FANUC MANUAL GUIDE 0*i*

OPERATOR'S MANUAL

B-64434EN/02

The products in this manual are controlled based on Japan's "Foreign Exchange and Foreign Trade Law". The export from Japan may be subject to an export license by the government of Japan.
Further, re-export to another country may be subject to the license of the government of the country from where the product is re-exported. Furthermore, the product may also be controlled by re-export regulations of the United States government.
Should you wish to export or re-export these products, please contact FANUC for advice.

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

SAFETY PRECAUTIONS

When using a machine equipped with the FANUC MANUAL GUIDE 0*i*, be sure to observe the following safety precautions.

DEFINITION OF WARNING, CAUTION, AND NOTE

This manual includes safety precautions protecting the user and preventing damage to the machine. Precautions are classified into Warning and Caution according to the degree of the risk or the severity of damage.
Also, supplementary information is described as Note.
Read the Warning, Caution, and Note thoroughly before attempting to use the machine.

⚠ WARNING
Applied when there is a danger to the user being injured or when there is a risk to the user, being injured, and the equipment, being damaged, if the warning statement is not followed up.

⚠ CAUTION
Applied when there is a danger to the equipment being damaged, if the caution statement is not followed up.

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

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

GENERAL WARNINGS AND CAUTIONS

To ensure safety while using a machine featuring the MANUAL GUIDE 0*i* function, observe the following precautions:

⚠ WARNING
1 Confirm, on the screen, that the data has been entered correctly before proceeding to the next operation. Attempting operation with incorrect data may cause the tool to strike the workpiece or machine, possibly breaking the tool and/or machine and/or injuring the operator.
2 When using constant surface speed control in the lathe, set the data item of the maximum spindle speed to the value allowed for the workpiece and workpiece holding unit. Otherwise, the workpiece or holding unit may be removed by centrifugal force damaging the machine or injuring the operator.
3 Set all necessary parameters and data items before starting MANUAL GUIDE 0*i* operations. Note that if the cutting conditions are not suitable for the workpiece, the tool may be damaged and/or the operator may be injured.

⚠ WARNING

4 After creating a machining program using MANUAL GUIDE 0*i* functions, do no run the machining program immediately on the machine . Before starting production machining, run the machine with no workpiece attached to the machine to make sure that the tool will not strike a workpiece or the machine. If the tool strikes the machine and/or workpiece, the tool and/or machine may be damaged, with possibility to injure the operator.

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

⚠ CAUTION

After pressing the power-on button, do not touch any keys on the keyboard until the initial screen appears. Some keys are used for maintenance or special operations such that pressing such a key may cause an unexpected operation.

OVERVIEW OF THIS MANUAL

This manual describes the functions of "MANUAL GUIDE 0*i*" for the Series 0*i* -TD/MD or Series 0*i* Mate-TD/MD.

For other functions, other than MANUAL GUIDE 0*i*, refer to the User's Manual for the Series 0*i* -TD/MD or Series 0*i* Mate-TD/MD.

In this manual, the description about the Series 0*i* Mate-MD or Series 0*i*-MD is marked with "Machining Center System" and the description about Series 0*i* Mate-TD or Series 0*i*-TD is marked with "Lathe System".

The specifications and features of MANUAL GUIDE 0*i* may differ from the specifications of the operator's manual supplied by the machine tool builder. Be sure to read the manual provided by the machine tool builder.

The functions of the CNC machine tool system are determined not only by the CNC, but by the combination of the machine tool, the power magnetic circuit of the machine tool, the servo system, the CNC, and the operator's panel.

It is impossible to cover all possible combinations of all functions, programming methods, and operations in a single manual.

This manual explains only the MANUAL GUIDE 0*i* operations provided for the CNC. For individual CNC machine tools, refer to applicable manuals supplied by the machine tool builders.

This manual explains as many detailed functions as possible. However, it is not possible to describe all the items which cannot be done or which the operator must not do. Therefore, please assume that functions not described in this manual cannot be performed.

Detailed information and special conditions are explained in notes. The readers may encounter new technical terms in the notes not previously defined or described. In this case, read this manual through first, and then review the details.

TABLE OF CONTENTS

V. CONTOUR PROGRAMMING

APPENDIX

I. WHAT'S MANUAL GUIDE 0*i* ?

1 WHAT IS MANUAL GUIDE 0*i*?

MANUAL GUIDE 0*i* is the programming operation guidance function that specializes in creation of a machining program and pursues simple operation.
The screen of MANUAL GUIDE 0*i* is displayed when soft key [C.A.P.] on the NC program edit screen is pressed.

MANUAL GUIDE 0*i* has the following features.

1. ISO code part programming

ISO code part programming can be used to directly input simple operation such as linear or arc machining with G codes or easily input complicated operation such as pocketing or turning with a machining cycle.

2. Easy specification of a tool, spindle, coolant, and feedrate

These conditions can easily be specified on a screen before machining.

3. Easy programming of G codes and M codes

G codes and M codes can easily be input with reference to the screen for G code description and illustration and the screen for M code description and illustration.

4. Preparation of machining cycles often used for machining centers and lathes

The following machining cycles are provided. A program of machining cycles can be created only if the necessary data is input according to the guide maps and messages displayed on the screen.

< Machining Cycle for Machining Center System >

(1) Hole machining (pattern figure)
(2) Pocketing (pattern figure)
(3) Facing (pattern figure)
(4) Grooving (pattern figure)

< Machining Cycle for Lathe System >

(1) Drilling (workpiece center)
(2) Turning (arbitrary figure)
(3) Grooving (normal groove, trapezoidal groove)
(4) Threading (general-purpose thread, metric thread, unified thread, PT/PF thread)

5. Easy input of contour figure

A contour figure consisting of lines and arcs can easily be input and be converted into an NC command block (G01/G02/G03). In addition, advance figure calculation functions including 11 auxiliary calculation functions can be performed.

NOTE
1 Soft key [C.A.P.] also appears when conversational programming with graphic is enabled. When both MANUAL GUIDE 0*i* and conversational programming with graphic are enabled, MANUAL GUIDE 0*i* takes precedence.
2 MANUAL GUIDE 0*i* is disabled in a mode other than the EDIT mode.
3 MANUAL GUIDE 0*i* is disabled in BG editing.
4 MANUAL GUIDE 0*i* operates only in path 1 and is not displayed in path 2. In path 2, the machining cycle of MANUAL GUIDE 0*i* is not performed.
5 MANUAL GUIDE 0*i* supports both the 8.4-inch and 10.4-inch display units. The screen configuration is the same in both display units.

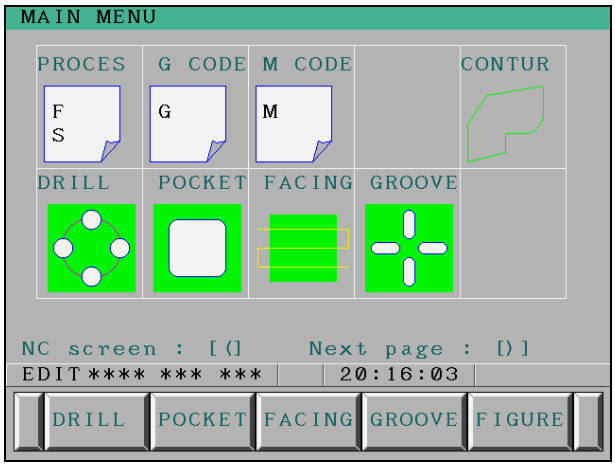
1.1 CONDITIONS FOR USING MANUAL GUIDE 0*i*

Increment system of only IS-B is available.

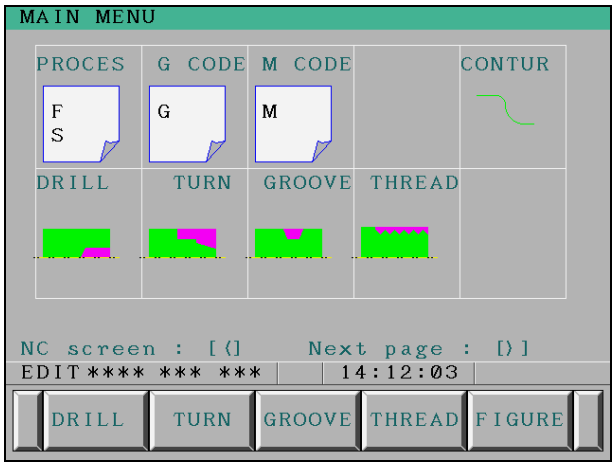
II. BASIC SCREEN AND OPERATIONS

1 MAIN MENU SCREEN

When soft key [C.A.P.] on the NC program edit screen is pressed in the EDIT mode, MANUAL GUIDE 0*i* is started and the main menu screen below appears.
A machining program can easily be created if menu selection is made according to the guide on the screen.



Main menu screen (Machining Center System)



Main menu screen (Lathe System)

On the main menu screen, the menus of the following functions can be selected.

(1) Process support function
→ Selected with soft key [PROCES] (next page).
(2) G code support function
→ Selected with soft key [G CODE] (next page).
(3) M code support function
→ Selected with soft key [M CODE] (next page).
(4) Machining cycle function
→ Selected with soft key [DRILL], [POCKET], [FACING], or [GROOVE]
→ Selected with soft key [DRILL], [TURN], [GROOVE], or [THREAD]
(5) Contour programming function
→ Selected with soft key [CONTUR] (next page).

When the leftmost soft key is pressed, the NC program edit screen is displayed again.

The above functions are described below.

NOTE
1 MANUAL GUIDE 0*i* requires a work program that stores a CNC program to be created temporarily. Set parameter No. 9358 to the number of this work program.
(If this parameter is not set, the screen of MANUAL GUIDE 0*i* does not appear.)

NOTE

2 The program to be displayed by soft key [C.A.P.] depends on the cursor position on the NC program edit screen.
- When the cursor positions in the block containing a 4-digit code for machining cycle, the machining cycle data input screen is displayed.
- When the cursor positions in a block not containing a 4-digit code for machining cycle, the main menu screen is displayed.

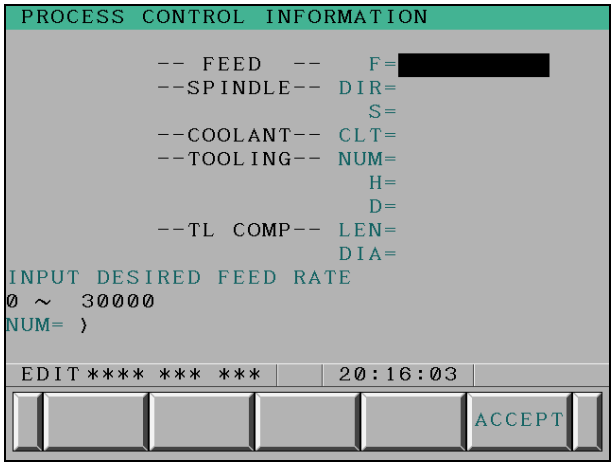
3 When soft key [C.A.P.] is pressed if there is no CNC program, a warning saying "INITIALIZATION OF MG0I CANNOT BE COMPLETED." is displayed. Be sure to create one or more CNC programs before pressing soft key [C.A.P.].

2 PROCESS SUPPORT FUNCTION

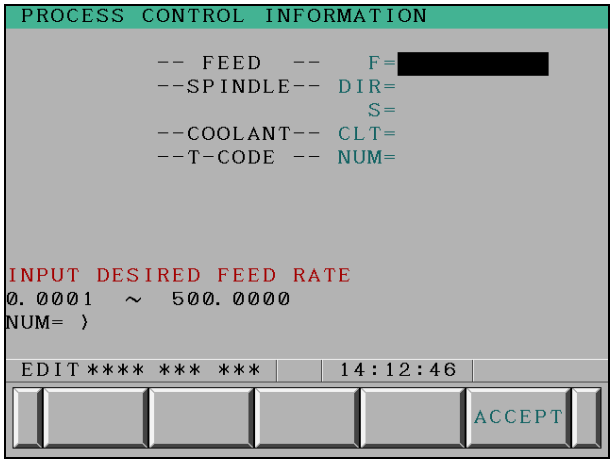
Chapter 2, "PROCESS SUPPORT FUNCTION" consists of the following sections:

2.1 SCREENS AND OPERATIONS

On the process support screen, it is possible to enter information about the conditions required for machining.
When soft key [PROCES] on the next page is pressed on the main menu screen, the process support screen below is displayed.



Process support screen (Machining Center System)



Process support screen (Lathe System)

Follow the steps below on the process support screen.

(1) Move the cursor to the desired item with the MDI cursor key.
(2) When the input item is numeric data, enter the data with MDI keys and press the [INPUT] key.
(3) When the input item is character data, make selection from displayed data with the cursor keys on the left and right sides of the MDI key panel.
(4) To add the entered data to the machining program, press soft key [ACCEPT]. The entered data is added to the machining program and the main menu screen is displayed again.
(5) To cancel the entered data, press the leftmost key to return to the main menu screen.

2.2 INPUT ITEMS

The input items on the process support screen are described in detail below.

(1) Input items for Machining Center System

Item name	Address	Code to be output	Description	Input range/alternatives
FEED	F	F	Feedrate	0.0 to 30000.0

Item name	Address	Code to be output	Description	Input range/alternatives
SPINDLE	DIR	M3/4/5	Rotation direction	CW CCW OFF
	S	S	Spindle speed	0 to 6000
COOLANT	CLT	M7/8/9	Selection of a coolant	FLOOD MIST OFF
TOOLING	NUM	T	Tool number	0 to 9999
	H	G49 H	Tool offset number	0 to 400
	D	G40 D	Cutter compensation number	0 to 400
TL COMP	LEN	G43/44/49	Selection of a tool compensation length	LEN PLUS LEN MINUS OFF
	DIA	G40/41/42	Selection of a tool compensation direction	COMP LEFT COMP RIGHT OFF
The following item is added when bit 2 of parameter No. 9300 is 1.				
Z POINT	Z	Z	Z-coordinate (absolute)	-99999.999 to 99999.999

(2) Input items for Lathe System

Item name	Address	Code to be output	Description	Input range/alternatives
FEED	F	F	Feedrate	0.0001 to 500.0000
SPINDLE	DIR	M3/4/5	Selection of a rotation direction	CW CCW OFF
	S	S	Spindle speed	0 to 6000
COOLANT	CLT	M7/8/9	Selection of a coolant	FLOOD MIST OFF
T-CODE	NUM	T	T code number	0 to 9999
The following items are added when bit 5 of parameter No. 9300 is 1.				
CSS/RPM	None	G96/97	Selection of surface speed/spindle speed	RPM CSS
MAX RPM	None	G50 S	Maximum spindle speed	0 to 60000
CYCLE ST POS X	X	G0 X. Y	Start X coordinate	-99999.999 to 99999.999
CYCLE ST POS Z	Y		Start Y coordinate	-99999.999 to 99999.999

3 G CODE SUPPORT FUNCTION

Chapter 3, "G SUPPORT FUNCTION" consists of the following sections:

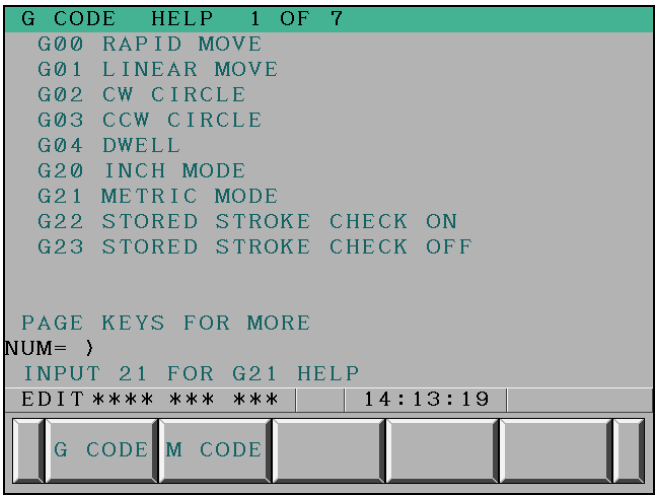
3.1 SCREENS AND OPERATIONS

On the G code support screen, it is possible to enter the desired G code with reference to the description and illustration of the G code.

The G code support screen includes the following screens.
(1) G code help screen
(2) G code text screen
(3) G code graph screen

3.1.1 G Code Help Screen

When soft key [G CODE] is pressed on the main menu screen, the G code help screen below is displayed. On the G code help screen, the supported G codes are listed. This menu is spread over several pages. The total number of help pages and the current page number are displayed at the top of the menu.

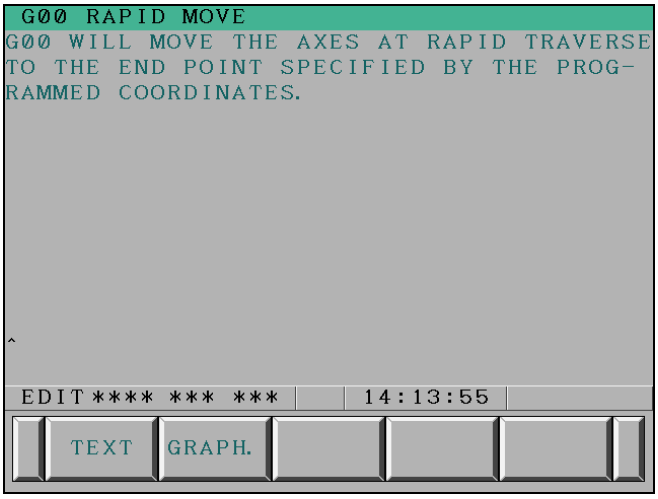


Follow the steps below on the G code help screen.

