN statement must be 7 or less.
PR0293	DO-END nesting over	The number of []'s for DO - END in the WHILE [<conditional>] DO - END statement has exceeded 27.	<ul style="list-style-type: none"> Reconsider the program and correct it so that the number of the DO - END statements does not exceed 27.
PR0294	DO and END not in proper pairs	The DOs and ENDS are not paired off properly.	<ul style="list-style-type: none"> Reconsider the program and correct it so that the DOs and ENDS are paired off properly.
PR0295	Tape WHILE/ GOTO	There is a WHILE or GOTO statement on the tape during tape operation.	<ul style="list-style-type: none"> During tape operation, a program which includes a WHILE or GOTO statement cannot be executed and so the memory operation mode is established instead.
PR0300	Variable name illegal	The variable names have not been commanded properly.	<ul style="list-style-type: none"> Reconsider the variable names in the program and correct them.
PR0301	Variable name duplicated	The name of the variable has been duplicated.	<ul style="list-style-type: none"> Correct the program so that the name is not duplicated.
PR0350	No scaling spec.	A scaling command (G50, G51) was issued even though there are no such scaling specifications.	<ul style="list-style-type: none"> Check the specifications.
PR0360	No program mirror spec.	A mirror image command (G50.1, G51.1) was issued even though there are no programmable mirror image specifications.	<ul style="list-style-type: none"> Check the specifications.
PR0380	No corner R/C spec.	A command was issued for corner rounding or corner chamfering even though there are no such specifications.	<ul style="list-style-type: none"> Check the specifications. Remove the corner rounding or chamfering command from the program.
PR0382	No corner movement	The block next to corner rounding/ chamfering is not a movement command.	<ul style="list-style-type: none"> Replace the G01 command by the block succeeding the corner rounding/chamfering command.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0383	Corner movement short	In the corner rounding or chamfering command, the movement distance was shorter than the value in the corresponding command.	<ul style="list-style-type: none"> • Make the corner rounding or chamfering less than the movement distance since this distance is shorter than the corner rounding or chamfering.
PR0384	Corner next movement short	When the corner rounding or chamfering command was input, the movement distance in the following block was shorter than the corner rounding or chamfering.	<ul style="list-style-type: none"> • Make the corner rounding or chamfering less than the movement distance since this distance in the following block is shorter than the corner rounding or chamfering.
PR0385	Corner during G0/G33	Corner rounding or chamfering command was given during G00 or G33 modal.	<ul style="list-style-type: none"> • Reconsider the program.
PR0386	Can't execute corner R (C)	The corner R/C movement cannot be created.	<ul style="list-style-type: none"> • Reconsider the program.
PR0390	No geometric spec.	A geometric command was issued even though there are no specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0391	No geometric arc spec.	An arc command was issued with the geometric command even though there are no arc specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0392	Angle less than 1 deg (GEO MT)	The angle difference of two straight lines in the geometric command was within 1°.	<ul style="list-style-type: none"> • Correct the geometric angle.
PR0393	Inc value in 2nd block (GEO MT)	The second block of the geometric command was an incremental value command.	<ul style="list-style-type: none"> • Command the second geometric block with an absolute value.
PR0394	No linear move command (GEO MT)	The second block of the geometric command was not linear.	<ul style="list-style-type: none"> • Command G01 in the second geometric block.
PR0395	Illegal address (GEO MT)	The plane selection horizontal axis and plane selection vertical axis are not commanded in the second block of the geometric command. There is no A command in the second block of the geometric command.	<ul style="list-style-type: none"> • Reconsider the program.
PR0396	Plane selected in GEO MT ctrl	The plane was changed between the first and second blocks of the geometric command.	<ul style="list-style-type: none"> • Change the plane before the geometric command.
PR0420	No parameter input spec.	A parameter input command was issued even though there are no parameter input specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0421	Parameter input error	Insufficient data was input in parameter. An address other than P, S, N, A or D was commanded. The P, S, N, A or D value is illegal. A character-string type parameter was input.	<ul style="list-style-type: none"> • Enter the correct parameter input data. • Reconsider the program.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0430	Return incomplete axis exists	A command was issued to move an axis, which has not returned to the reference point, away from that reference point.	<ul style="list-style-type: none"> Execute reference point return manually.
PR0431	No 2nd 3rd 4th ref-p ret spec.	The 2nd, 3rd and 4th reference point return was commanded even though there are no 2nd, 3rd and 4th reference point return specifications.	<ul style="list-style-type: none"> Check the specifications.
PR0435	G27 and M commands in same block	An M individual command was issued simultaneously in the G27 command block.	<ul style="list-style-type: none"> An M individual command cannot be issued in a G27 command block and so the G27 command and M individual command must be placed in separate blocks.
PR0436	G29 and M commands in same block	An M individual command was issued simultaneously in the G29 command block.	<ul style="list-style-type: none"> An M individual command cannot be issued in a G29 command block and so the G29 command and M individual command must be placed in separate blocks.
PR0437	No. G54.1 spec.	Although there are no specifications of workpiece coordinate system offset set addition, G54.1 has been specified.	<ul style="list-style-type: none"> Check the specifications.
PR0438	Local coord set during exp wk co	Local coordinate system command (G52) is given during the G54.1 mode.	<ul style="list-style-type: none"> Reconsider the program.