(1) Use the MDI page key to switch to a list including the desired G code. The cursor page key on the MDI key panel can be used to control the display of the G code help page. The menu screen is of cyclic type.
(2) When the desired G code number is input and the [INPUT] key is pressed, the G code text screen of the desired G code is displayed.
(3) When soft key [M CODE] is pressed, the M code help screen is displayed.
(4) When the leftmost soft key is pressed the main menu screen is displayed again.

3.1.2 G Code Text Screen

This screen describes the G code specified on the G code help screen using text.
In addition, a G code block can be added to the machining program on this screen.



G code text screen of G00

Follow the steps below on the G code text screen.

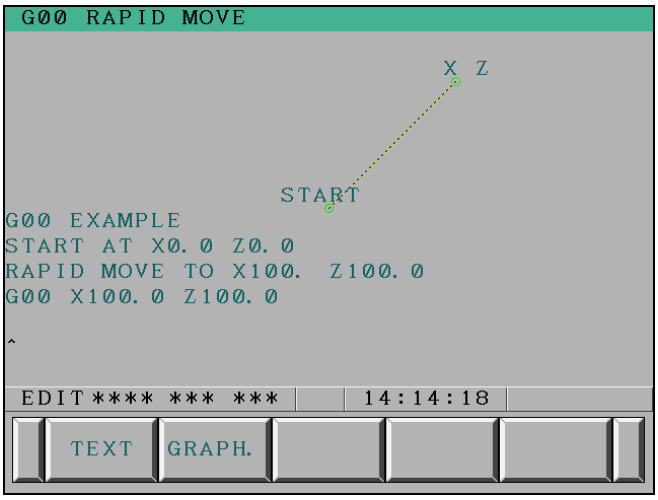
(1) For addition to the machining program
The entered NC statement block can be output to the machining program.
  (a) Press the MDI key to input an NC statement block.
  (b) Press the [INSERT] key.
  (c) The entered NC statement block is output to the machining program and the main menu is displayed again.

(2) For a return to the G code help screen
When the leftmost soft key is pressed, the G code help screen is displayed again.

(3) For switching to the G code graph screen
When soft key [GRAPH.] is pressed, the G code graph screen of the displayed G code is displayed again.

3.1.3 G Code Graph Screen

This screen displays information about the G code specified on the G code help screen using illustration and a sample program.
On this screen, a G code block can be generated and inserted into the machining program.



G code graph screen of G00

NOTE
Some G codes have no graphical help screen. In this case, no graphical help screen appears even if [GRAPH.] is pressed.

Follow the steps below on the G code graph screen.

(1) For addition to the machining program
The input NC statement block can be output to the machining program.
  (a) Press the MDI key to input an NC statement block.
  (b) Press the [INSERT] key.
  (c) The input NC statement block is output to the machining program and the main menu is displayed again.

(2) For a return to the G code help screen
When the leftmost soft key is pressed, the G code help screen is displayed again.

(3) For switching to the G code graph screen
When soft key [GRAPH.] is pressed, the G code text screen of the displayed G code is displayed again.

3.2 SUPPORTED G CODES

The G codes supported by the G code support function are listed below.

(1) G codes for Machining Center System

G code	Description	Text screen	Graph screen
G00	Positioning	○	○
G01	Linear interpolation	○	○
G02	Circular interpolation CW	○	○
G03	Circular interpolation CCW	○	○
G04	Dwell	○	○
G08	Advanced preview control	○	−
G09	Exact stop	○	−

G code	Description	Text screen	Graph screen
G17	XY plane selection	○	○
G18	ZX plane selection	○	○
G19	YZ plane selection	○	○
G20	Input in inch	○	○
G21	Input in mm	○	○
G22	Stored stroke check function on	○	–
G23	Stored stroke check function off	○	–
G25	Spindle speed fluctuation detection off	○	–
G26	Spindle speed fluctuation detection on	○	–
G27	Reference position return check	○	–
G28	Automatic return to reference position	○	○
G29	Movement from reference position	○	○
G31	Skip function	○	○
G40	Cutter compensation : cancel	○	–
G41	Cutter compensation : left	○	○
G42	Cutter compensation : right	○	○
G43	Tool length compensation +	○	○
G44	Tool length compensation -	○	○
G45	Tool offset : increase	○	–
G46	Tool offset : decrease	○	–
G47	Tool offset : double increase	○	–
G48	Tool offset : double decrease	○	–
G49	Tool length compensation cancel	○	–
G52	Local coordinate system setting	○	○
G53	Machine coordinate system setting	○	–
G60	Single direction positioning	○	○
G61	Exact stop mode	○	–
G63	Tapping mode	○	–
G64	Cutting mode	○	–

(2) G codes for Lathe System

G code	Description	Text screen	Graph screen
G00	Positioning	○	○
G01	Linear interpolation	○	○
G02	Circular interpolation CW	○	○
G03	Circular interpolation CCW	○	○
G04	Dwell	○	○
G20	Input in inch	○	–
G21	Input in mm	○	–
G22	Stored stroke check function on	○	○
G23	Stored stroke check function off	○	○
G25	Spindle speed fluctuation detection off	○	–
G26	Spindle speed fluctuation detection on	○	–
G27	Reference position return check	○	–
G28	Automatic return to reference position	○	○
G31	Skip function	○	○
G32	Threading	○	○
G34	Variable lead threading	○	–
G40	Tool nose radius compensation : cancel	○	–
G41	Tool nose radius compensation : left	○	○
G42	Tool nose radius compensation : right	○	○
G50	Coordinate system setting	○	○
G52	Local coordinate system setting	○	○

G code	Description	Text screen	Graph screen
G53	Machine coordinate system setting	○	–
G54	Workpiece coordinate system 1 selection	○	–
G55	Workpiece coordinate system 2 selection	○	–
G56	Workpiece coordinate system 3 selection	○	–
G57	Workpiece coordinate system 4 selection	○	–
G58	Workpiece coordinate system 5 selection	○	–
G59	Workpiece coordinate system 6 selection	○	–
G65	Macro call	○	–
G66	Macro modal call	○	–
G67	Macro modal call cancel	○	–
G70	Finishing cycle	○	–
G71	Stock removal in turning	○	○
G72	Stock removal in facing	○	○
G73	Pattern repeating cycle	○	○
G74	End face peck drilling cycle	○	○
G75	Outer diameter/internal diameter drilling cycle	○	–
G76	Multiple-thread cutting cycle	○	○
G80	Canned cycle cancel for drilling	○	–
G83	G87 Cycle for face/side drilling	○	○
G84	Cycle for face/side tapping	○	○
G85	Cycle for face/side boring	○	○
G90	Outer diameter/internal diameter cutting cycle	○	○
G92	Threading cycle	○	○
G94	End face turning cycle	○	○
G96	Constant surface speed control	○	–
G97	Constant surface speed control cancel	○	–
G98	Feed per minute	○	–
G99	Feed per revolution	○	–
G65	Macro call	○	–
G66	Macro modal call	○	–
G67	Macro modal call cancel	○	–
G73	Peck drilling cycle	○	○
G74	Reverse tapping cycle	○	○
G76	Fine boring cycle	○	○
G80	Canned cycle cancel	○	–
G81	Drilling cycle	○	○
G82	Drilling cycle	○	○
G83	Peck drilling cycle	○	○
G84	Tapping cycle	○	○
G85	Boring cycle	○	○
G86	Boring cycle	○	○
G87	Back boring cycle	○	○
G88	Boring cycle	○	○
G89	Boring cycle	○	○
G90	Absolute programming	○	○
G91	Incremental programming	○	○
G92	Workpiece coordinate system setting	○	○
G94	Feed per minute	○	–
G95	Feed per revolution	○	–
G98	Canned cycle : return to initial level	○	○
G99	Canned cycle : return to R point level	○	○

4 M CODE SUPPORT FUNCTION

Chapter 4, "M CODE SUPPORT FUNCTION" consists of the following sections:

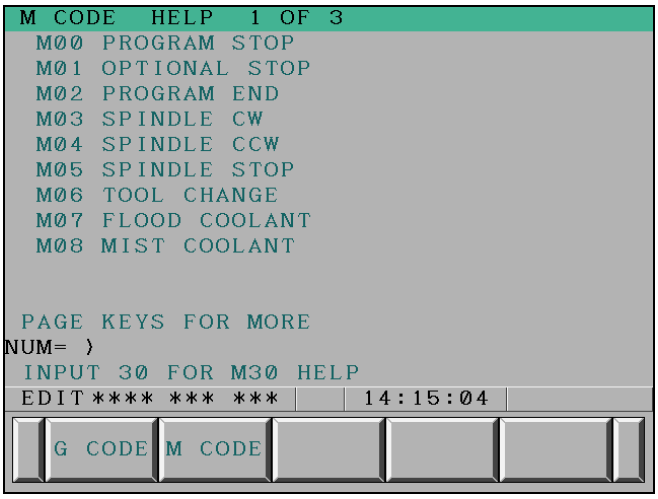
4.1 SCREENS AND OPERATIONS

On the M code support screen, it is possible to enter the desired M code with reference to the description and illustration of the M code.

The M code support screen includes the following screens.
(1) M code help screen
(2) M code text screen

4.1.1 M Code Help Screen

When soft key [M CODE] is pressed on the main menu screen, the M code help screen below is displayed.
On the M code help screen, the supported M codes are listed. This menu is spread over several pages. The total number of help pages and the current page number are displayed at the top of the menu.

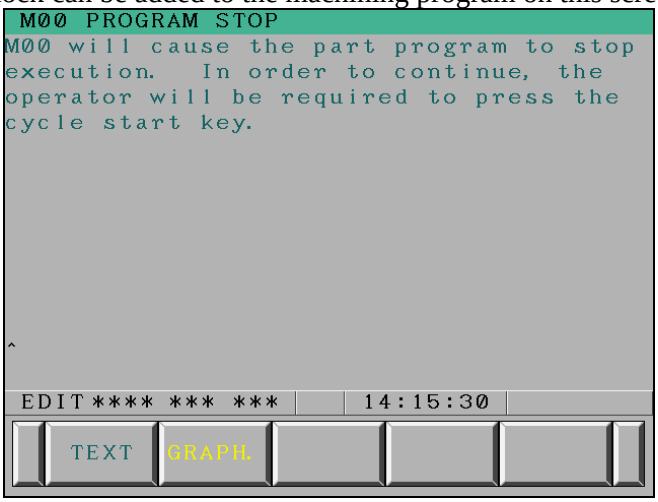


Follow the steps below on the M code help screen.

(1) Use the MDI page key to switch to a list including the desired M code. The cursor page key on the MDI key panel can be used to control the display of the M code help page. The menu screen is of cyclic type.
(2) When the desired M code number is input and the [INPUT] key is pressed, the M code text screen of the desired M code is displayed.
(3) When soft key [G CODE] is pressed, the G code help screen is displayed.
(4) When the leftmost soft key is pressed the main menu screen is displayed again.

4.1.2 M Code Text Screen

This screen describes the M code specified on the M code help screen using text.
In addition, an M code block can be added to the machining program on this screen.



M code text screen of M00

Follow the steps below on the M code text screen.

(1) For addition to the machining program
The entered NC statement block can be output to the machining program.
(a) Press the MDI key to input an NC statement block.
(b) Press the [INSERT] key.
(c) The entered NC statement block is output to the machining program and the main menu is displayed again.

(2) For a return to the M code help screen
When the leftmost soft key is pressed, the M code help screen is displayed again.

4.2 SUPPORTED M CODES

The M codes supported by the M code support function are listed below.

M code	Description	Text screen	Graph screen
M00	Program stop	○	—
M01	Optional stop	○	—
M02	End of program	○	—
M03	Forward spindle rotation	○	—
M04	Reverse spindle rotation	○	—
M05	Spindle stop	○	—
M06	Tool change	○	—
M07	Fluid coolant	○	—
M08	Mist coolant	○	—
M09	Coolant off	○	—
M19	Spindle orientation	○	—
M30	Program stop and rewind	○	—
M48	Spindle speed override enabled	○	—
M49	Spindle speed override 100%	○	—
M67	Motor 1 start	○	—
M68	Motor 2 start	○	—
M69	Motor 3 start	○	—

M code	Description	Text screen	Graph screen
M77	Motor 1 stop	○	—
M78	Motor 2 stop	○	—
M79	Motor 3 stop	○	—
M98	Calling of subprogram	○	—
M99	End of subprogram	○	—

5 MACHINING CYCLE FUNCTION

MANUAL GUIDE 0*i* provides the following machining cycles, which can be used to easily program complicated operation such as pocketing or turning.

< Machining Cycle for Machining Center System >

(1) Hole machining (pattern figure)
(2) Pocketing (pattern figure)
(3) Facing (pattern figure)
(4) Grooving (pattern figure)

< Machining Cycle for Lathe System >

(1) Drilling (workpiece center)
(2) Turning (arbitrary figure)
(3) Grooving (normal groove, trapezoidal groove)
(4) Threading (general-purpose thread, metric thread, unified thread, PT/PF thread)

For details, see Part III, "MILLING CYCLE (MACHINING CENTER SYSTEM)" and Part IV, "TURNING CYCLE (LATHE SYSTEM)".

6 CONTOUR PROGRAMMING FUNCTION

MANUAL GUIDE 0*i* has the contour programming function, which can be used to input a contour figure consisting of lines and arcs. This contour programming includes high performance contour calculation functions such as 30 blocks of pending and 11 patterns of auxiliary calculation.
The operator can input a contour figure consisting of lines and arcs by using contour programming. The contour figure can be converted into a standard ISO code program (such as G01/G02/G03) that traces the contour figure or the figure data block of a turning cycle (such as G1400/G1401/G1402/G1403/G1406).
The type of a program created by operation can also be selected.
During input of a contour figure, auxiliary calculation other than intersection calculation can be used to obtain the end coordinates of the figure.
For details, see Part V, "CONTOUR PROGRAMMING".

NOTE
1 A program created by contour programming of MANUAL GUIDE 0*i* cannot be edited on the contour programming screen.
2 A maximum of 30 figures can be input by contour programming.

III. MILLING CYCLE
(MACHINING CENTER SYSTEM)

1 MACHINING TYPE

MANUAL GUIDE 0*i* provides the following machining cycles.

Hole machining			
	Machining type block	G1000	Center drilling
		G1001	Drilling
		G1002	Tapping
		G1003	Reaming
		G1004	Boring
		G1005	Fine boring
		G1006	Back boring
	Figure block	G1210	Hole pattern - Points
		G1211	Hole pattern - Line (Equal interval)
		G1212	Hole pattern - Line (Not Equal interval)
		G1213	Hole pattern - Grid
		G1214	Hole pattern - Square
		G1215	Hole pattern - Circle
		G1216	Hole pattern - Arc (Equal interval)
		G1217	Hole pattern - Arc (Not Equal interval)
Face machining			
	Machining type block	G1020	Facing - Rough
		G1021	Facing - Finish
	Figure block	G1220	Fixed figure - Square
		G1221	Fixed figure - Circle
Pocket machining			
	Machining type block	G1040	Pocketing - Rough
		G1041	Pocketing - Bottom finish
		G1042	Pocketing - Side finish
		G1043	Pocketing - Chamfer
		G1045	Pocketing - Drilling
	Figure block	G1220	Fixed figure - Square
		G1221	Fixed figure - Circle
		G1222	Fixed figure - Track
Groove machining			
	Machining type block	G1050	Grooving - Rough
		G1051	Grooving - Bottom finish
		G1052	Grooving - Side finish
		G1053	Grooving - Chamfer
		G1055	Grooving - Drilling
	Figure block	G1223	Fixed figure - Radial line grooves

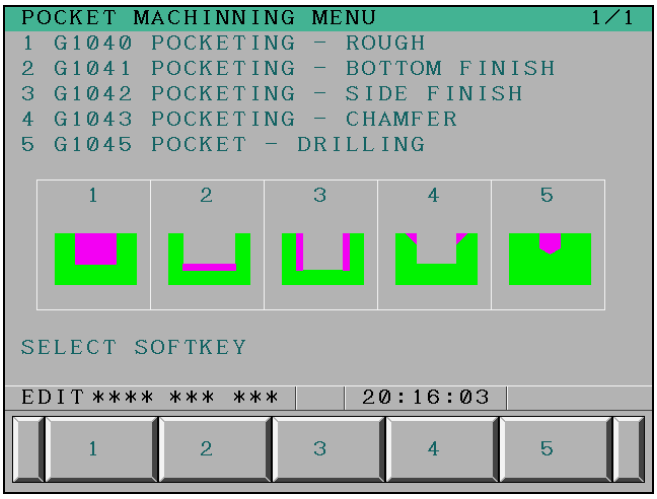
2 OPERATING METHOD

Chapter 2, "OPERATING METHOD", consists of the following sections:

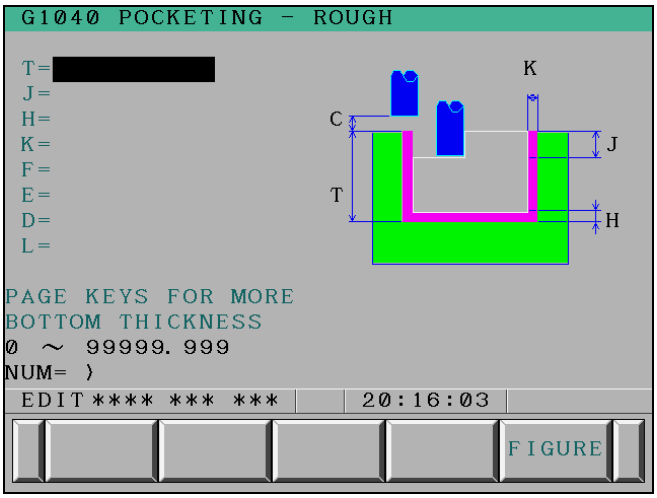
2.1 NEWLY CREATING A MACHINING CYCLE

The method for newly creating a machining cycle is described below.