PR0451	No check-before-move spec.	The pre-movement stroke check (G22/G23 (G22.1/G23.1 for M2/M0 format)) was commanded even though there are no pre-movement stroke check function specifications.	<ul style="list-style-type: none"> Check the specifications.
PR0452	Movement limit	When the axis movement start point or end point is in the prohibited area designated with the pre-movement stroke check function, an alarm will occur before executing that movement block.	<ul style="list-style-type: none"> Reconsider the axis address coordinate value in the program.
PR0460	Tape I/O error	An error has arisen in the tape reader or, alternatively, in the printer during macro printing.	<ul style="list-style-type: none"> Check the power supply and cable of the connected devices. Check the input/output device parameters.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0461	File I/O	A file of the machining program cannot be read.	<ul style="list-style-type: none"> • If this error occurs during memory operation, the program stored in the memory may be destroyed. • Output the program and all other tool data and work offset data, then format the memory. • When using the FD, replace the floppy disk. • During IC card operation, confirm that the IC card is correctly inserted.
PR0462	Computer Link error	A communication error occurred during computer link A, B, high-speed server running.	<ul style="list-style-type: none"> • After resetting, check the cables and power supply of the connected devices, and computer link parameters.
PR0463	Read forced quit	The process was interrupted during input/output by resetting.	<ul style="list-style-type: none"> • Carry out input/output again.
PR0470	No Computer Link A	Computer link A was executed even though there are no computer link A specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0471	No Computer Link B	Computer link B was executed even though there are no computer link B specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0490	No disk operation	Disk operation was attempted even though there are no disk operation specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0491	No FLD operation	FLD operation was attempted even though there are no FLD operation specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0492	No IC card operation	IC card operation was attempted even though there are no IC card operation specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0493	No dataserver operation	Data server operation was attempted when the data server operation specifications were not available.	<ul style="list-style-type: none"> • Check the specifications.
PR0600	No auto TLM	An automatic tool length measurement command (G37) was executed even though there are no such command specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0601	No skip	A skip command (G31) was issued even though there are no such command specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0602	No multi skip	The multi-step skip command (G31.n) was issued even though there are no multi-step skip specifications.	<ul style="list-style-type: none"> • Check the specifications.
PR0603	Skip speed 0	The skip speed is 0.	<ul style="list-style-type: none"> • Designate the skip speed.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0604	G37 illegal axis	No axis or more than 2 axes was designated in the automatic tool length measurement block.	<ul style="list-style-type: none"> Designate only one axis.
PR0605	H and G37 commands in same block	The H code is in the same block as the automatic tool length measurement command.	<ul style="list-style-type: none"> Designate the H code before the automatic tool length measurement command block.
PR0606	H command not found before G37	The H code was not yet designated in automatic tool length measurement.	<ul style="list-style-type: none"> Designate the H code before the automatic tool length measurement command block.
PR0607	Signal turned illegally with G37	Before the area commanded by the D command or the parameter deceleration area D, the measurement position arrival signal went ON, or resulted in not going ON.	<ul style="list-style-type: none"> Reconsider the program.
PR0608	Skip during radius compen	A skip command was issued during the radius compensation processing.	<ul style="list-style-type: none"> Designate a radius compensation cancel command (G40), or remove the skip command.
PR0611	No hi-prec (G61.1) spec.	A high-accuracy mode (G61.1) command was issued even though there are no such command specifications.	<ul style="list-style-type: none"> Check the specifications. Remove the high-accuracy mode command from the program.
PR0612	No hi-prec die (G61.2/3) spec.	A high-accuracy (die) (G61.2/G61.3) command was issued even though there are no such command specifications.	<ul style="list-style-type: none"> Check the specifications. Remove the high-accuracy (die) command from the program.
PR0615	No hi-prec part (G61.5) spec.	A high-accuracy (part) (G61.5) command was issued even though there are no such command specifications.	<ul style="list-style-type: none"> Check the specifications. Remove the high-accuracy (part) command from the program.
PR0621	NURBS knot error	The knot was not correctly designated in the NURBS command.	<ul style="list-style-type: none"> Designate the knot following the NURBS command format.
PR0622	NURBS weight error	The weight was not correctly designated in the NURBS command.	<ul style="list-style-type: none"> Designate the weight following the NURBS command format.
PR0623	NURBS degree error	The number of stages was not correctly designated in the NURBS command.	<ul style="list-style-type: none"> Designate the number of stages following the NURBS command format.
PR0624	NURBS control point error	The control point was not correctly designated in the NURBS command.	<ul style="list-style-type: none"> Designate the control point following the NURBS command format.
PR0625	NURBS misc error	The general precautions were not observed for the NURBS command.	<ul style="list-style-type: none"> Check the commands and modes that cannot be used with the NURBS interpolation. Check the restrictions described in the Operation Manual.
PR0626	No NURBS spec.	The NURBS interpolation command was issued even though there are no specifications.	<ul style="list-style-type: none"> Check the specifications.