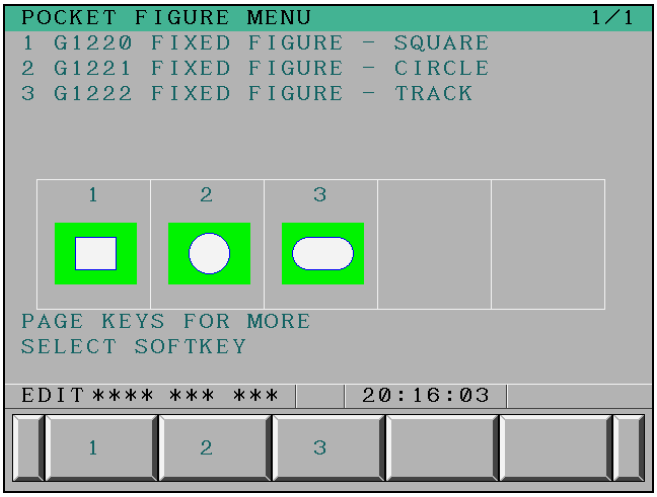
(1) On the main menu screen, select the machining type (hole machining, pocketing, facing, or grooving) of the desired machining cycle with the corresponding soft key. When, for example, pocketing is selected, the machining menu screen shown below appears.



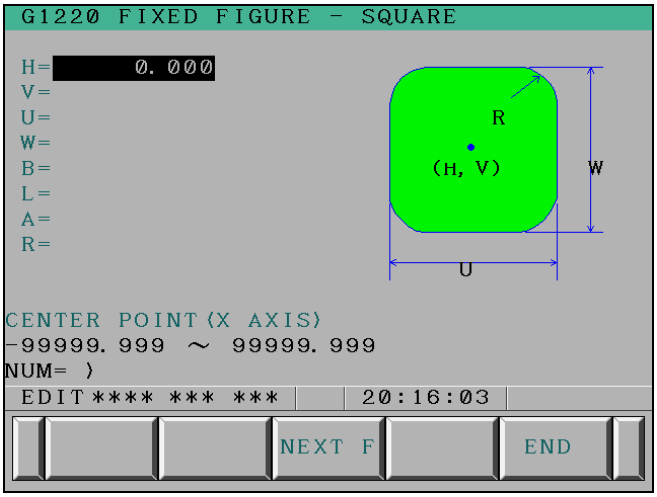
(2) Select the desired machining type with the corresponding soft key. The data input screen of the selected machining type block (when rough machining is selected) as shown below appears. For details on the input items of each machining type block, see the next section. To cancel insertion, press the leftmost soft key.



(3) When the necessary data is entered and soft key [FIGURE] is pressed, a figure menu screen as shown below appears. Only the menu of figures that can be used in the machining type block entered immediately before is displayed. The machining type block entered here is added to the machining program.



(4) When a figure is selected with the corresponding soft key, an data input screen as shown below appears (SQUARE is selected in the following figure). For details on the input item of each figure block, see the next section.



(5) When the necessary data has been entered, press soft key [NEXT F] or [END]. When [END] is pressed, the entered figure block is added to the machining program and the main menu is displayed.
To enter a figure continuously, press [NEXT F]. Then, the entered figure block is added to the machining program and the figure menu screen is displayed again.
To enter a machining type block and figure block and then add a figure block, press soft key [FIGURE] on the main menu to display the figure menu screen and then select the necessary figure.

(6) To return to the CNC program screen, press the leftmost soft key on the main menu screen.

NOTE
When a machining type block and figure block are entered in a machining cycle of MANUAL GUIDE 0*i*, these blocks must be paired.

2.2 EDITING A MACHINING CYCLE

Press function key [PROG] several times until soft key [C.A.P.] is displayed. To modify the machining type clock and figure block of a machining cycle, move the cursor to a part other then EOB of the block and press soft key [C.A.P.].
When the data input screen of the selected block is displayed, modify the necessary data and press soft key [OK] to modify the machining program. Then, the CNC screen is displayed again.
If the leftmost soft key is pressed on the input data screen, modification of data is canceled and the CNC screen is displayed again.

3 INPUT ITEM OF EACH CYCLE

Chapter 3, "INPUT ITEM OF EACH CYCLE", consists of the following sections:

3.1 HOLE MACHINING

3.1.1 Machining Type Block

Center drilling : G1000

Data item		Comment
W	MACHINING TYPE	1 : Drilling without dwell 2 : Drilling with dwell
C	CLEARANCE	Distance from the surface of workpiece to the start point of actual cutting motion
I	REFERENCE POSITION RETURN	1 : Return to initial point at moving to the next hole position 2 : Return to R point (start point of actual cutting motion) at moving to the next hole position
F	FEEDARATE	Cutting feedrate
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.

Drilling : G1001

Data item		Comment
W	MACHINING TYPE	1 : Drilling without dwell 2 : Drilling with dwell 3 : Peck drilling 4 : High speed peck drilling
C	CLEARANCE	Distance from the surface of workpiece to the start point of actual cutting motion
I	REFERENCE POSITION RETURN	1 : Return to initial point at moving to the next hole position 2 : Return to R point (start point of actual cutting motion) at moving to the next hole position
Q	CUTTING DEPTH	Cutting in depth of one cutting motion
F	FEEDARATE	Cutting feedrate
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.

Tapping : G1002

Data item		Comment
W	MACHINING TYPE	1 : Normal tapping 2 : Reverse tapping 3 : Rigid tapping 4 : Reverse rigid tapping
C	CLEARANCE	Distance from the surface of workpiece to the start point of actual cutting motion
I	REFERENCE POSITION RETURN	1 : Return to initial point at moving to the next hole position 2 : Return to R point (start point of actual cutting motion) at moving to the next hole position
F	FEEDARATE	Cutting feedrate
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.
L	THREADING LEAD	Lead of a tapping thread
S	SPINDLE SPEED	Spindle rotating speed (min-1)

Reaming : G1003

Data item		Comment
W	MACHINING TYPE	1 : Escape by G01 motion 2 : Escape by G00 motion 3 : Escape by G01 motion after dwell at bottom
C	CLEARANCE	Distance from the surface of workpiece to the start point of actual cutting motion
I	REFERENCE POSITION RETURN	1 : Return to initial point at moving to the next hole position 2 : Return to R point (start point of actual cutting motion) at moving to the next hole position
F	FEEDARATE	Cutting feedrate
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.

Boring : G1004

Data item		Comment
W	MACHINING TYPE	1 : Escape by G01 motion 2 : Escape by G00 motion 3 : Escape by G01 motion after dwell at bottom
C	CLEARANCE	Distance from the surface of workpiece to the start point of actual cutting motion
I	REFERENCE POSITION RETURN	1 : Return to initial point at moving to the next hole position 2 : Return to R point (start point of actual cutting motion) at moving to the next hole position
F	FEEDARATE	Cutting feedrate
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.

Fine boring : G1005

Data item		Comment
C	CLEARANCE	Distance from the surface of workpiece to the start point of actual cutting motion
I	REFERENCE POSITION RETURN	1 : Return to initial point at moving to the next hole position 2 : Return to R point (start point of actual cutting motion) at moving to the next hole position
F	FEEDARATE	Cutting feedrate
Q	SHIFT AMOUNT	Shift motion distance after oriented spindle stop at the hole bottom
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.

Back boring : G1006

Data item		Comment
C	CLEARANCE	Distance from the bottom surface of workpiece to the start point of actual cutting motion (positive value)
F	FEEDARATE	Cutting feedrate
Q	SHIFT AMOUNT	Shift motion distance after oriented spindle stop at the hole bottom
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.

3.1.2 Figure Block

Points - Hole pattern : G1210

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Hole depth (usually negative value)
H	POINT-1 (X-AXIS)	X-coordinate of 1st hole point
V	POINT-1 (Y-AXIS)	Y-coordinate of 1st hole point
A	POINT-2 (X-AXIS)	X-coordinate of 2nd hole point
C	POINT-2 (Y-AXIS)	Y-coordinate of 2nd hole point
D	POINT-3 (X-AXIS)	X-coordinate of 3rd hole point
E	POINT-3 (Y-AXIS)	Y-coordinate of 3rd hole point

NOTE
It is not necessary to enter all of the hole point data item, but a pair of X and Y data for one hole position.

Line (Equal interval) - Hole pattern : G1211

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Hole depth (usually negative value)
H	START POINT (X-AXIS)	Start point X-coordinate of the line (1st hole point)
V	START POINT (Y-AXIS)	Start point Y-coordinate of the line (1st hole point)
A	ANGLE	Angle of the line
C	NUMBER OF HOLE	Number of hole on the line
D	PITCH WIDTH	Distance on the line between 2 holes (positive value only)
E	LINE LENGTH	Distance on the line between line start point and the last hole point (positive value only)
F	OMITTING POINT-1	Hole number of 1st machining omitting hole
I	OMITTING POINT-2	Hole number of 2nd machining omitting hole
J	OMITTING POINT-3	Hole number of 3rd machining omitting hole
K	OMITTING POINT-4	Hole number of 4th machining omitting hole

NOTE
Data entering of either PICTH WIDTH or LINE LENGHTH is permitted.
It is not necessary to enter all of the OMITTING HOLE data item.

Line (Not Equal interval) - Hole pattern : G1212

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Hole depth (usually negative value)
H	START POINT (X-AXIS)	Start point X-coordinate of the line (1st hole point)
V	START POINT (Y-AXIS)	Start point Y-coordinate of the line (1st hole point)
A	ANGLE	Angle of the line

Data item		Comment
C	PITCH WIDTH-1	Distance on the line between 1st hole point and 2nd hole point (positive value only)
D	PITCH WIDTH-2	Distance on the line between 2nd hole point and 3rd hole point (positive value only)
E	PITCH WIDTH-3	Distance on the line between 3rd hole point and 4th hole point (positive value only)
F	PITCH WIDTH-4	Distance on the line between 4th hole point and 5th hole point (positive value only)
I	PITCH WIDTH-5	Distance on the line between 5th hole point and 6th hole point (positive value only)
J	PITCH WIDTH-6	Distance on the line between 6th hole point and 7th hole point (positive value only)
K	PITCH WIDTH-7	Distance on the line between 7th hole point and 8th hole point (positive value only)
M	PITCH WIDTH-8	Distance on the line between 8th hole point and 9th hole point (positive value only)
P	PITCH WIDTH-9	Distance on the line between 9th hole point and 10th hole point (positive value only)
Q	PITCH WIDTH-10	Distance on the line between 10th hole point and 11th hole point (positive value only)

NOTE
It is not necessary to enter all of the PITCH WIDTH data item.

Grid - Hole pattern : G1213

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Hole depth (usually negative value)
H	START POINT (X-AXIS)	Start point X-coordinate of the 1st side (1st hole point)
V	START POINT (Y-AXIS)	Start point Y-coordinate of the 1st side (1st hole point)
U	LENGTH ALONG TO THE 1ST SIDE	Length along to the 1st side
W	LENGTH ALONG TO THE 2ND SIDE	Length along to the 2nd side
I	NUMBER OF HOLE ALONG TO THE 1ST SIDE	Number of holes along to the 1st side
J	NUMBER OF HOLE ALONG TO THE 2ND SIDE	Number of holes along to the 2nd side
K	ANGLE OF THE 1ST SIDE	Angle of the 1st side
M	ANGLE OF THE 2ND SIDE	Angle of the 2nd side
A	OMITTING POINT-1	Hole number of 1st machining omitting hole
C	OMITTING POINT-2	Hole number of 2nd machining omitting hole
D	OMITTING POINT-3	Hole number of 3rd machining omitting hole
E	OMITTING POINT-4	Hole number of 4th machining omitting hole

NOTE
It is not necessary to enter all of the OMITTING HOLE data item.

Square - Hole pattern : G1214

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Hole depth (usually negative value)
H	START POINT (X-AXIS)	Start point X-coordinate of the 1st side (1st hole point)
V	START POINT (Y-AXIS)	Start point Y-coordinate of the 1st side (1st hole point)

Data item		Comment
U	LENGTH ALONG TO THE 1ST SIDE	Length along to the 1st side
W	LENGTH ALONG TO THE 2ND SIDE	Length along to the 2nd side
I	NUMBER OF HOLE ALONG TO THE 1ST SIDE	Number of holes along to the 1st side
J	NUMBER OF HOLE ALONG TO THE 2ND SIDE	Number of holes along to the 2nd side
K	ANGLE OF THE 1ST SIDE	Angle of the 1st side
M	ANGLE OF THE 2ND SIDE	Angle of the 2nd side
A	OMITTING POINT-1	Hole number of 1st machining omitting hole
C	OMITTING POINT-2	Hole number of 2nd machining omitting hole
D	OMITTING POINT-3	Hole number of 3rd machining omitting hole
E	OMITTING POINT-4	Hole number of 4th machining omitting hole

NOTE
It is not necessary to enter all of the OMITTING HOLE data item.

Circle - Hole pattern : G1215

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Hole depth (usually negative value)
H	CENTER POINT (X-AXIS)	Center point X-coordinate of the circle
V	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the circle
R	RADIUS	Radius of the circle
A	START ANGLE	Center angle of the 1st hole point
C	NUMBER OF HOLE	Number of holes along to the circle
D	OMITTING POINT-1	Hole number of 1st machining omitting hole
E	OMITTING POINT-2	Hole number of 2nd machining omitting hole
F	OMITTING POINT-3	Hole number of 3rd machining omitting hole
I	OMITTING POINT-4	Hole number of 4th machining omitting hole

NOTE
It is not necessary to enter all of the OMITTING HOLE data item.

Arc (Equal interval) - Hole pattern : G1216

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Hole depth (usually negative value)
H	CENTER POINT (X-AXIS)	Center point X-coordinate of the arc
V	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the arc
R	RADIUS	Radius of the arc
A	START ANGLE	Center angle of the 1st hole point
C	PITCH ANGLE	Center angle between 2 holes point
D	NUMBER OF HOLE	Number of holes along to the arc
E	OMITTING POINT-1	Hole number of 1st machining omitting hole
F	OMITTING POINT-2	Hole number of 2nd machining omitting hole
I	OMITTING POINT-3	Hole number of 3rd machining omitting hole
J	OMITTING POINT-4	Hole number of 4th machining omitting hole

NOTE
It is not necessary to enter all of the OMITTING HOLE data item.

Arc (Not Equal interval) - Hole pattern : G1217

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Hole depth (usually negative value)
H	CENTER POINT (X-AXIS)	Center point X-coordinate of the arc
V	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the arc
R	RADIUS	Radius of the arc
A	START ANGLE	Center angle of the 1st hole point
C	PITCH ANGLE-1	Center angle between 1st hole and 2nd hole
D	PITCH ANGLE-2	Center angle between 2nd hole and 3rd hole
E	PITCH ANGLE-3	Center angle between 3rd hole and 4th hole
F	PITCH ANGLE-4	Center angle between 4th hole and 5th hole
I	PITCH ANGLE-5	Center angle between 5th hole and 6th hole
J	PITCH ANGLE-6	Center angle between 6th hole and 7th hole
K	PITCH ANGLE-7	Center angle between 7th hole and 8th hole
M	PITCH ANGLE-8	Center angle between 8th hole and 9th hole
P	PITCH ANGLE-9	Center angle between 9th hole and 10th hole
Q	PITCH ANGLE-10	Center angle between 10th hole and 11th hole

3.2 FACE MACHINING

3.2.1 Machining Type Block

Facing - Rough : G1020

Data item		Comment
T	THICKNESS	Removal amount at roughing
J	CUT DEPTH OF TOOL –AXIS	Cutting depth of one cutting along the tool axis direction (Z-axis)
H	FINISHING AMOUNT	Finishing amount
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
D	TOOL DIAMETER	Tool diameter
L	CUT AMOUNT OF TOOL-RADIUS	Cutting amount of the tool radius direction
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	MACHINING TYPE	1 : Single directional cutting on X-Y plane 2 : Both directional cutting on X-Y plane
M	CLEARANCE ALONG TO TOOL RADIUS	Distance from the side surface of workpiece to the cutting start point on X-Y plane
B	START POINT	1 : Start cutting at point 1 2 : Start cutting at point 2 3 : Start cutting at point 3 4 : Start cutting at point 4

Facing - Finish : G1021

Data item		Comment
F	FEEDRATE	Finishing feedrate along the tool radius direction
D	TOOL DIAMETER	Tool diameter
L	CUT AMOUNT OF TOOL RADIUS	Cutting amount of the tool radius direction
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	MACHINING TYPE	1 : Single directional cutting on X-Y plane 2 : Both directional cutting on X-Y plane
M	CLEARANCE ALONG TO TOOL RADIUS	Distance from the side surface of workpiece to the cutting start point on X-Y plane

Data item		Comment
B	START POINT	1 : Start cutting at point 1 2 : Start cutting at point 2 3 : Start cutting at point 3 4 : Start cutting at point 4

3.2.2 Figure Block

Square - Fixed figure : G1220

Data item		Comment
H	CENTER POINT (X-AXIS)	Center point X-coordinate of the square
V	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the square
U	LENGTH ALONG TO THE X-AXIS	Length of square side along X-axis
W	LENGTH ALONG TO THE Y-AXIS	Length of square side along Y-axis
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Not used in facing, so need not enter
A	ANGLE	Angle from X-axis
R	CORNER RADIUS	Radius of each corner of the square

NOTE
It is not necessary to enter the DEPTH and CORNER RADIUS data item.