APPENDIX 3. LIST OF ALARMS

3.1 Program Errors

Error No.	Message	Details	Remedy
PR0900	No normal control	A normal line direction control command (G40.1/G41.1/G42.1) was issued even though there are no such command specifications.	<ul style="list-style-type: none"> • Check the specifications. • Remove the normal line direction control command from the program.
PR0901	G92 command on normal axis	The normal line control axis work offset rewrite command (G92) was issued during normal line control.	<ul style="list-style-type: none"> • Reconsider the program.
PR0902	Normal axis set as linear axis	Normal line control was set for a linear axis.	<ul style="list-style-type: none"> • Set normal line control for a rotation axis.
PR0903	At block joint turn angle over	The turning angle at the block seams exceeded the limit during normal line control.	<ul style="list-style-type: none"> • Reconsider the program.
PR0920	No 3DIM coord conversion spec.	There is no three-dimensional coordinate conversion function.	<ul style="list-style-type: none"> • Check the specifications.
PR0921	3DIM coord convert error	A G code that cannot be commanded during the three-dimensional coordinate conversion modal was issued.	<ul style="list-style-type: none"> • If the control parameter "Init cnst prphl spd" is ON, turn the parameter OFF, or designate constant surface speed control cancel (G97).
PR0922	Illegal 3DIM coord convert	A three-dimensional coordinate conversion command was issued in a modal incapable of three-dimensional coordinate conversion.	<ul style="list-style-type: none"> • If the control parameter "Initial high precis" is ON, turn the parameter OFF, or command the cutting mode (G64).
PR0990	Illegal process	An error occurred during the program analysis process, and it became impossible to continue.	<ul style="list-style-type: none"> • Call the service center with as many details about the operation status as possible.

APPENDIX 3. LIST OF ALARMS
3.2 Program Warnings

3.2 Program Warnings

No.	Message	Details	Remedy
PRW0001	Overlap address exists	Because the No. of addresses that can be simultaneously commanded in one block was exceeded, the previously designated address was ignored. No. of addresses that can be simultaneously commanded in one block : M = Max. 4 addresses. Other than M = 1 address	<ul style="list-style-type: none"> • Reconsider the program.
PRW0002	Overlap G code exists	Because several G codes were commanded in the same group, the previously designated G code was ignored.	<ul style="list-style-type: none"> • Reconsider the program.
PRW0003	Ignored G code exists	A G code exists that was ignored due to a G code priority order check.	<ul style="list-style-type: none"> • Reconsider the program.
PRW0004	Unused address exists	An address exists that was not used in the program block.	<ul style="list-style-type: none"> • Reconsider the program.
PRW0005	Ref point ret in pattern rot	Reference point return was attempted while rotating the figure.	<ul style="list-style-type: none"> • Reconsider the program.
PRW0006	No end point operation in arc	There is no end point coordinate command for the R-designated arc.	<ul style="list-style-type: none"> • Reconsider the program.
PRW0007	Cylinder mode warning	A command illegal for the cylindrical mode was issued.	<ul style="list-style-type: none"> • Reconsider the program.

APPENDIX 4. M2/M0 MACRO CONVERTOR

4.1 Restrictions

During tape operation in the M2/M0 format, M2/M0 macro conversion is performed and data is stored in buffers.

Data in one block (a maximum of 64 characters) can only be read into one buffer.

Macro single (although the control parameter Macro single is not set to on) operation is performed and preread is not made.

② Program display and buffer correction during tape operation

During tape operation in the M2/M0 format, only two to four program lines can be displayed on each screen of position display and buffer correction because of the tape operation system described above.

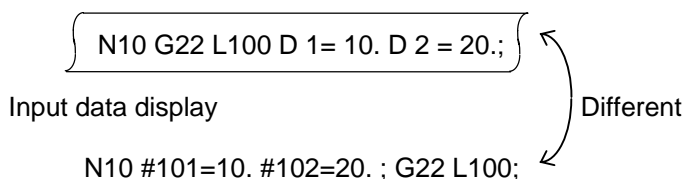
(3) Data input

① Data input

When the system is operated in the M2/M0 format, data on which the M2/M0 macro convertor has been executed is input.

When data is input, the data after M2/M0 conversion is displayed. Thus, the actual tape contents may differ from the input data display.

Tape contents

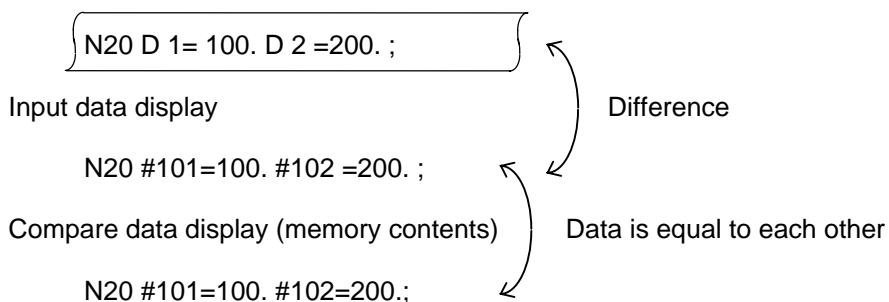


② Data comparison

When the system is operated in the M2/M0 format, the M2/M0 macro convertor is executed on data input from an input device, and the resultant data is compared with the data in memory.

In data comparison, input data after M2/M0 macro conversion is displayed. Thus, the actual tape contents may differ from the input data.

Tape contents



APPENDIX 4. M2/M0 MACRO CONVERTOR
4.2 Operation

When the resultant data is divided into blocks after the M2/M0 macro convertor has been executed on input data, the input data is displayed as shown below:

Tape contents

N10 G22 L100 D 1=100. D 2=200. ;

1) Comparing with the first block of blocks into which data is divided

Input data display

N10 #101=100. #102=200. ; G22 L100;

Compare data display (memory contents)

N10 #101=100. #102=200.;

Data is equal to each other

2) Comparing with the second block of blocks into which data is divided

Input data display

G22 L100;

Compare data display (memory contents)

G22 L100;

Data is equal to each other

4.2 Operation

If normal tape input and tape operation are performed when the machine parameter M2/M0 format is effective, tape input and tape operation are performed while macro conversion is being performed.

4.3 Detailed description

Macro variable conversion is made according to Tables 1 and 2 given below. However, when a variable contained in MELDAS-M2/M0 but not in MELDAS 600 and sequent series is encountered, an error results.

(Variable D given in the description is applied when label is L; variable L when label O.)

(1) Conversion of variable contained in M2/M0 but not in M600 and sequent series

Error format:

1ERR();



Block where conversion cannot be made

This is added preceding the block.

Example:

D1=D4008; 1ERR(#101=D4008);

APPENDIX 4. M2/M0 MACRO CONVERTOR
4.3 Detailed description

(2) Error display

After tape input is complete, an error message is displayed in the following case in tape input (except tape operation):

Message	Details	Action
Variable conversion error	It results when a variable contained in M2/M0 but not in M600 and sequent series is encountered or tool offset amount variable is converted when M600 and sequent series tool compensation method is of type II.	Correct the program.

Note: As in (1), error conversion is made during tape operation.
 Perform operation after correcting the program by buffer correction function.

(3) Others

① Conversion is not made on the following D or L:

- 1) Compensation or offset amount D (H) [G12, G13, G39-G42, G43-G44, G45-G48] (See 10 of Table 1.)
- 2) Subprogram call L (A) [G22, G65, G66] (See 12 to 14 of Table 1.)
- 3) Number of fixed cycle repetitions L (P) [G73, G74, G76, G80-G89] (See 15 of Table 1.)

② G command modal

M2/M0 and M600 series differ in the G command modal variable (D4007-D4014, D6007-D6014) contents.

Then, the variables are converted as follows: (See 17 of Table 1.)

D4007	→	[#4010-98]	Return level
D4010	→	[91-#4003]	Absolute/incremental
D4011	→	[#4005-94]	Synchronous/asynchronous
D4012	→	[71-#4006]	Inch/metric
D4014	→	[44-#4008]	Tool length offset

(Interrupt G code modal variables D6007-D6014 are also converted likewise.)

③ () in comment statement

The conversion condition of parentheses to square brackets in the M2/M0 macro convertor conforms to the M2/M0 comment restriction that "if the character following a left parenthesis is a sign, variable character (D or L), or macro word (such as OR, XOR, or SIN), the statement is not regarded as a comment statement". Parentheses () other than in M2/M0 comment restriction are converted into square brackets [].

Example: (SIN) → [SIN]

APPENDIX 4. M2/M0 MACRO CONVERTOR
4.3 Detailed description

④ Parenthesis pair nesting and variable nesting level

Up to five parenthesis pairs can be nested.
 Up to five variables can be nested.

Example: More than five ((((((D1)))))) → [[[[[[D1]]]]]]
 DDDDD1=1 → D#####101=1

⑤ Variable nesting

Conversion of value accompanying nesting cannot be made.

Example D1=2; Assign 2 to common variable D1
 DD1=3; Assign 3 to common variable D2

↓

 #101=2; Assign 3 to common variable #101
 ##101=3; Assign 3 to local variable #2

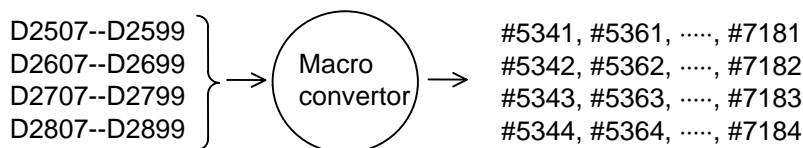
⑥ Variable numbers of program tool offset input (G11) and coordinate read function (G14)

G11 P transfer destination offset number
 Q transfer destination variable number

G14 Variable number where X coordinate is stored
 Variable number where Y coordinate is stored
 Variable number where Z coordinate is stored
 Variable number where α coordinate is stored
 P coordinate type

The G11 Q variable number and G14 X, Y, Z, and α variable numbers cannot be converted.

⑦ In a macro variable conversion, variables that are not defined in the M2/M3 and M600 series cause a "Variable conversion error" except for the following variables which can be converted.



APPENDIX 4. M2/M0 MACRO CONVERTOR
4.3 Detailed description

Table 1 Conversion Pattern Examples

	M2/M0 macro example	MELDAS Standard Format macro example	Remarks
			(Assignment statement)
1	D1=100; (L)	#101=100;	
2	D1/100; (L)	#101=100;	
3	D1/100 D2/200; (L) (L)	#101=100 #102 200;	
4	DD1=100; (L)(L)	##101=100;	
5	D1=D1/D2; (L) (L)(L)	#101=#101/#102;	
6	D1/D1/D2; (L)(L)(L)	#101=#101/#102;	
			(Specified value)
7	G00 XD1/100; (L)	G00 X#101/100;	
8	G00 XD1+D2+100; (L) (L)	G00 X#101+#102+100;	
9	G00 X-D1; (L)	G0 X-#101;	
			(Offset amount)
10	G41 X100D1; (H)	G41 X100 D1; (H)	Tool offset D(H) is not converted.
11	G41 X100DD1; (H)(L)	G41 X100 D#101; (H)	
			(Subprogram call)
12	G22 L1 D1/100; (A) (L)	#101=100 G22 L1; (A)	Conversion to two blocks. Label L(A) is not converted.
13	G65 D1 L5; (A)	G65 D1 L5; (A)	Local variable name D is not converted.
14	G65 DD1 L6; (L) (A)	G65 D#101 L6; (A)	
			(Fixed cycle repetition)
15	G73 L5 (P)	G73.....L5 (P)	Repetition L(P) is not converted.
16	G00 XD1 D2=100;	G00 X#101; #102=100;	(Block division)
17	D1=D4010;	#101=[91-#4003];	(G command modal)