Circle - Fixed figure : G1221

Data item		Comment
H	CENTER POINT (X-AXIS)	Center point X-coordinate of the circle
V	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the circle
R	RADIUS	Radius of the circle
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Not used in facing, so need not enter

NOTE
It is not necessary to enter the DEPTH data item.

3.3 POCKET MACHINING

3.3.1 Machining Type Block

Pocketing - Rough : G1040

Data item		Comment
T	BOTTOM THICKNESS	Removal amount of pocket part, distance from the bottom of pocket
J	CUT DEPTH OF TOOL –AXIS	Cutting depth of one cutting along the tool axis direction (Z-axis)
H	BOTTOM FINISH AMOUNT	Finishing amount of the bottom part
K	SIDE FINISH AMOUNT	Finishing amount of the side wall part
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate along the tool axis (Z) direction
D	TOOL DIAMETER	Tool diameter
L	CUT AMOUNT OF TOOL-RADIUS	Cutting amount of the tool radius direction
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	UP CUT/DOWN CUT	1 : Carry out down cutting way 2 : Carry out up cut way

Pocketing - Bottom finish : G1041

Data item		Comment
T	BOTTOM THICKNESS	Removal amount at bottom part
K	SIDE FINISH AMOUNT	Finishing amount of the side wall part
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate along the tool axis (Z) direction
D	TOOL DIAMETER	Tool diameter
L	CUT AMOUNT OF TOOL-RADIUS	Cutting amount of the tool radius direction
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	UP CUT/DOWN CUT	1 : Carry out down cutting way 2 : Carry out up cut way

Pocketing - Side finish : G1042

Data item		Comment
T	BOTTOM THICKNESS	Removal amount at bottom part
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate along the tool axis (Z) direction
D	TOOL DIAMETER	Tool diameter
S	CUTTER COMPENSATION NO.	Offset number of cutter-R compensation
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	UP CUT/DOWN CUT	1 : Carry out down cutting way 2 : Carry out up cut way
R	APPROACH/ESCAPE RADIUS	Arc radius of approaching and escaping motion

Pocketing - Chamfer : G1043

Data item		Comment
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate along the tool axis (Z) direction
Z	CHAMFER AMOUNT	Chamfering amount at the top of side wall part
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	UP CUT/DOWN CUT	1 : Carry out down cutting way 2 : Carry out up cut way
M	APPROACH RADIUS	Arc radius of approaching motion
K	TOOL SMALL DIAMETER	Diameter of the chamfering tool end part
H	TOOL END CLEARANCE	Clearance amount at the chamfering tool end part

Pocketing - Drilling : G1045

Data item		Comment
T	BOTTOM THICKNESS	Removal amount of pocket part
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
R	MACHINING TYPE	1 : Normal drilling with out dwell 3 : Peck drilling without dwell Note) Type 2 and 4 are cannot be used.
F	FEEDRATE	Cutting feedrate
Q	CUTTING DEPTH	Cutting in depth of one cutting motion

NOTE
This cycle is used as pre-hole drilling before pocket machining.

3.3.2 Figure Block

Square - Fixed figure : G1220

Data item		Comment
H	CENTER POINT (X-AXIS)	Center point X-coordinate of the square
V	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the square
U	LENGTH ALONG TO THE X-AXIS	Length of square side along X-axis
W	LENGTH ALONG TO THE Y-AXIS	Length of square side along Y-axis
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Depth of pocket, distance from workpiece surface to the bottom of pocket, usually negative value
A	ANGLE	Angle from X-axis
R	CORNER RADIUS	Radius of each corner of the square

NOTE
It is not necessary to enter the CORNER RADIUS data item, if the actual figure need not it.

Circle - Fixed figure : G1221

Data item		Comment
H	CENTER POINT (X-AXIS)	Center point X-coordinate of the circle
V	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the circle
R	RADIUS	Radius of the circle
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Depth of pocket, distance from workpiece surface to the bottom of pocket, usually negative value

Track - Fixed figure : G1222

Data item		Comment
H	CENTER POINT (X-AXIS)	Center point X-coordinate of the 1st circle
V	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the 1st circle
U	DISTANCE BETWEEN CENTERS	Distance between 1st circle center and 2nd circle center
R	RADIUS	Radius of the 2 circles (must be same radius)
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	DEPTH	Depth of pocket, distance from workpiece surface to the bottom of pocket, usually negative value
A	ANGLE	Angle from X-axis around the center of 1st circle

3.4 GROOVE MACHINING

3.4.1 Machining Type Block

Grooving - Rough : G1050

Data item		Comment
T	BOTTOM THICKNESS	Removal amount of groove part, distance from the bottom of groove
J	CUT DEPTH OF TOOL –AXIS	Cutting depth of one cutting along the tool axis direction (Z-axis)
H	BOTTOM FINISH AMOUNT	Finishing amount of the bottom part
K	SIDE FINISH AMOUNT	Finishing amount of the side wall part
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate along the tool axis (Z) direction
D	TOOL DIAMETER	Tool diameter
L	CUT AMOUNT OF TOOL-RADIUS	Cutting amount of the tool radius direction

Data item		Comment
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	UP CUT/DOWN CUT	1 : Carry out down cutting way 2 : Carry out up cut way

Grooving - Bottom finish : G1051

Data item		Comment
T	BOTTOM THICKNESS	Removal amount at bottom part
K	SIDE FINISH AMOUNT	Finishing amount of the side wall part
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate along the tool axis (Z) direction
D	TOOL DIAMETER	Tool diameter
L	CUT AMOUNT OF TOOL-RADIUS	Cutting amount of the tool radius direction
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	UP CUT/DOWN CUT	1 : Carry out down cutting way 2 : Carry out up cut way

Grooving - Side finish : G1052

Data item		Comment
T	BOTTOM THICKNESS	Removal amount at bottom part
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate along the tool axis (Z) direction
D	TOOL DIAMETER	Tool diameter
S	CUTTER COMPENSATION NO.	Offset number of cutter-R compensation
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	UP CUT/DOWN CUT	1 : Carry out down cutting way 2 : Carry out up cut way
R	APPROACH/ESCAPE RADIUS	Arc radius of approaching and escaping motion

Grooving - Chamfer : G1053

Data item		Comment
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate along the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate along the tool axis (Z) direction
Z	CHAMFER AMOUNT	Chamfering amount at the top of side wall part
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
W	UP CUT/DOWN CUT	1 : Carry out down cutting way 2 : Carry out up cut way
M	APPROACH RADIUS	Arc radius of approaching motion
K	TOOL SMALL DIAMETER	Diameter of the chamfering tool end part
H	TOOL END CLEARANCE	Clearance amount at the chamfering tool end part

Grooving - Drilling : G1055

Data item		Comment
T	BOTTOM THICKNESS	Removal amount of pocket part
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point along Z-axis
R	MACHINING TYPE	1 : Normal drilling with out dwell 3 : Peck drilling without dwell Note) Type 2 and 4 are cannot be used.
F	FEEDRATE	Cutting feedrate
Q	CUTTING DEPTH	Cutting in depth of one cutting motion

NOTE
This cycle is used as pre-hole drilling before groove machining.

3.4.2 Figure Block

Radial line grooves - Fixed figure : G1223

Data item		Comment
U	DISTANCE BETWEEN CENTERS	Distance between 2 circle centers at the each end of groove
D	GROOVE WIDTH	Width of the groove
L	DEPTH	Depth of groove, distance from workpiece surface to the bottom of pocket, usually negative value
E	GROOVE ANGLE	Angle of the groove
A	ANGLE	Angle of the 1st radial groove center point from X-axis
H	CENTER POINT (X-AXIS)	X-coordinate of center point of the plural radial grooves
V	CENTER POINT (Y-AXIS)	Y-coordinate of center point of the plural radial grooves
R	GROOVE POSITION RADIUS	Radius of arc on which plural radial grooves are positioned
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
C	PITCH ANGLE	Center angle between 2 radial grooves
M	NUMBER OF GROOVE	Number of radial grooves

NOTE
1 Position of a 1st radial groove is calculated from CENTER POINT, GROOVE POSITION RADIUS and ANGLE, so these data must be entered always.
2 If only one groove is necessary, PITCH ANGLE need not be entered.

IV. TURNING CYCLE
(LATHE SYSTEM)

1 MACHINING TYPE

MANUAL GUIDE 0*i* provides the following machining cycles.

Drilling			
	Machining type block	G1100	Center drilling
		G1101	Drilling
		G1102	Tapping
		G1103	Reaming
		G1104	Boring
Turning			
	Machining type block	G1120	Outer Bar Roughing
		G1121	Internal Bar Roughing
		G1122	End Face Roughing
		G1123	Outer Bar Finishing
		G1124	Internal Bar Finishing
		G1125	End Face Finishing
	Figure block	G1400	Start
		G1401	Line
		G1402	Arc CW
		G1403	Arc CCW
		G1406	Figure End
Grooving			
	Machining type block	G1130	Outer Groove Roughing
		G1131	Internal Groove Roughing
		G1132	End face Groove Roughing
		G1133	Outer Groove Finishing
		G1134	Inner Groove Finishing
		G1135	End face Groove Finishing
	Figure block	G1460	Normal Groove
		G1461	Trapezoidal Groove
Threading			
	Machining type block	G1140	Outer Threading
		G1141	Inner Threading
	Figure block	G1450	Thread figure

2 OPERATING METHOD

Chapter 2, "OPERATING METHOD", consists of the following sections:

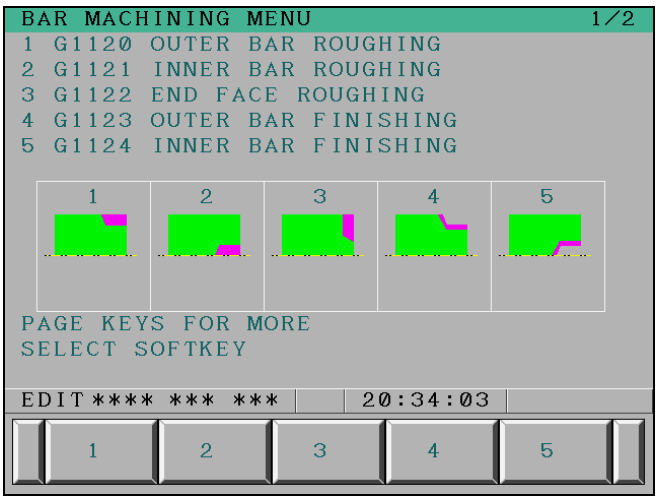
NOTE
1 When using this machining cycle, be sure to specify a command to move to the cutting start point before a machining type block (see the next section).
Set the cutting start point to the position away from the figure start point by the amount of clearance (the amount of tool-nose radius compensation or more).
2 When using this machining cycle, be sure to set parameters No. 9364, No. 9365, No. 9381, and No.9382. For details on these parameters, see Appendix B, "PARAMETERS".

2.1 NEWLY CREATING A MACHINING CYCLE

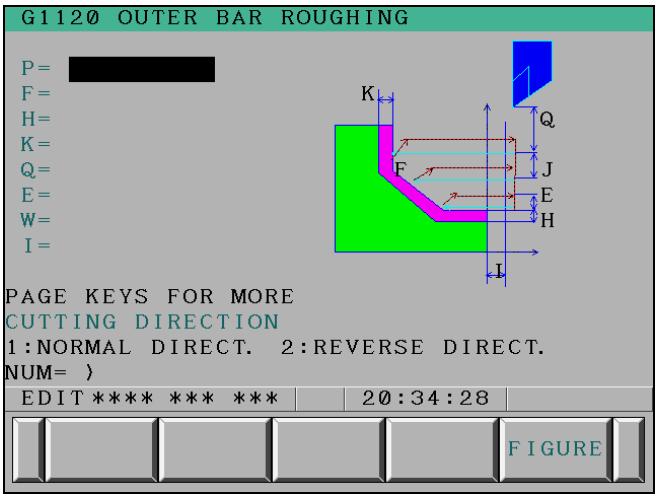
The method for newly creating a machining cycle is described below.

2.1.1 Operating Method of Turning

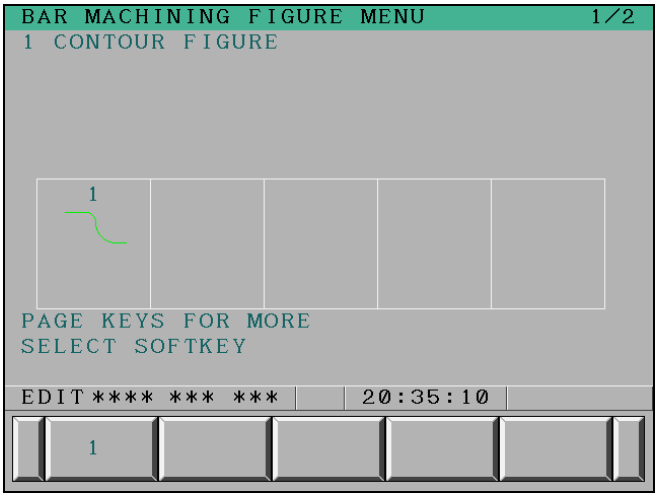
(1) Select turning with the corresponding soft key on the main menu screen. The machining menu screen below is displayed.



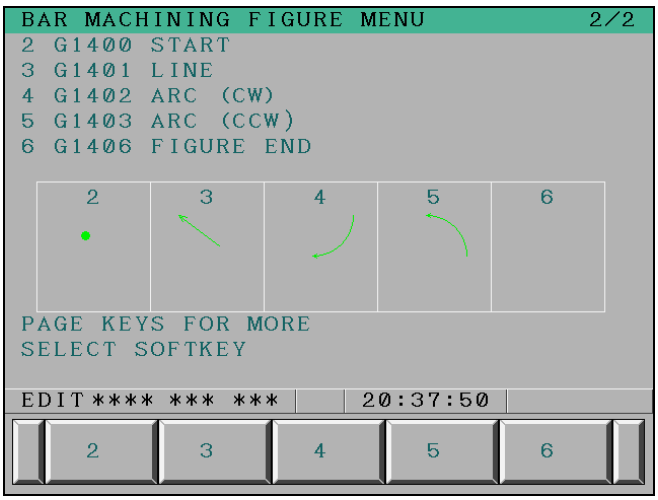
(2) Select the desired machining type with the corresponding soft key. The data input screen of the selected machining type block (when rough machining is selected) as shown below appears. For details on the input items of each machining type block, see the next section. To cancel insertion, press the leftmost soft key.



(3) When the necessary data is entered and soft key [FIGURE] is pressed, the turning figure menu screen below is displayed.



When MDI page key [↓] is pressed, turning figure menu screen (2) below is displayed.



(a) When using method with calculating end point automatically
When [1 CONTOUR FIGURE] is selected on the BAR MACHINING FIGURE menu, a free-format figure consisting of lines and arcs can be input using all functions of contour calculation. For details, see Section 4.1 ""AUTOMATIC CALCULATING END POINT" TYPE".

(b) When using method with entering end point directly
When any of items 2 to 6 is selected on the BAR MACHINING FIGURE menu, a figure block can be entered directly. At this time, the figure is created so that G1400 (start) is entered first and it ends with G1406 (figure end) finally. Therefore, the end coordinates and other necessary values need to be input for each figure block. Contour calculation cannot be used. For details, see Section 4.2 ""INPUTING DIRECTLY END POINT" TYPE".

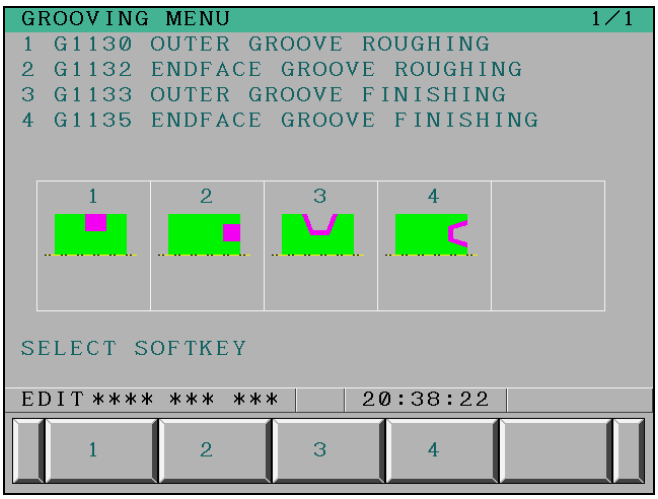
(4) When a figure is selected with the corresponding soft key, a data input screen as shown below is displayed. For details on the input items for each figure block, see the next section.