APPENDIX 4. M2/M0 MACRO CONVERTOR
4.3 Detailed description

Table 2 Variable Correspondence between M2/M0 and M600 Series

M2/M0	MELDAS Standard	Function
D1 to D49	#101 to #149	Common variables Note: Standard specifications : 100 sets Additional specifications : 200 sets
D101 D102 D103 D104 D105 D106 D107 D108 D109 D110 D111 D112 to D126 D127 to D132	#1 #2 #3 #7 #8 #9 #10 #11 #4 #5 #6 #12 to #26 #27 to #32	Local variables A B C D E F G H I J K L Z

System variables		

D1000 to D1015 D1016	#1000 to #1015 #1032	Macro interface input 1 bit 1 bit 16 bit (the high-order 16 bits in M600 series are undefined)

D1100 to D1115 D1116	#1100 to #1115 #1132	Macro interface output 1 bit 1 bit 16 bit (the high-order 16 bits in M600 series are undefined)

APPENDIX 4. M2/M0 MACRO CONVERTOR
4.3 Detailed description

M2/M0	MELDAS Standard	Function
D2001 to D2040	#2001 to #2040	Tool offset Standard specifications (40 sets)
D2001 to D2200	#2001 to #2200	Additional specifications (200 sets)
D2001 to D2400	#2001 to #2400	Additional specifications (400 sets)
D1200 to D1202	} Error	Skip signal input
D2500 D2600 D2700 D2800 D2900	#5201 #5202 #5203 #5204 #5205	Workpiece coordinate system offset External workpiece offset X Y Z 4 5 G54
D2501 D2601 D2701 D2801 D2901	#5221 #5222 #5223 #5224 #5225	X Y Z 4 5 G55
D2502 D2602 D2702 D2802 D2902	#5241 #5242 #5243 #5244 #5245	X Y Z 4 5 G56
D2503 D2603 D2703 D2803 D2903	#5261 #5262 #5263 #5264 #5265	X Y Z 4 5 G57
D2504 D2604 D2704 D2804 D2904	#5281 #5282 #5283 #5284 #5285	X Y Z 4 5

APPENDIX 4. M2/M0 MACRO CONVERTOR
4.3 Detailed description

M2/M0	MELDAS Standard	Function
		G58
D2505	#5301	X
D2605	#5302	Y
D2705	#5303	Z
D2805	#5304	4
D2905	#5305	5
		G59
D2506	#5321	X
D2606	#5322	Y
D2706	#5323	Z
D2806	#5324	4
D2906	#5325	5
D3000	#3000	NC alarm
D3001	#3001	Integrating time 1
D3002	#3002	Integrating time 2
D3003	#3003	Control of single block stop and miscellaneous function completion signal wait
D3004	#3004	Effective/ineffective of feed hold, feed override, and G90
D3006	#3006	Message display stop
D3007	#3007	Mirror image
		G command modal
D4001	#4001	Interpolation mode G00:0, G01:1, G02:2, G03:3
D4003	#4002	Plane selection G17:17, G18:18, G19:19
D4004	#4007	Tool radius compensation G40:40, G41:41, G42:42
D4005	#4009	Fixed cycle G80:80, G72-G79:72-79, G81-G89:81-89
D4006	#4012	Workpiece coordinate system G54-G59:54-59
D4007	#4010	Return level G98:0, G99:1
D4008	Error	Subprogram G22:1, G23:0
D4010	#4003	Absolute/incremental G90:1, G91:0
D4011	#4005	Synchronous/asynchronous G94:0, G95:1
D4012	#4006	Inch/metric G70:1, G71:0
D4014	#4008	Tool length compensation G43:1, G44:0

APPENDIX 4. M2/M0 MACRO CONVERTOR
4.3 Detailed description

M2/M0	MELDAS Standard	Function
		Modal other than G codes
D4101	} Error	Second miscellaneous function A
D4102		Second miscellaneous function B
D4103		Second miscellaneous function C
D4104		
D4105	#4107	Tool radius compensation number H, D
D4106	#4109	Feedrate F
D4107	#4111	Tool length offset number H
D4108	#4111	Tool length offset number H
D4109	#4111	Tool length offset number H
D4110	#4111	Tool length offset number H
D4111	#4111	Tool length offset number H
D4112	#4115	Label number L
D4113	#4113	Miscellaneous function M
D4114	#4114	Sequence number N
D4115	#4115	Label number O
D4119	#4119	Spindle function S
D4120	#4120	Tool function T
D4300		H number
<hr style="border-top: 1px dashed black;"/>		
		Position information
		End point coordinates of the immediately preceding block
D5001	#5001	X
D5002	#5002	Y
D5003	#5003	Z
D5004	#5004	4
D5005	#5005	5
		Machine coordinates
D5101	#5021	X
D5102	#5022	Y
D5103	#5023	Z
D5104	#5024	4
D5105	#5025	5
		Workpiece coordinates
D5201	#5041	X
D5202	#5042	Y
D5203	#5043	Z
D5204	#5044	4
D5205	#5045	5
		Tool length measurement
D5301	} Error	X
D5302		Y
D5303		Z
D5304		4
D5305		5
		Start point coordinates of the immediately preceding block
D5401	} Error	X
D5402		Y
D5403		Z
D5404		4
D5405		5

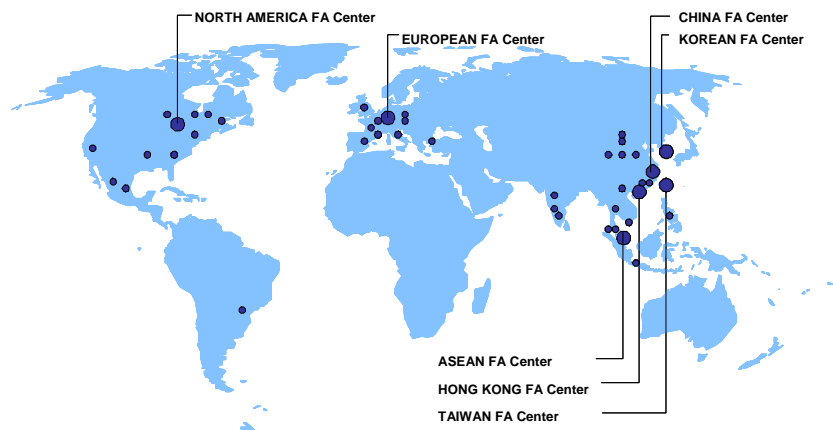
APPENDIX 4. M2/M0 MACRO CONVERTOR
4.3 Detailed description

M2/M0	MELDAS Standard	Function	
		Skip signal input coordinates	
D5501	#5061	X	
D5502	#5062	Y	
D5503	#5063	Z	
D5504	#5064	4	
D5505	#5065	5	
		Interrupt variables	G code modal
D6001	#4401	Interpolation mode	G00:0, G01:1, G02:2, G03:3
D6003	#4402	Plane selection	G17:17, G18:18, G19:19
D6004	#4407	Tool radius compensation	G40:40, G41:41, G42:42
D6005	#4409	Fixed cycle	G80:80, G72-G79:72-79, G81-G89:81-89
D6006	#4412	Workpiece coordinate system	G54-G59:54-59
D6007	#4410	Return level	G98:0, G99:1
D6008	Error	Subprogram	G22:1, G23:0
D6010	#4403	Absolute/incremental	G90:1, G91:0
D6011	#4405	Synchronous/asynchronous	G94:0, G95:1
D6012	#4406	Inch/metric	G70:1, G71:0
D6014	#4408	Tool length offset	G43:1, G44:0
<hr style="border-top: 1px dashed black;"/>			
		Interrupt variables	Modal other than G codes
D6101	} Error	Second miscellaneous function	A
D6102		Second miscellaneous function	B
D6103		Second miscellaneous function	C
D6104	#4511	Tool length, position offset number	H, D
D6105	#4507	Tool radius compensation number	H, D
D6106	#4509	Feedrate	F
D6112	#4515	Label number	L
D6113	#4513	Miscellaneous function	M
D6114	#4514	Sequence number	N
D6115	#4515	Label number	O
D6119	#4519	Spindle function	S
D6120	#4520	Tool function	T

Revision History

Date of revision	Manual No.	Revision details
Oct. 1998		First edition created.
Oct. 1998	BNP-B2240B	<p>The incorrect descriptions are corrected and unnecessary sections are erased. The document number of this manual changed as below.</p> <p style="padding-left: 40px;">Old : BNP B3950-017 New : BNP B2240</p>
Mar. 2004	BNP-B2240D	<p>The details are revised to correspond to the MELDAS 600M Series Version D0. The following explanations and related descriptions are added.</p> <ul style="list-style-type: none"> • 6.9 NURBS interpolation • 7.12 F1-digit feed • 15 Protection function <p>The details are revised to correspond to the MELDAS 600M Series Version F0. The following explanations and related descriptions are added.</p> <ul style="list-style-type: none"> • 6.10 Thread cutting • 13.18 Three-dimensional Coordinate Conversion Function • 13.21 Tapping Mode • 13.27 N Number Single Command Skip <p>Design of the cover and the back cover were changed. MODEL, MODEL CODE, and Manual No. were added on the back cover.</p> <p>Unnecessary sections are erased. Incorrect descriptions are corrected.</p>
Apr. 2006	BNP-B2240E	<p>The description of internal sub-micrometric system was added. Incorrect descriptions were corrected.</p>

Global service network



North America FA Center (MITSUBISHI ELECTRIC AUTOMATION INC.)

Illinois CNC Service Center
500 CORPORATE WOODS PARKWAY, VERNON HILLS, IL. 60061, U.S.A.
TEL: +1-847-478-2500 (Se) FAX: +1-847-478-2650 (Se)

California CNC Service Center
5665 PLAZA DRIVE, CYPRESS, CA. 90630, U.S.A.
TEL: +1-714-220-4796 FAX: +1-714-229-3818

Georgia CNC Service Center
2810 PREMIERE PARKWAY SUITE 400, DULUTH, GA., 30097, U.S.A.
TEL: +1-678-258-4500 FAX: +1-678-258-4519

New Jersey CNC Service Center
200 COTTONTAIL LANE SOMERSET, NJ. 08873, U.S.A.
TEL: +1-732-560-4500 FAX: +1-732-560-4531