2.1.2 Operating Method of Drilling, Grooving, or Threading

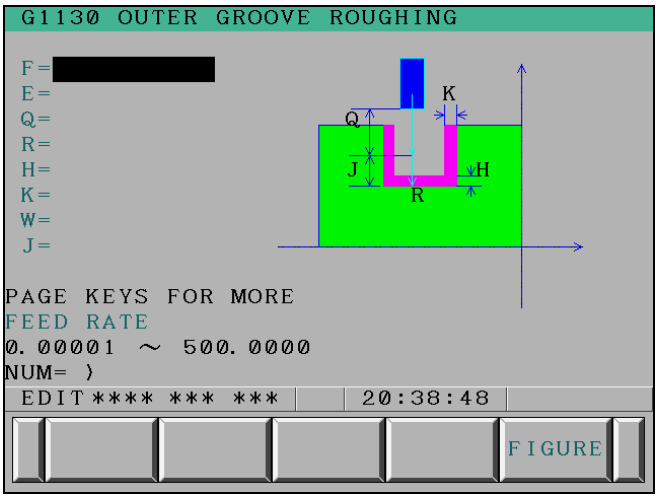
Select the machining type (drilling, grooving, or threading) of a desired machining cycle on the main menu screen with the corresponding soft key.

(Example of operating procedure)

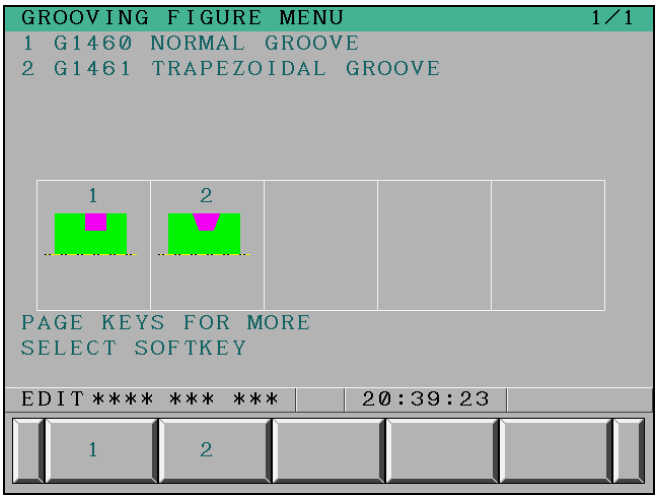
(1) When, grooving is selected, the machining menu screen shown below is displayed.



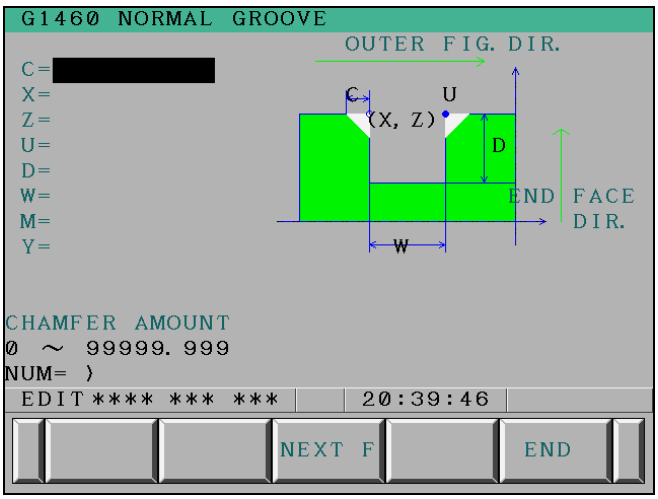
(2) Select the desired machining type with the corresponding soft key. The data input screen of the selected machining type block (when rough machining is selected) as shown below appears. For details on the input items of each machining type block, see the next section. To cancel insertion, press the leftmost soft key.



(3) When the necessary data is entered and soft key [FIGURE] is pressed, a figure menu screen as shown below appears. Only the menu of figures that can be used in the machining type block entered immediately before is displayed. The machining type block entered here is added to the machining program.



(4) When a figure is selected with the corresponding soft key, an data input screen as shown below appears (NORMAL GROOVE is selected in the following figure). For details on the input item of each figure block, see the next section.



(5) When the necessary data has been entered, press soft key [NEXT F] or [END]. When [END] is pressed, the entered figure block is added to the machining program and the main menu is displayed.
To enter a figure continuously, press [NEXT F]. Then, the entered figure block is added to the machining program and the figure menu screen is displayed again.
To enter a machining type block and figure block and then add a figure block, press soft key [FIGURE] on the main menu to display the figure menu screen and then select the necessary figure.

(6) To return to the CNC program screen, press the leftmost soft key on the main menu screen.

NOTE
When a machining type block and figure block are entered in a machining cycle of MANUAL GUIDE 0*i*, these blocks must be paired.

2.2 EDITING A MACHINING CYCLE

Press function key [PROG] several times until soft key [C.A.P.] is displayed. To modify the machining type clock and figure block of a machining cycle, move the cursor to a part other then EOB of the block and press soft key [C.A.P.].
When the data input screen of the selected block is displayed, modify the necessary data and press soft key [OK] to modify the machining program. Then, the CNC screen is displayed again.
If the leftmost soft key is pressed on the input data screen, modification of data is canceled and the CNC screen is displayed again.

3 INPUT ITEM OF EACH CYCLE

Chapter 3, "INPUT ITEM OF EACH CYCLE", consists of the following sections:

3.1 DRILLING

3.1.1 Machining Type Block

Center drilling : G1100

Data item		Comment
F	FEED RATE	Cutting feedrate
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.
B	START POINT	Z-coordinate of cutting start point
L	DEPTH	Depth of hole (positive value)

Drilling : G1101

Data item		Comment
W	MACHINING TYPE	1 : Drilling 2 : Peck drilling 4 : High speed peck drilling
F	FEED RATTE	Cutting in depth of one cutting motion
Q	CUTTING DEPTH	Cutting in depth of one cutting motion
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.
B	START POINT	Z-coordinate of cutting start point
L	DEPTH	Depth of hole (positive value)

Tapping : G1102

Data item		Comment
W	MACHINING TYPE	1 : Normal tapping 2 : Reverse tapping 3 : Rigid tapping 4 : Reverse rigid tapping
F	THREAD PITCH	Lead of a tapping thread
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.
B	START POINT	Z-coordinate of cutting start point
L	DEPTH	Depth of hole (positive value)

Reaming : G1103

Data item		Comment
F	FEED RATE	Cutting feedrate
Q	CUTTING LIP LENGTH	Cutting lip length of a reamer at cutting start end
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.
B	START POINT	Z-coordinate of cutting start point
L	DEPTH	Depth of hole (positive value)

Boring : G1104

Data item		Comment
F	FEED RATE	Cutting feedrate
Q	SHIFT AMOUNT	Shift in retraction for boring (positive value)
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec.
B	START POINT	Z-coordinate of cutting start point
L	DEPTH	Depth of hole (positive value)

3.2 TURNING

3.2.1 Machining Type Block

Outer Bar Roughing : G1120
Internal Bar Roughing : G1121
End Face Roughing : G1122

Data item		Comment
P	CUTTING DIRECTION	1 : Normal direction (from end face toward chuck) 2 : Reverse direction (from chuck toward end face)
F	FEED RATE	Cutting feedrate
H	FINISH AMOUNT X	Finishing amount of X-axis direction (diameter)
K	FINISH AMOUNT Z	Finishing amount of Z-axis direction (radius)
Q	1ST CUTTING DEPTH	Depth of 1st cut in rough machining (diameter)
E	ESCAPE AMOUNT	Movement along the X-axis (in outer/inner surface machining : diameter) or along the Z-axis (in end facing : radius) for retraction after cutting
W	ESCAPE TYPE	Escaping type after cutting in a roughing motions. 1 : Standard The tool is retracted after cutting along the final figure outline 2 : Rapid The tool is retracted immediately after cutting
I	END FACE REMOVAL	removal amount at the end face part (radius)
J	RATE OF CUTTING DEPTH	Ratio of the depth of an actual cut to the depth of cut entered by Q
U	MINIMUM CUT DEPTH	Limit of depth of cut (diameter)
V	EDGE ANGLE	Angle of the cutting edge. In case of that it was set to the value less than 90 degrees, cutting edge compensation will be done automatically.
A	NOSE ANGLE	Angle of the tool. In the case of that pocket figure was inputted as contour figure, cutting back compensation will be done automatically.

Outer Bar Finishing : G1123
Internal Bar Finishing : G1124
End Face Bar Finishing : G1125

Data item		Comment
P	CUTTING DIRECTION	1 : Normal direction (from end face toward chuck) 2 : Reverse direction (from chuck toward end face)

Data item		Comment
F	FEED RATE	Cutting feedrate
E	ESCAPE AMOUNT	Movement along the X-axis (in outer/inner surface machining : diameter) or along the Z-axis (in end facing : radius) for retraction after cutting
V	EDGE ANGLE	Angle of the cutting edge. In case of that it was set to the value less than 90 degrees, cutting edge compensation will be done automatically.
A	NOSE ANGLE	Angle of the tool. In the case of that pocket figure was inputted as contour figure, cutting back compensation will be done automatically.

3.2.2 Figure Block

Refer to the chapter 4.”DEFINING FIGURE FOR TURNING ”.

3.3 GROOVING

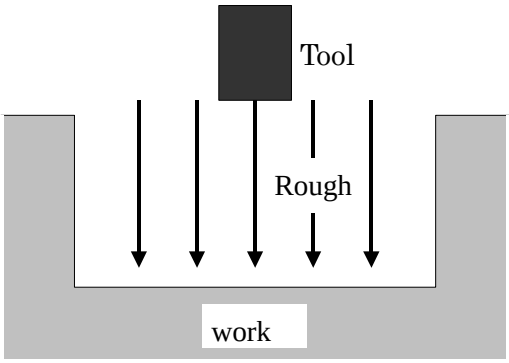
3.3.1 Machining Type Block

(In Case of the parameter No.9302#4=1)

Outer Groove Roughing : G1130
Internal Groove Roughing : G1131
End face Groove Roughing : G1132

Data item		Comment
F	FEED RATE	Cutting feedrate
E	TOOL WIDTH	Width of grooving tool at cutting edge (radius)
Q	CUTTING DEPTH	Depth of each cutting motion (outer/inner : diameter, end face : radius)
R	DWELL TIME	Dwell time at the bottom of grove, in unit of msec.
H	FINISH AMOUNT X	Finishing amount of X-axis direction (diameter)
K	FINISH AMOUNT Z	Finishing amount of Z-axis direction (radius)
W	CUTTING METHOD	Cutting sequence of groove part 1 : Single direction Cutting is done from end face toward chuck side 2: Both direction At first, cutting at the center of groove is done, and expanded to end face side and chuck side mutually
J	RATE OF CUTTING DEPTH	Ratio of the depth of an actual cut to the depth of cut entered by Q
U	MINIMUM CUT DEPTH	Limit of depth of cut (diameter)

In case of the parameter No.9302#4=1, only Rouging is executed as follows.

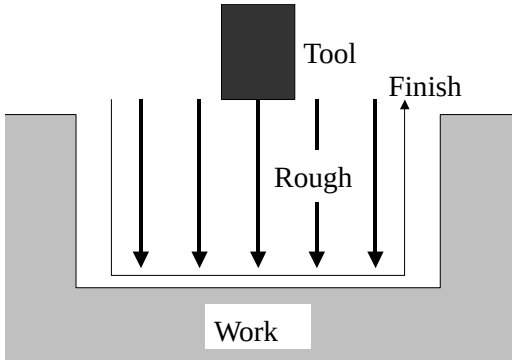


(In Case of the parameter No.9302#4=0)

Outer Groove Roughing and Finishing : G1130
Internal Groove Roughing and Finishing : G1131
End face Groove Roughing and Finishing : G1132

Data item		Comment
F	FEED RATE	Cutting feedrate
E	TOOL WIDTH	Width of grooving tool at cutting edge (radius)
Q	CUTTING DEPTH	Depth of each cutting motion (outer/inner : diameter, end face : radius)
R	DWELL TIME	Dwell time at the bottom of grove, in unit of msec.
H	FINISH AMOUNT X	Finishing amount of X-axis direction (diameter) When Normal groove is specified, this data is ignored.
K	FINISH AMOUNT Z	Finishing amount of Z-axis direction (radius) When Normal groove is specified, this data is ignored.
W	CUTTING METHOD	Cutting sequence of groove part 1 : Single direction Cutting is done from end face toward chuck side 2: Both direction At first, cutting at the center of groove is done, and expanded to end face side and chuck side mutually
J	RATE OF CUTTING DEPTH	Ratio of the depth of an actual cut to the depth of cut entered by Q
U	MINIMUM CUT DEPTH	Limit of depth of cut (diameter)

In machining of normal groove, Finishing is executed after Rouging as follows. But, in machining of trapezoidal groove, only Roughing is executed .



Outer Groove Finishing : G1133
Internal Groove Finishing : G1134
End face Groove Finishing : G1135

Data item		Comment
F	FEED RATE	Cutting feedrate
E	TOOL WIDTH	Width of grooving tool at cutting edge (radius)
W	CUTTING METHOD	Cutting sequence of groove part 1 : Single direction Cutting is done from end face toward chuck side 2: Both direction At first, cutting at the center of groove is done, and expanded to end face side and chuck side mutually

3.3.2 Figure Block

Normal Groove : G1460

Data item		Comment
C	CHAMFER AMOUNT	Chamfer amount of a groove (radius)
X	START POINT X	X-axis coordinate of a point where grooving starts
Z	START POINT Z	Z-axis coordinate of a point where grooving starts
U	END POINT (X/Z-AXIS)	X-axis or Z-axis coordinate of the end point of a groove. This data is necessary only at the depth of a groove is different. If no value is inputted, depth of start point and end point becomes equal. X value is used at outer groove, and Z value is used at end face groove.
D	GROOVE DEPTH	Depth of a groove to be machine (radius)
W	GROOVE WIDTH	Width of a groove to be machined (radius)
M	NUMBER OF GROOVE	Number of grooves when more than one groove are to be machined at regular intervals.
Y	PITCH	Distance between grooves when more than one groove are machined. When minus value is entered, grooves will be positioned from end face side toward chuck side. When plus value, it will be reversed (radius)

Trapezoidal Groove : G1461

Data item		Comment
X	START POINT X	X-axis coordinate of a start point
Z	START POINT Z	Z-axis coordinate of a start point
A	END POINT-1 (X-AXIS)	X-axis coordinate of a 1st line from start point
B	END POINT-1 (Z-AXIS)	Z-axis coordinate of a 1st line from start point
C	CORNER R 1	Corner-R radius of the end point of line-1
D	CHAMFER 1	Chamfering amount of the end point of line-1
E	END POINT-2 (X-AXIS)	X-axis coordinate of a 2nd line from 1st end point
F	END POINT-2 (Z-AXIS)	Z-axis coordinate of a 2nd line from 1st end point
H	CORNER R 2	Corner-R radius of the end point of 2nd line
I	CHAMFER 2	Chamfering amount of the end point of 2nd line
J	END POINT-3 (X-AXIS)	X-axis coordinate of a 3rd line from 2nd end point
K	END POINT-3 (Z-AXIS)	Z-axis coordinate of a 3rd line from 2nd end point
L	CORNER R 3	Corner-R radius of the end point of 3rd line
T	CHAMFER 3	Chamfering amount of the end point of 3rd line
P	END POINT-4 (X-AXIS)	X-axis coordinate of a 4th line from 3rd end point
Q	END POINT-4 (Z-AXIS)	Z-axis coordinate of a 4th line from 3rd end point
R	CORNER R 4	Corner-R radius of the end point of 4th line
S	CHAMFER 4	Chamfering amount of the end point of 4th line
V	FINAL POINT (X-AXIS)	X-axis coordinate of a 5th line from 4th end point

Data item		Comment
W	FINAL POINT (Z-AXIS)	Z-axis coordinate of a 5th line from 4th end point
M	NUMBER OF GROOVE	Number of grooves when more than one groove are to be machined at regular intervals.
Y	PITCH	Distance between grooves when more than one groove are machined. When minus value is entered, grooves will be positioned from end face side toward chuck side. When plus value, it will be reversed

NOTE
When both of Corner R and Chamfering data are entered at the same time for each point respectively, Corner R data is used and Chamfering data will be neglected.

3.4 THREADING

3.4.1 Machining Type Block

Threading : G1140

Data item		Comment
Q	CUTTING DEPTH	Depth of cut of the 1st pass (diameter)
J	CUTTING TIMES	Threading is repeated by this number including spark-out motion
W	CUTTING METHOD	1 : Single side cutting, constant cutting amount 2 : Both side cutting, constant cutting amount 3 : Straight cutting, constant cutting amount 4 : Single side cutting, constant cutting depth 5 : Both side cutting, constant cutting depth 6 : Straight cutting, constant cutting depth
L	SPARK OUT	Number of finishing cutting to be done
H	FINISH AMOUNT	Finishing amount (diameter)
C	CLEARANCE X	Clearance along the X-axis in threading (diameter)
M	CLEARANCE Z	Clearance amount along the Z-axis in threading, and it means the distance required for the spindle to reach a stable speed (radius)
A	NOSE ANGLE	Tool tip angle used for only GENERAL thread

NOTE
When both of Cutting depth and Cutting number are entered at the same time, Cutting depth data is used for actual threading motion, and Cutting number will be neglected.