Michigan CNC Service Satellite
2545 38TH STREET, ALLEGAN, MI., 49010, U.S.A.
TEL: +1-847-478-2500 FAX: +1-269-673-4092

Ohio CNC Service Satellite
62 W. 500 S., ANDERSON, IN., 46013, U.S.A.
TEL: +1-847-478-2608 FAX: +1-847-478-2690

Texas CNC Service Satellite
1000, NOLEN DRIVE SUITE 200, GRAPEVINE, TX. 76051, U.S.A.
TEL: +1-817-251-7468 FAX: +1-817-416-1439

Canada CNC Service Center
4299 14TH AVENUE MARKHAM, ON. L3R 0J2, CANADA
TEL: +1-905-475-7728 FAX: +1-905-475-7935

Mexico CNC Service Center
MARIANO ESCOBEDO 69 TLALNEPANTLA, 54030 EDO. DE MEXICO
TEL: +52-55-9171-7662 FAX: +52-55-9171-7698

Monterrey CNC Service Satellite
ARGENTINA 3900, FRACC. LAS TORRES, MONTERREY, N.L., 64720, MEXICO
TEL: +52-81-8365-4171 FAX: +52-81-8365-4171

Brazil MITSUBISHI CNC Agent Service Center
(AUTOMOTION IND. COM. IMP. E EXP. LTDA.)
ACESSO JOSE SARTORELLI, KM 2.1 18550-000 BOITUVA – SP, BRAZIL
TEL: +55-15-3363-9900 FAX: +55-15-3363-9911

European FA Center (MITSUBISHI ELECTRIC EUROPE B.V.)

Germany CNC Service Center
GOTHAER STRASSE 8, 40880 RATINGS, GERMANY
TEL: +49-2102-486-0 FAX: +49-2102486-591

South Germany CNC Service Center
KURZE STRASSE. 40, 70794 FILDERSSTADT-BONLANDEN, GERMANY
TEL: +49-711-3270-010 FAX: +49-711-3270-0141

France CNC Service Center
25, BOULEVARD DES BOUVETS, 92741 NANTERRE CEDEX FRANCE
TEL: +33-1-41-02-83-13 FAX: +33-1-49-01-07-25

Lyon CNC Service Satellite

U.K CNC Service Center
TRAVELLERS LANE, HATFIELD, HERTFORDSHIRE, AL10 8XB, U.K.
TEL: +44-1707-282-846 FAX: +44-1707-278-992

Italy CNC Service Center
ZONA INDUSTRIALE VIA ARCHIMEDE 35 20041 AGRATE BRIANZA, MILANO ITALY
TEL: +39-039-60531-342 FAX: +39-039-6053-206

Spain CNC Service Satellite
CTRÀ. DE RUBI, 76-80 -APDO.420 08190 SAINT CUGAT DEL VALLES, BARCELONA SPAIN
TEL: +34-935-65-2236 FAX:

Turkey MITSUBISHI CNC Agent Service Center
(GENEL TEKNİK SİSTEMLER LTD. STI.)
DARULACEZE CAD. FÂMAS İS MERKEZİ A BLOK NO.43 KAT2 80270 OKMEYDANI İSTANBUL,
TURKEY
TEL: +90-212-320-1640 FAX: +90-212-320-1649

Poland MITSUBISHI CNC Agent Service Center (MPL Technology Sp. z o. o)
UL SLICZNA 34, 31-444 KRAKOW, POLAND
TEL: +48-12-632-28-85 FAX:

Wroclaw MITSUBISHI CNC Agent Service Satellite (MPL Technology Sp. z o. o)
UL KOBIERZYCKA 23, 52-315 WROCLAW, POLAND
TEL: +48-71-333-77-53 FAX: +48-71-333-77-53

Czech MITSUBISHI CNC Agent Service Center
(AUTOCONT CONTROL SYSTEM S.R.O.)
NEMOCNICNI 12, 702 00 OSTRAVA 2 CZECH REPUBLIC
TEL: +420-596-152-426 FAX: +420-596-152-112

ASEAN FA Center (MITSUBISHI ELECTRIC ASIA PTE. LTD.)

Singapore CNC Service Center
307 ALEXANDRA ROAD #05-01/02 MITSUBISHI ELECTRIC BUILDING SINGAPORE 159943
TEL: +65-6473-2308 FAX: +65-6476-7439

Thailand MITSUBISHI CNC Agent Service Center (F. A. TECH CO., LTD)
888/19,20,21,22 S.V. CITY BUILDING OFFICE TOWER 1 FLOOR 12,14 RAMA III RD BANGPONGPANG,
YANNAWA, BANGKOK 10120. THAILAND
TEL: +66-2-682-6522 FAX: +66-2-682-6020

Malaysia MITSUBISHI CNC Agent Service Center
(FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
60, JALAN USJ 10/1B 47620 UEP SUBANG JAYA SELANGOR DARUL EHSAN MALAYSIA
TEL: +60-3-5631-7605 FAX: +60-3-5631-7636

JOHOR MITSUBISHI CNC Agent Service Satellite
(FLEXIBLE AUTOMATION SYSTEM SDN. BHD.)
NO. 16, JALAN SHAHBANDAR 1, TAMAN UNGKU TUN AMINAH, 81300 SKUDAI, JOHOR MALAYSIA
TEL: +60-7-557-8218 FAX: +60-7-557-3404

Indonesia MITSUBISHI CNC Agent Service Center
(PT. AUTOTEKNIINDO SUMBER MAKMUR)
WISMA NUSANTARA 14TH FLOOR JL. M.H. THAMRIN 59, JAKARTA 10350 INDONESIA
TEL: +62-21-3917-144 FAX: +62-21-3917-164

India MITSUBISHI CNC Agent Service Center (MESSUNG SALES & SERVICES PVT. LTD.)
B-36FF, PAVANA INDUSTRIAL PREMISES M.I.D.C., BHOASRI PUNE 411026, INDIA
TEL: +91-20-2711-9484 FAX: +91-20-2712-8115

BANGALORE MITSUBISHI CNC Agent Service Satellite
(MESSUNG SALES & SERVICES PVT. LTD.)
S 615, 6TH FLOOR, MANIPAL CENTER, BANGALORE 560001, INDIA
TEL: +91-80-509-2119 FAX: +91-80-532-0480

Delhi MITSUBISHI CNC Agent Parts Center (MESSUNG SALES & SERVICES PVT. LTD.)
1197, SECTOR 15 PART-2, OFF DELHI-JAIPUR HIGHWAY BEHIND 32ND MILESTONE GURGAON
122001, INDIA
TEL: +91-98-1024-8895 FAX:

Philippines MITSUBISHI CNC Agent Service Center
(FLEXIBLE AUTOMATION SYSTEM CORPORATION)
UNIT No.411, ALABAMG CORPORATE CENTER KM 25. WEST SERVICE ROAD SOUTH SUPERHIGHWAY,
ALABAMG MUNTINLUPA METRO MANILA, PHILIPPINES 1771
TEL: +63-2-807-2416 FAX: +63-2-807-2417

Vietnam MITSUBISHI CNC Agent Service Center (SA GIANG TECHNO CO., LTD)
47-49 HOANG SA ST. DAKAO WARD, DIST.1 HO CHI MINH CITY, VIETNAM
TEL: +84-8-910-4763 FAX: +84-8-910-2593

China FA Center (MITSUBISHI ELECTRIC AUTOMATION (SHANGHAI) LTD.)

China CNC Service Center
2/F., BLOCK 5 BLDG.AUTOMATION INSTRUMENTATION PLAZA, 103 CAOBAO RD. SHANGHAI 200233,
CHINA
TEL: +86-21-6120-0808 FAX: +86-21-6494-0178

Shenyang CNC Service Center
TEL: +86-24-2397-0184 FAX: +86-24-2397-0185

Beijing CNC Service Satellite
9/F, OFFICE TOWER1, HENDERSON CENTER, 18 JIANGUOMENNEI DAJIE, DONGCHENG DISTRICT,
BEIJING 100005, CHINA
TEL: +86-10-6518-8830 FAX: +86-10-6518-8030

China MITSUBISHI CNC Agent Service Center
(BEIJING JIAYOU HIGHTECH TECHNOLOGY DEVELOPMENT CO.)
RM 709, HIGH TECHNOLOGY BUILDING NO.229 NORTH SI HUAN ZHONG ROAD, HAIDIAN DISTRICT,
BEIJING 100083, CHINA
TEL: +86-10-8288-3030 FAX: +86-10-6518-8030

Tianjin CNC Service Satellite
RM909, TAIHONG TOWER, NO220 SHIZILIN STREET, HEBEI DISTRICT, TIANJIN, CHINA 300143
TEL: +86-22-2653-9090 FAX: +86-22-2635-9050

Shenzhen CNC Service Satellite
RM02, UNIT A, 13/F, TIANAN NATIONAL TOWER, RENMING SOUTH ROAD, SHENZHEN, CHINA 518005
TEL: +86-755-2515-6691 FAX: +86-755-8218-4776

Changehun Service Satellite
TEL: +86-431-50214546 FAX: +86-431-5021690

Hong Kong CNC Service Center
UNIT A, 25/F RYODEN INDUSTRIAL CENTRE, 26-38 TA CHUEN PING STREET, KWAI CHUNG, NEW
TERRITORIES, HONG KONG
TEL: +852-2619-8588 FAX: +852-2784-1323

Taiwan FA Center (MITSUBISHI ELECTRIC TAIWAN CO., LTD.)

Taichung CNC Service Center
NO.8-1, GONG YEH 16TH RD., TAICHUNG INDUSTRIAL PARK TAICHUNG CITY, TAIWAN R.O.C.
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Taipei CNC Service Satellite
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Tainan CNC Service Satellite
TEL: +886-4-2359-0688 FAX: +886-4-2359-0689

Korean FA Center (MITSUBISHI ELECTRIC AUTOMATION KOREA CO., LTD.)

Korea CNC Service Center
DONGSEO GAME CHANNEL BLDG. 2F. 660-11, DEUNGCHON-DONG KANGSEO-KU SEOUL, 157-030
KOREA
TEL: +82-2-3660-9607 FAX: +82-2-3663-0475

Notice

Every effort has been made to keep up with software and hardware revisions in the contents described in this manual. However, please understand that in some unavoidable cases simultaneous revision is not possible.

Please contact your Mitsubishi Electric dealer with any questions or comments regarding the use of this product.

Duplication Prohibited

This instruction manual may not be reproduced in any form, in part or in whole, without written permission from Mitsubishi Electric Corporation.

© 1998-2006 MITSUBISHI ELECTRIC CORPORATION
ALL RIGHTS RESERVED.



MODEL	M600M Series
MODEL CODE	008—112
Manual No.	BNP-B2240E(ENG)

Specifications subject to change without notice.
Printed in Japan on recycled paper.