3.4.2 Figure Block

Thread figure : G1450

Data item		Comment
R	THREAD TYPE	1 : General thread 2 : Metric thread 3 : Unified thread 4 : PT thread 5 : PF thread
L	THREADING LEAD	Thread lead. The least increment is 0.0001mm or 0.000001inch. This data is used for General, Metric, PT and PF thread.
N	NUMBER OF THREAD PER 1 INCH	Number of thread per 1 inch. The least input increment is 0.1. This data is used for Unified thread only.

Data item		Comment
H	THREAD HEIGHT	Thread height (radius)
P	MULTI THREAD NUMBER	Number of thread at multiple thread screw
X	START POINT X	X-coordinate of a thread start point
Z	START POINT Z	Z-coordinate of a thread start point
A	END POINT 1 (X)	X-coordinate of a 1st thread end point
B	END POINT 1 (Z)	Z-coordinate of a 1st thread end point
C	END POINT 2 (X)	X-coordinate of a 2nd continuous thread end point
D	END POINT 2 (Z)	Z-coordinate of a 2nd continuous thread end point
E	END POINT 3 (X)	X-coordinate of a 3rd continuous thread end point
F	END POINT 3 (Z)	Z-coordinate of a 3rd continuous thread end point
I	END POINT 4 (X)	X-coordinate of a 4th continuous thread end point
J	END POINT 4 (Z)	Z-coordinate of a 4th continuous thread end point
Q	END POINT 5 (X)	X-coordinate of a 5th continuous thread end point
S	END POINT 5 (Z)	Z-coordinate of a 5th continuous thread end point
V	END POINT 6 (X)	X-coordinate of a 6th continuous thread end point
W	END POINT 6 (Z)	Z-coordinate of a 6th continuous thread end point

NOTE

End point 2 to 6 are used at only when continuous thread machining under General thread is selected.
When Metric, Unified, PT or PF is selected, only End point 1 is available.

4 DEFINING FIGURE FOR TURNING

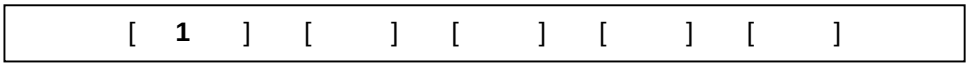
The method of defining figure for turning is the following two.
(1) Automatic Calculating End Point" Type
(2) "Inputting End Point Directly" Type"

4.1 "AUTOMATIC CALCULATING END POINT" TYPE

In this programming method, end point of figure is calculated automatically.

4.1.1 Start-up Method

Press soft key [1] on the page 1/2 of "Bar Machining Figure Menu" screen.

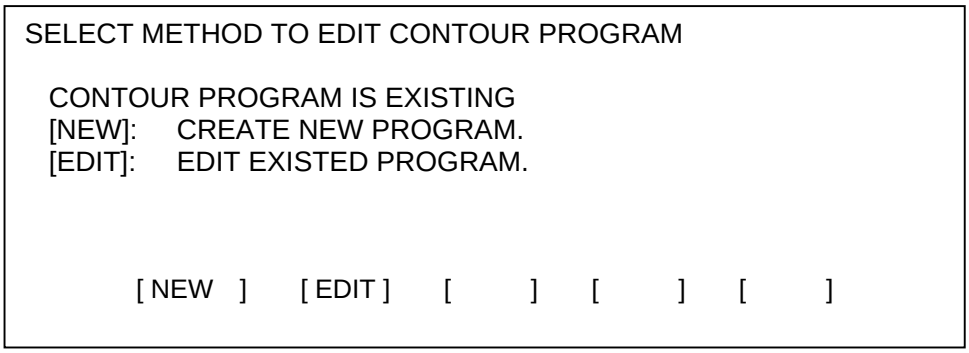


After the soft key [1] is pressed, the screen for defining Start Point is displayed.

4.1.2 Selecting Method of Editing The Turning Figure

If turning figure is existing, when soft key [1] is depressed, the following initial screen is displayed.
An operator can select whether a new turning figure is to be created, or whether an existing one is to be edited.

When you want to create a new turning figure, press soft key [NEW]. To work with an existing turning figure, press soft key [EDIT].



After the soft key [NEW] is pressed, input data screen of Start Point is displayed.
After the soft key [EDIT] is pressed, program list screen is displayed.

NOTE
In figure programming, a temporary working program must be allocated to hold a temporary NC program. A program number of this temporary program should be set in parameter No.9330 using a non-zero value. In this manual, all explanations assume that 9999 is set.

4.1.3 Method of Creating of turning Figure

(1) Start point

In case of creating of turning figure, after the operation explained in the section 4.1.1 or 4.1.2, the data input screen of Start Point is displayed.
In this screen, the following soft keys are displayed.

[AUX]	[ ]	[ ]	[OK]	[EXIT]

[AUX.] : Call auxiliary calculating screen. The result of it will be set into start point coordinate data.
[OK] : Fix start point data and store into memory.
[EXIT] : Cancel start point entering and exit from contour programming.

(2) Select type of turning figure

After the start point is defined, program list screen is displayed. On the program list screen, the following figure selecting or other operating soft-key will be displayed. In this screen operations for turning figure entering can be done.

[LINE]	[ARC↻]	[ARC↺]	[CORNER]	[CHAMF.]

By pushing right end soft-key, next soft-key page will be displayed.

[MODIFY]	[RECALC]	[GRAPH]	[NC CNV]	[STOP]

[LINE] : Select a LINE
[ARC ↻] : Select a CW arc
[ARC ↺] : Select a CCW arc
[CORNER] : Select a corner-R
[CHAMF.] : Select a chamfering
[MODIFY] : A data entering menu screen for each figure is displayed, and can be used for modifying a data that was previously entered.
[RECALC] : Calculation for whole turning figures is done again, and must be done after modifying a part of turning figures or entering new figure.
[GRAPH] : Figure drawing screen is displayed, and can be used for checking the entered figure. Enlarging, scale-down and others are available.
[NC CNV] : Convert entered turning figures into NC motion program.
After finishing conversion, a programming of turning figure will be finished and exit to former screen.
[STOP] : Stop a programming of turning figure and after replying the prompt of it, it can exit to former screen. NC program will be never made.

NOTE
During a programming of turning figure, only line / arc / chamfering / corner-R on an XZ-plane can be entered.

(3) Modify turning figures

There are two ways of modifying turning figure data that has already been fixed and stored.

(a) Method 1

Use a figure block data input screen

On the turning figure list screen, position the cursor to the figure block that is to be modified, and then press [MODIFY]. The figure block data input screen corresponding to the selected figure is displayed, allowing you to enter new data. Enter the required new data and then press [OK]. Then, press [RECALC] to calculate all of the contour figures by using the newly entered data.

(b) Method 2

Modify the data directly on the turning figure list screen

On the turning figure list screen, position the cursor to the figure block that is to be modified, enter a new value, and then press INPUT key.

Then, press [RECALC] to calculate all of the turning figures by using the newly entered data.

NOTE

To clear a data entry, press CAN key and then INPUT key.

(4) Insert a new figure block

Position the cursor to the figure block immediately before the position where a new figure should be inserted. Then, using the procedure described in examples 2 and 3, enter new figure block.
Then, press [RECALC] to calculate all of the contour figures using the newly entered data.

(5) Delete a figure block

Position the cursor to the head of the figure block or figure symbol to be deleted, and then press DELETE key. The prompt "ARE YOU SURE TO DELETE BLOCK?" will appear. Press [YES] to delete the figure. Press [NO] to abandon the deletion.
Then, press [RECALC] to calculate all of the contour turning figures using the newly entered data.

(6) Change a figure block

To change the figure type of a previously input contour figure, first delete the old figure block, and then insert a new figure block.

(7) Checking the turning figures

Entered turning figures can be checked on the screen by means of operations such as zooming-in, zooming-out, and so on.
On the turning figure list screen, press [GRAPH]. The graphic drawing screen shown below will appear. A drawing scale appears at the bottom of the screen.

[LARGE] [SMALL] [AUTO] [REAL] [RETURN]

Pressing the right-end soft-key causes the next soft-key page to appear.

[←] [→] [↑] [↓] [CENTER]

[LARGE] : Double a scale factor.
[SMALL] : Make a scale factor half.
[AUTO] : Decide a scale factor automatically in order to draw a whole part within a screen.
[REAL] : Draw turning figures in real scale.
[RETURN] : Return to the turning figure list screen.
[←] [→] [↑] [↓] : Move a view point to each direction. Cursor key can be used also to move.
[CENTER] : Move a drawing figure at the center of a screen.

(8) Output to NC program

Entered turning figures can be outputted to NC programs in the form of G-code.
Press [NC CNV]. The following screen appears.

```
CONFIRM CONVERSION OF NC PROGRAM

YOU CAN SELECT NC CONVERSION TYPE

PUSH [YES] WITH NO DATA INPUT
 → NC PROGRAM WILL BE ENTERED TO ACTUAL PROGRAM

ENTER SUB PROGRAM NUMBER, THEN PUSH [YES]
 → NC PROGRAM WILL BE ENTERED AS A SUB PROGRAM


        [ YES ]   [ NO ]   [      ]   [      ]   [      ]
```

By following the message displayed on the screen, press [YES] immediately or press it after entering a sub program number.
If you press [YES] without entering a sub program number, the contour figures will be entered immediately after the cursor before the start of Contour programming.
If you press [YES] after entering a new sub program number, the contour figure will be entered into the newly made sub program, and a sub-program calling block such as M98 Pxxxx will be entered immediately after the cursor before the start of Contour programming.
By pressing [NO], you can abandon the conversion operation.
Contour figures can be converted to the following G-code programs.

Type of Figure	Symbol	G-code
Start point	●	G1400
Line	→	G1401
Arc (CW)	↻	G1402
Arc (CCW)	↺	G1403
Corner R	R	G1402 or G1403
Chamfering	C	G1401

NOTE
1 Converted NC program blocks are stored immediately after the block to which the cursor was positioned.
After a return to these previous screens, the cursor will be positioned to the head of the NC program that was newly stored after conversion.
2 If there are any figures for which the end points are pending, NC program conversion will be done for the contour figure immediately before the pending block.
3 A value of up to eight digits can be entered for the axis command (X/Z) of a converted NC program, and the value must always have a decimal point. The decimal digits conform to the minimum setting units, with any lower digits being rounded off.

(1) IS-B form

	Least input increment	Least command increment	Max.
mm	0.001 mm	0.001 mm	±99999.999 mm
inch	0.0001 inch	0.0001 inch	±9999.9999 inch

NOTE
4 By setting bit 0 (IKR) of parameter No.9341 to 1, address "R" is output as the radius data of arc n. By setting it to 0, data "I" and "K" are output as center point coordinates.
5 Even if data having the same value is output continuously for a given address, they will never be canceled.

4.1.4 Input Method of Figure Block

This chapter details figure block data to be entered on the figure block data input screen.
The contour figure data of a start point and straight line is described in detail in the previous chapter. See the previous chapter if necessary.

(1) START POINT

Start : G1400

Data item		Comment
X	START POINT (X-AXIS)	X-axis coordinate of figure start point
Z	START POINT (Z-AXIS)	Z-axis coordinate of figure start point

The following soft keys are displayed.
[AUX.] : Call auxiliary calculating screen. The result of it will be set into start point coordinate data.
[OK] : Fix start point entering and store into memory.
[EXIT] : Cancel start point entering and exit from programming of turning figure.

(2) LINE

Line : G1401

If you select a line, the line screen is displayed, allowing you to enter the figure data written on a drawing.
Even if the end point is not input, it can be determined by calculating the cross point coordinate between this figure and that to be entered next.

Data item	Comment
END POINT X	X coordinate of a line end point
END POINT Z	Z coordinate of a line end point
ANGLE A	Angle of line from +Z-axis. A positive angle is counter clockwise direction.
TOUCH STATE	Select from the following soft-key menu whether tangential or not between neighbor figure. [NO] : Not tangential [LAST] :Tangential with the preceding figure

[AUX.] : Call auxiliary calculating screen. The result of it will be set into end point coordinate or angle data.
[OK] : Fix line figure entering and store into memory.
[CANCEL] : Cancel line figure entering and return to the turning figure list screen.

(3) ARC

Arc CW : G1402
Arc CCW : G1403

Data item	Comment
END POINT X	X coordinate of an arc end point
END POINT Z	Z coordinate of an arc end point
RADIUS R	Radius of an arc, but plus value only

Data item	Comment
CENTER I	X coordinate of an arc center
CENTER K	Z coordinate of an arc center
TOUCH STATE	Select from the following soft-key menu whether tangential or not between neighbor figure. [NO] : Not tangential [LAST] :Tangential with the preceding figure

NOTE
When entering the coordinates of an arc end point, do not round off the calculated value.

The following soft keys are displayed.
[AUX.] : Call auxiliary calculating screen. The result of it will be set into start point coordinate data.
[OK] : Fix arc figure entering and store into memory.
[CANCEL] : Cancel arc figure entering and return to the turning figure list screen.

(4) CORNER R

Corner R CW : G1402
Corner R CCW : G1403

Data item	Comment
RADIUS R	Radius of a corner R, but plus value only

The following soft keys are displayed.
[OK] : Fix corner R figure entering and store into memory.
[CANCEL] : Cancel corner R figure entering and return to the turning figure list screen.

(5) CHAMFERING

Chamfering : G1401

Data item	Comment
CAMFER C	Chamfering amount, but plus value only

The following soft keys are displayed.
[OK] : Fix chamfering figure entering and store into memory.
[CANCEL] : Cancel chamfering figure entering and return to the turning figure list screen.

(6) SELECT CROSSING POINT OF FIGURE

During the calculation of a contour figure, such as between a line and an arc for example, there may be cases in which two or more cross points or figures are possible. In this case, the screen for selecting a cross point or figure appears.

The following soft keys are displayed.
[PREV.]/[NEXT] : Cross point or figure should be selected is changed. The active one blinks among some figures should be selected.
[OK] : Select an active figure, blinking, finally.
[CANCEL] : Cancel a selecting operation. And then, store the actual figure into memory still keeping it in pending.

(7) FIGURE END

Figure End: G1406

No data is necessary in this figure block. And it's not necessary for an operator to program figure end because figure end block is created automatically when figure data is outputted to NC.

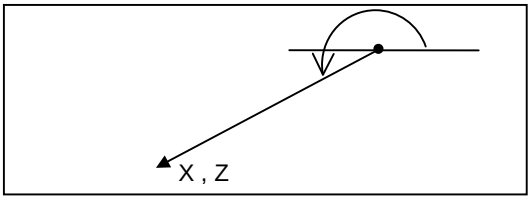
4.1.5 Automatic Calculation of Figure Block

This chapter explains the details of automatic calculations of figure block, such as those for cross points or tangential points, that is supported by contour programming.
A figure block for which an end point has not yet been determined is said to be in the pending state. A pending figure block is indicated by a dotted line.
On the screen for entering figure block, more data input items than required will appear. These data items are used to calculate the cross points with the immediately preceding pending figure block, and also to calculate the end point.
Ten successive figure blocks can be specified as pending blocks.

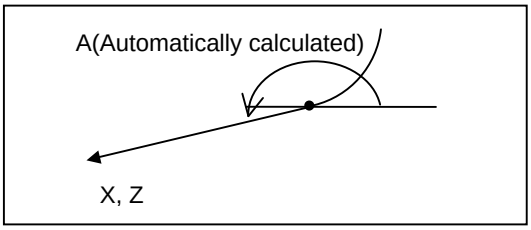
NOTE
It is impossible to enter successive straight lines with only their angles specified. (They are not assumed to be pending blocks.)

4.1.5.1 LINE

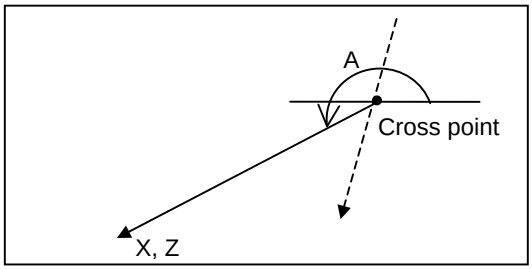
(1) When the preceding figure is not pending
  (a) Only X is inputted
    → This line is determined as a vertical line.
  (b) Only Z is inputted
    → This line is determined as a horizontal line.
  (c) A and either X or Z are inputted
    → The end point that is not inputted is calculated.



(2) When the preceding figure specifying an arc is not pending, and "TOUCH LAST" is specified in the line.
  (a) Either X or Z is inputted
    → The angle A is calculated automatically, and an end point is determined.
    If neither X nor Z is inputted, this line will be pending.

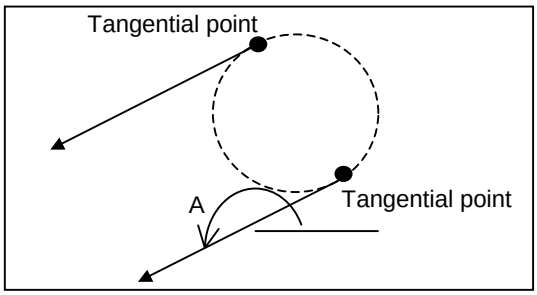


(3) When the preceding figure is pending, and "TOUCH LAST" is specified in the line.
(a) Both X and Z, and A are inputted
→ The cross point between the preceding figure is calculated.

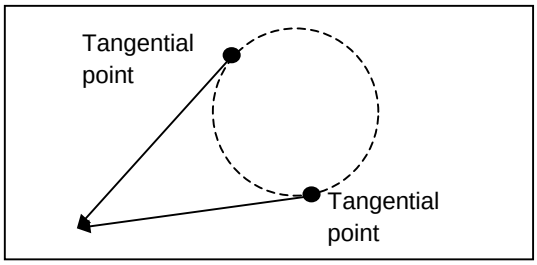


When the preceding figure is an arc, the cross point selection screen is displayed, so select a necessary one.

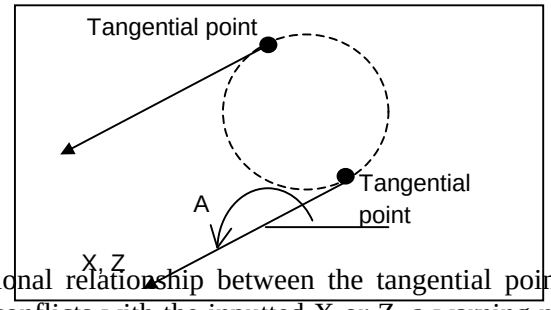
(4) When the preceding figure is pending arc, and "TOUCH LAST" is specified in the line.
It is assumed that the radius and the center point coordinate (I,K) of arc have already inputted.
(a) Only A is inputted
→ The tangential point selection screen is displayed, so select a necessary one.
This line will be pending.



(b) Both X and Z are inputted
→ The tangential point selection screen is displayed, so select a necessary one.
This line will be determined.



(c) A and either X or Z are inputted
→ The tangential point selection screen is displayed, so select a necessary one.
This line will be determined.



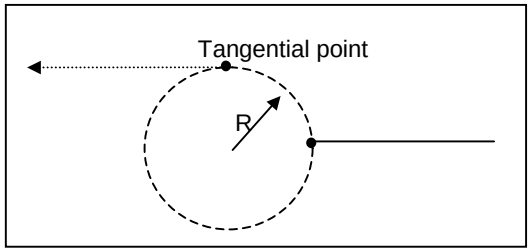
If the positional relationship between the tangential point and the line is such that an inputted A conflicts with the inputted X or Z, a warning message is displayed to indicate that invalid data has been inputted.

4.1.5.2 ARC

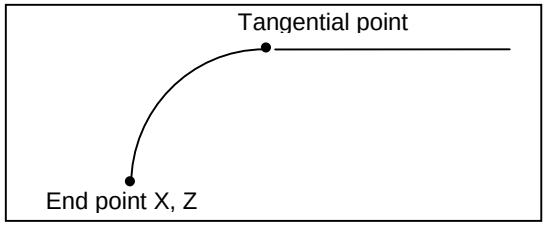
(1) When the preceding figure is not pending, and "TOUCH LAST" is not specified in the arc.
(a) I and K are inputted.
→ This arc will be pending.
(b) X, Z, and R are inputted.
→ A short path arc is uniquely defined.
(c) X, Z, I, and K are inputted.
→ A short path arc is uniquely defined.

NOTE
If the distance (radius) between the start point and center differs from the end point and center, the figure is displayed based on the actual form, and the actual figure will not be machined correctly.

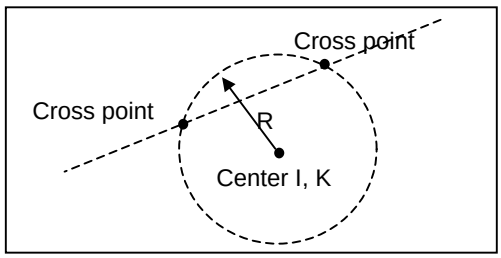
(d) Only R is inputted
→ By specifying "TOUCH LAST" and inputting a line with A=0 degree and X coordinate as an immediately after figure, this arc can be determined.



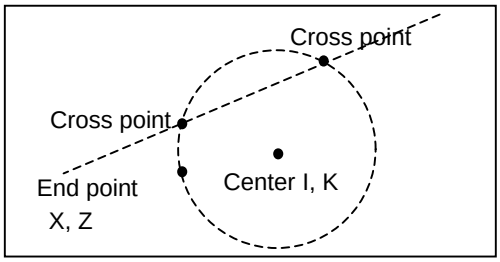
(2) When the preceding figure is not pending, and "TOUCH LAST" is specified in the arc.
(a) X and Z are inputted.
→ The radius is automatically calculated and this arc will be determined.



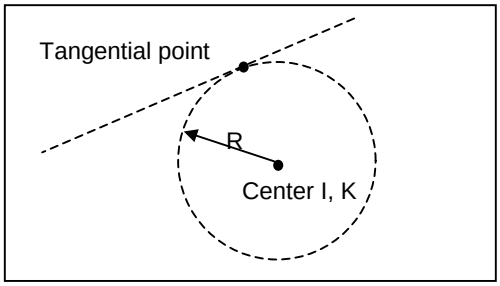
(3) When the preceding figure is pending (foe which the start point has been determined), and "TOUCH LAST" is not specified in this arc
(a) R, I, and K are inputted.
→ The cross point selection screen is displayed, so select a necessary one. This arc will be pending.



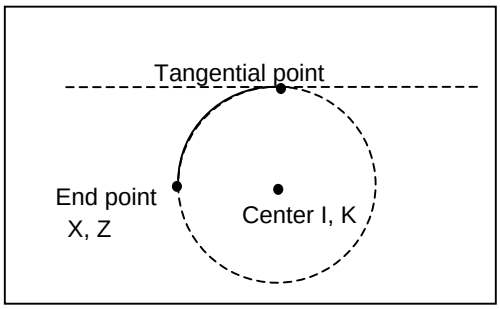
(b) X, Z, I, and K are inputted.
→ The cross point selection screen is displayed, so select a necessary one. This arc will be determined.



(4) When the preceding figure is pending (for which the start point has been determined), and "TOUCH LAST" is specified in the arc
(a) R, I, and K are inputted.
→ The tangential point is calculated, and this arc will be pending.

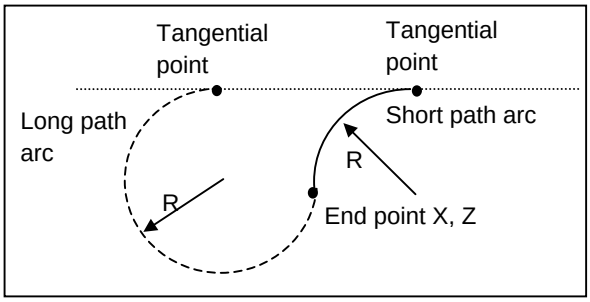


(b) X, Z, I , and K are inputted.
→ The tangential point is calculated, and this arc will be determined.



NOTE
If the distance (radius) between the start point and center differs from the end point and center, the figure is displayed based on the actual form, and the actual figure will not be machined correctly.

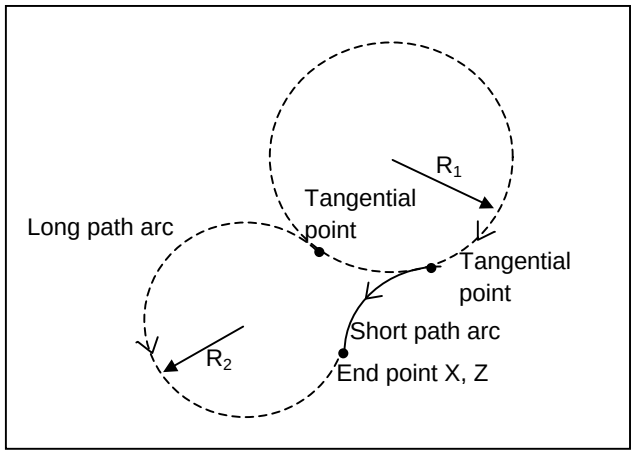
(c) R, X, and Z are inputted.
→ The tangential point is calculated and a short path arc is uniquely defined.



(5) When the preceding figure "arc" is pending (for which the start point has been determined and only R is to be inputted), and "TOUCH LAST" is specified in the arc.

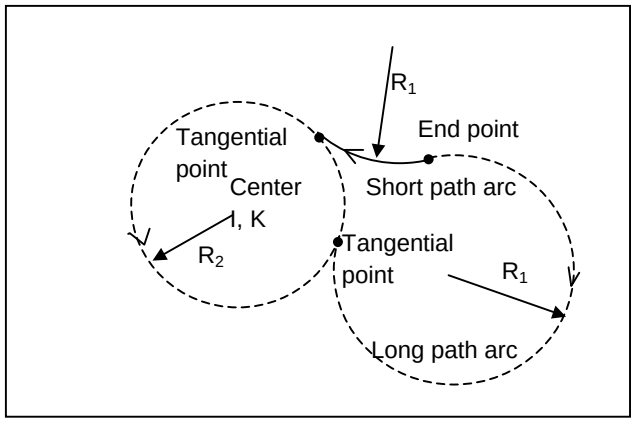
(a) R, X, and Z are inputted.

→ The tangential point is calculated and a short path arc is uniquely defined.

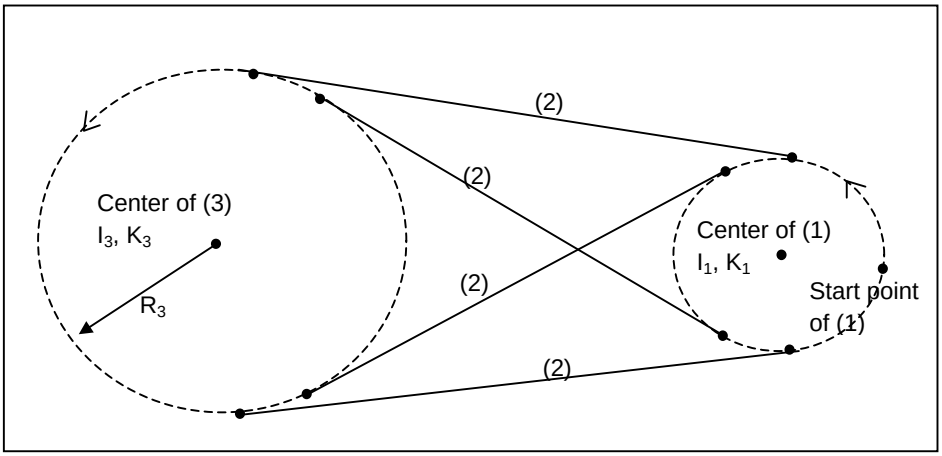


(b) R, X, and Z are inputted.

→ The tangential point is calculated and a short path arc is uniquely defined.



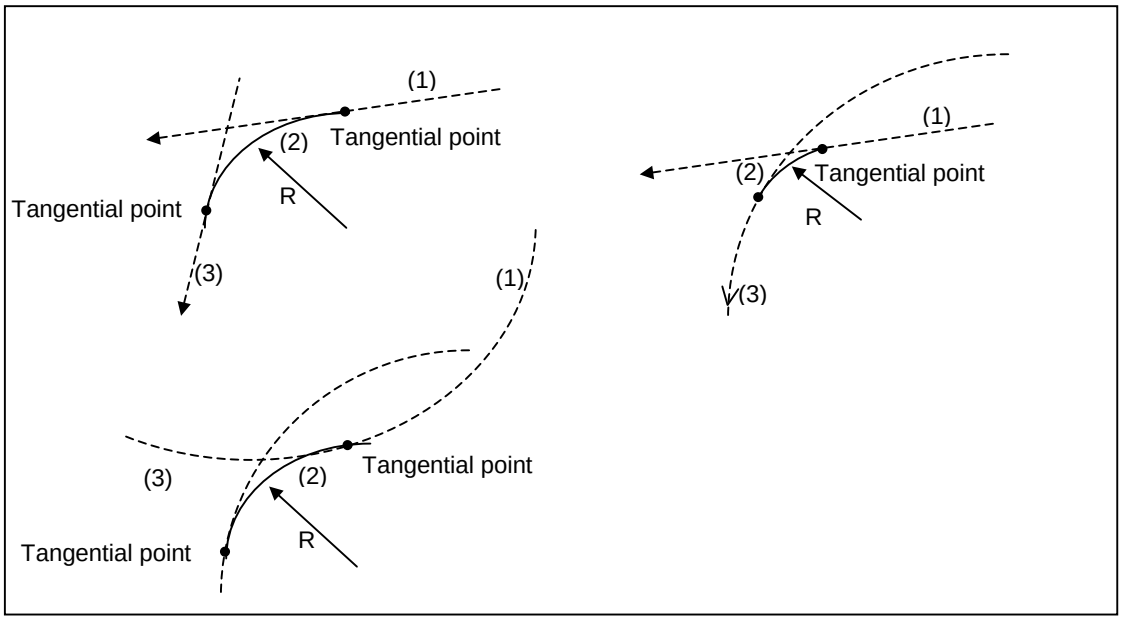
4.1.5.3 LINE TANGENTIAL TO TWO ARCS



By inputting three successive figures as follows, line (2) that is tangential to two arcs can be specified as shown in the above drawing. The end points of (1) and (2) are determined, while (3) is left pending. Among the above four possible lines, depending on the direction of the two arcs, the line that makes the smoothest connection to the arcs is automatically selected.

Arc (1) :
I and K are inputted. (A start point is determined. This arc is pending.)
Line (2) :
Only "TOUCH LAST" is inputted.
Arc (3) :
R, I and K are inputted.

4.1.5.4 ARC THAT CONTACTS TO BOTH CROSSING LINES AND ARCS

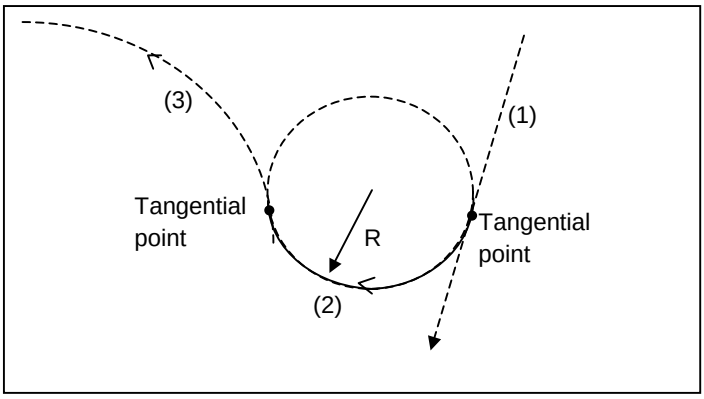


By inputting three successive figures as follows, arc (2) that is tangential to two lines or arcs can be specified as shown in the above drawing. The end points of (1) and (2) are determined, while (3) is left pending. When (3) is a line, it is determined.

Line (1) or Arc (1) :
Line that is pending (for which A is inputted and the start point has been determined), or Arc that is pending (for which I and K are inputted and the start point has been determined)
Arc (2) :
R and "TOUCH LAST" are inputted.
Line (3) or Arc (3) :
Line with A, X and, or Arc with R, I, K and "TOUCH LAST"

When either figure (1) or (3) is an arc, or when both of them are arcs, the selection screen for multiple arcs is displayed, so select a suitable one.

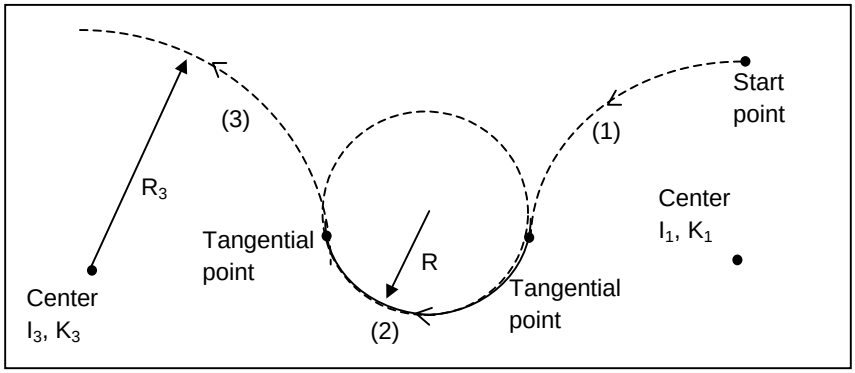
4.1.5.5 ARC THAT CONTACTS TO UNCROSSING LINE AND ARC



By inputting three successive figures as follows, arc (2) that is tangential to line (1) and arc (3) that do not cross, can be specified as shown in the above drawing. The end points of (1) and (2) are determined, while (3) is left pending.
Among the multiple possible arcs shown above, the arc that makes the smoothest connection to line (1) and arc (3) is automatically selected. But, even though a "Long path arc" and "Short path arc" remain, select whichever is necessary from the selection screen.

Line (1) :
Line that is pending (for which A is inputted and the start point has been determined)
Arc (2) :
R and "TOUCH LAST" are inputted.
Arc (3) :
Arc with R, I and K

4.1.5.6 ARC THAT CONTACTS TO UNCROSSING 2 ARCS



By inputting three successive figures as follows, arc (2) that is tangential to arcs (1) and (3) that do not cross can be specified as shown in the above drawing.
The end points of (1) and (2) are determined, while (3) is left pending.
Among the multiple possible arcs shown above, the arc that makes the smoothest connection to arcs (1) and (3) is automatically selected. But, even though a "Long path arc" and "Short path arc" remain, select whichever is necessary from the selection screen.

Arc (1) :
Arc with I and K, and it is pending (for which the start point has been determined)
Arc (2) :
R and "TOUCH LAST" are inputted.
Arc (3) :
Arc with R, I, and K

4.1.6 Auxiliary Calculation

This chapter explains the details of the auxiliary calculation.
By using this auxiliary calculation, the coordinates of a point or the angle of a line can be determined. Furthermore, the figure block, such as the amount of shift from the original form for a tool radius can be entered easily.

(1) Data items in which auxiliary calculation can be used
  (a) Start point
    - Coordinate (X, Z) of start point
  (b) Line
    - Coordinate (X, Z) of end point
    - Angle of a line (A)
  (c) Arc
    - Coordinate (X, Z) of end point
    - Coordinate (I, K) of center
    - Specifying an arc

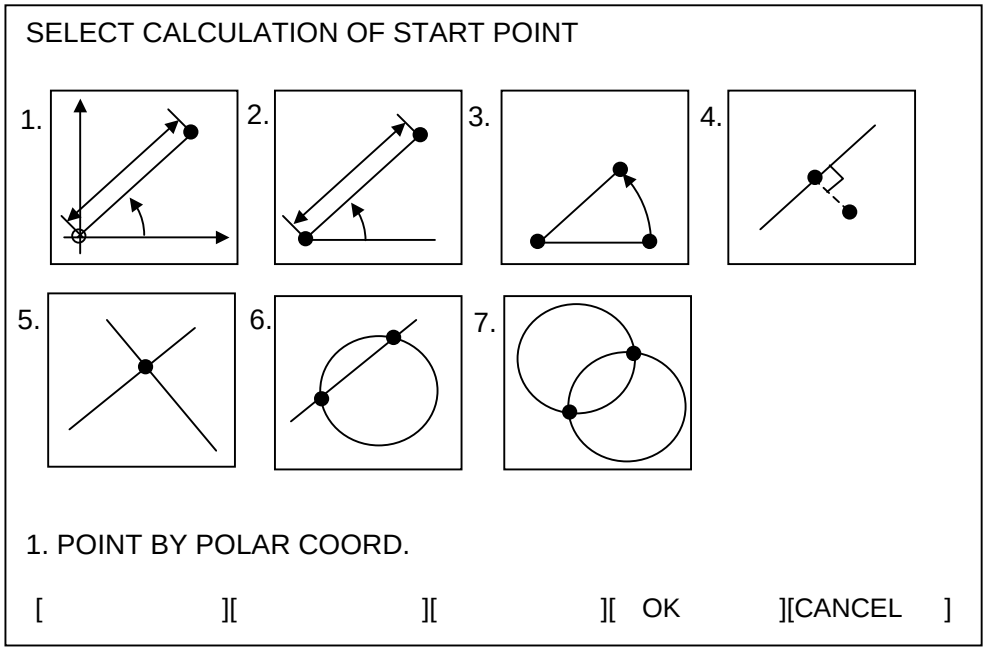
(2) Calculation type available in auxiliary calculation
  (a) Calculation of coordinate
    - A point specified by polar coordinate
    - A point specified by a point, angle and distance
    - A point specified by rotating a point
    - Neighbor point of a line
    - Cross point between 2 lines
    - Cross point between line and arc
    - Cross point between 2 arcs
  (b) Calculation of angle
    - Angle of the line passes 2 points
    - Angle of a line rectangular to the line passes 2 points
  (c) Specify an arc (Center and radius)
    - An arc passes 1 point and its center coordinate has been determined
    - An arc passes 2 points and its radius has been determined
    - An arc passes 3 points

4.1.6.1 START POINT

(1) Selecting type of calculation

On the data-entry screen for a start point, press [AUX.]. The following calculation type menu screen will appear.

By pressing the cursor key, scroll through the menu comments until you find the one to select.



The following soft keys are displayed.

[OK] : Use the active calculation type

[CANCEL] : Cancel selecting operations and return to the previous screen

NOTE

When bit 5 (AUX) of parameter No.9305 is set to 1, the menu for auxiliary calculation is displayed as shown above. By setting it to 0, the menu is displayed as a list of comments for each calculation type.

This parameter is available for other auxiliary calculation menus.

(2) Entering data for calculation

(a) A point specified by polar coordinate

Data item	Comment
DIST. D	Distance between the point and work coordinate origin
ANGLE A	Angle of line from +Z-axis. A positive angle is counter clockwise direction.

(b) A point specified by a point, angle and distance

Data item	Comment
POINT X	X coordinate of the base point
POINT Z	Z coordinate of the base point
DIST. D	Distance between the point and the base point
ANGLE A	Angle of line from +Z-axis. A positive angle is counter clockwise direction.

(c) A point specified by rotating a point

Data item	Comment
POINT X	X coordinate of the base point
POINT Z	Z coordinate of the base point
CENTER I	X coordinate of the rotating center
CENTER K	Z coordinate of the rotating center
ANGLE A	Angle of line from +Z-axis. A positive angle is counter clockwise direction.

(d) Neighbor point of a line

You can determine the nearest point to the line. Furthermore, you can determine the nearest point to the line that has been moved through a specific distance.
This can be useful for determining the end point of an approaching motion from a certain point close to the line.

(1) When specifying a line with one point and an angle

Data item	Comment
BASE POINT X	X coordinate of the base point positioned apart from a line
BASE POINT Z	Z coordinate of the base point positioned apart from a line
PASS POINT U	X coordinate of a certain point on the line
PASS POINT W	Z coordinate of a certain point on the line
ANGLE A	Angle of line from +X or +Z-axis. A positive angle is counter clockwise direction.
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

(2) When specifying a line with two points
By pressing soft key [XZ,XZ], you can specify a line with two passing points.
By pressing soft key [XZ, A], you can select the above type by using an angle again.

Data item	Comment
BASE POINT X	X coordinate of the base point positioned apart from a line
BASE POINT Z	Z coordinate of the base point positioned apart from a line
PASS POINT U	X coordinate of the 1st passing point on the line
PASS POINT W	Z coordinate of the 1st passing point on the line
PASS POINT P	X coordinate of the 2nd passing point on the line
PASS POINT Q	Z coordinate of the 2nd passing point on the line
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting distance
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

(e) Cross point between 2 lines

The cross point of two lines can be calculated. Furthermore, this calculation can also be done for a line that is shifted from its original position by some distance.
This can be useful when entering a tool path that is shifted from the original figure by an amount equal to the tool radius.

(1) When specifying a line with one point and an angle
Initially, the following data items are displayed for Line-1.

Data item	Comment
PASS POINT X	X coordinate of a certain point on the line
BASE POINT Z	Z coordinate of a certain point on the line
ANGLE A	Angle of line from +X or +Z-axis. A positive angle is counter clockwise direction.
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting Distance
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

By pressing [NEXT] after entering the necessary data, a similar data item screen for Line-2 is displayed. By pressing [PREV.], you can return to the previous screen for Line-1.

(2) When specifying a line with two points
By pressing soft key [XZ,XZ], you can specify a line with two passing points.
By pressing soft key [XZ, A], you can select the above type by using an angle again.
The following data items are displayed respectively for Line-1 and Line-2.

Data item	Comment
PASS POINT X	X coordinate of the 1st passing point on the line
PASS POINT Z	Z coordinate of the 1st passing point on the line
PASS POINT U	X coordinate of the 2nd passing point on the line
PASS POINT W	Z coordinate of the 2nd passing point on the line
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

(f) Cross point between line and arc

The cross point between a line and arc can be calculated. For the line, one that has been shifted through some distance can be used for the calculation.
This is useful for entering a tool path that has been shifted from the original figure by an amount equal to the tool radius.

(1) When specifying a line with one point and an angle
Initially, the following data items are displayed for the Line.

Data item	Comment
PASS POINT X	X coordinate of the 1st passing point on the line
PASS POINT Z	Z coordinate of the 1st passing point on the line
ANGLE A	Angle of line from +X or +Z-axis. A positive angle is counter clockwise direction.
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

By pressing [NEXT] after entering the necessary data, the following screen for the Arc is displayed.
By pressing [PREV.], you can return to the previous screen for the Line.
By pressing [NEXT] after entering the necessary data, a similar data item screen for Line-2 is displayed. By pressing [PREV.], you can return to the previous screen for Line-1.

Data item	Comment
CENTER I	X coordinate of an arc center
CENTER K	Z coordinate of an arc center
RADIUS R	Radius of an arc, but plus value only
SELECT	Select necessary cross point from two possible points by arrow soft-keys

(2) When specifying a line with two points
By pressing [XZ,XZ], you can specify a line with two passing points.
By pressing [XZ, A], you can select the above type using an angle again.

Data item	Comment
PASS POINT X	X coordinate of the 1st passing point on the line
PASS POINT Z	Z coordinate of the 1st passing point on the line
PASS POINT U	X coordinate of the 2nd passing point on the line
PASS POINT W	Z coordinate of the 2nd passing point on the line
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting distance
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

Data screen for the arc is displayed quite similar to the case of (1).

(g) Cross point between 2 arcs

On the screen as shown below, data for two arcs can be entered and the cross point between them can be calculated.

Data item	Comment
CENTER X1	X coordinate of an arc-1 center
CENTER Z1	Z coordinate of an arc-1 center
RADIUS R1	Radius of an arc-1, but plus value only
CENTER X2	X coordinate of an arc-2 center
CENTER Z2	Z coordinate of an arc-2 center
RADIUS R2	Radius of an arc-2, but plus value only
SELECT	Select necessary cross point from two possible points by arrow soft-keys

(3) Execution of auxiliary calculation

After entering all of the necessary data for each of the above calculation types, press [OK]. The auxiliary calculation is done, and then the result is entered into the coordinate data item (X, Z) of the start point.
By pressing [CANCEL], you can return to the auxiliary calculation menu screen.

4.1.6.2 LINE

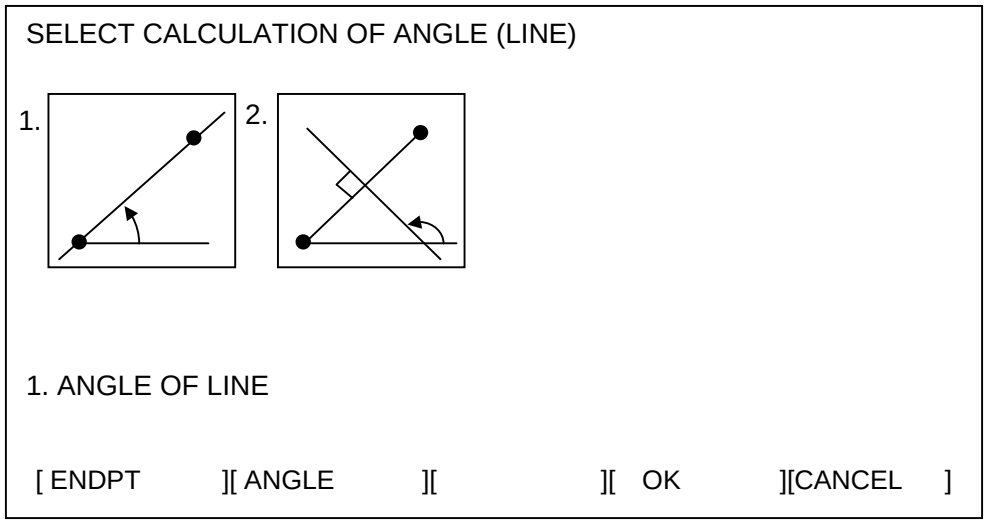
As part of the auxiliary calculation for a line, the end point coordinate and angle can be calculated.
The following soft-keys are displayed on the auxiliary calculation menu screen.

[ENDPNT] : Menu screen for end point calculation is displayed
[ANGLE] : Menu screen for angle calculation is displayed

For the end point calculation, these items are similar to those for the start point, so refer to the preceding explanation.

(1) Selecting type of calculation

On the menu screen for the auxiliary calculation of a line, press [ANGLE]. The following menu screen for angle calculation appears.
By pressing the cursor key, you can scroll through the menu comments until you find the type that you want to select.



(2) Entering data for calculation

(a) Angle of the line passes 2 points

Data item	Comment
POINT X	X coordinate of the 1st passing point on the line

Data item	Comment
POINT Z	Z coordinate of the 1st passing point on the line
POINT U	X coordinate of the 2nd passing point on the line, this should be apart from the 1st point
POINT W	Z coordinate of the 2nd passing point on the line, this should be apart from the 1st point

(b) Angle of a line rectangular to the line passes 2 points

The angle of a line that is rectangular to a line and which passes through two points can be calculated.

Data item	Comment
POINT X	X coordinate of the 1st passing point on the line
POINT Z	Z coordinate of the 1st passing point on the line
POINT U	X coordinate of the 2nd passing point on the line, this should be apart from the 1st point
POINT W	Z coordinate of the 2nd passing point on the line, this should be apart from the 1st point

(3) Execution of auxiliary calculation

After entering all of the necessary data for each of the above calculation types, press [OK]. The auxiliary calculation is performed, after which the result is entered into the end point coordinate (X, Z) or angle (A) of the line.
By pressing [CANCEL], you can return to the auxiliary calculation menu screen.

4.1.6.3 ARC

As part of the auxiliary calculation for an arc, the end point coordinate and center coordinate can be calculated. Furthermore, the arc itself can be specified by entering the likes of three passing points.
The following soft-keys are displayed on the auxiliary calculation menu screen.

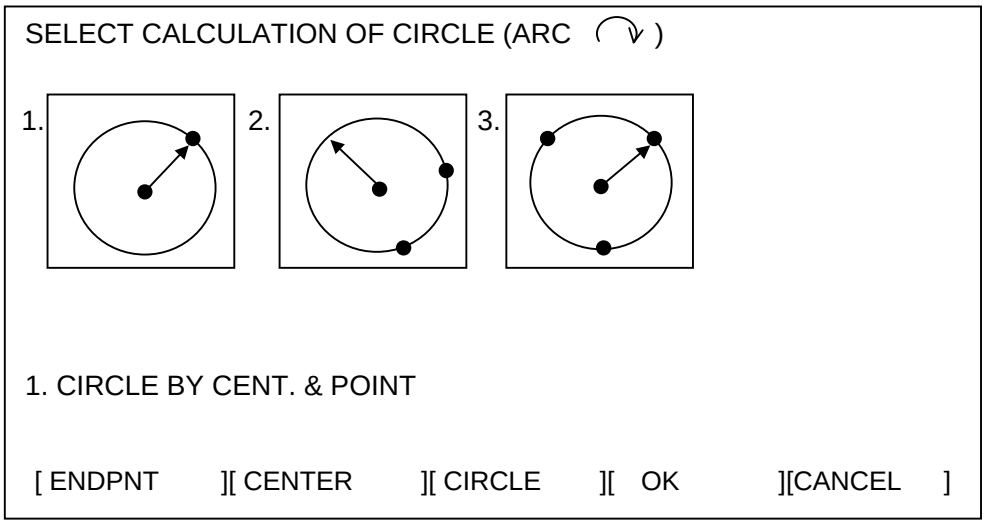
[ENDPNT] : Menu screen for end point calculation is displayed
[CENTER] : Menu screen for center point calculation is displayed
[CIRCLE] : Menu screen for circle specifying is displayed

For the end point and center point calculation, they are quite similar to those for the start point case, so refer to the preceding explanation.

(1) Selecting type of calculation

On the menu screen for auxiliary calculation of an arc, press [CIRCLE]. The following menu screen for specifying a circle is displayed.

By pressing the cursor keys, you can scroll through the menu items until you find the item that you want to select. The currently selected item is highlighted in yellow.



(2) Entering data for calculation

(a) An arc passes 1 point and its center coordinate has been determined

Data item	Comment
POINT X	X coordinate of a certain point on the arc
POINT Z	Z coordinate of a certain point on the arc
CENTER I	X coordinate of an arc center
CENTER K	Z coordinate of an arc center

(b) An arc passes 2 points and its radius has been determined

Data item	Comment
POINT X	X coordinate of the 1st passing point on the arc
POINT Z	Z coordinate of the 1st passing point on the arc
POINT U	X coordinate of the 2nd passing point on the arc, this should be apart from other points
POINT W	Z coordinate of the 2nd passing point on the arc, this should be apart from other points
RADIUS R	Radius of the arc, but plus value only
SELECT	Select necessary one from two possible arcs by arrow soft-keys

(c) An arc passes 3 points

Data item	Comment
POINT X	X coordinate of the 1st passing point on the arc
POINT Z	Z coordinate of the 1st passing point on the arc
POINT U	X coordinate of the 2nd passing point on the arc, this should be apart from other points
POINT W	Z coordinate of the 2nd passing point on the arc, this should be apart from other points
POINT P	X coordinate of the 3rd passing point on the arc, this should be apart from other points
POINT Q	Z coordinate of the 3rd passing point on the arc, this should be apart from other points

(3) Execution of auxiliary calculation

After entering all the necessary data for each of the above calculation types, press [OK]. The auxiliary calculation is performed, after which the result is entered into the end point coordinate (X, Z) or center point coordinate (I, K) of an arc.
When circle specification is selected by [CIRCLE], the radius and center point coordinate are calculated and the results are entered into these data items.

By pressing [CANCEL], you can return to the auxiliary calculation menu screen.

4.1.6.4 OTHERS

(1) Calculation of Inputting Data

When entering data on the figure block data input screen, calculation as shown below can be made as in a desktop calculator.

Addition :

10+10 [INPUT] → 20

Subtraction :

10-10 [INPUT] → 0

Multiplication :

10*10 [INPUT] → 100

Division :

10/10 [INPUT] → 1

SIN :

S30 [INPUT] → 0.5

COS :

C60 [INPUT] → 0.5

TAN :

T45 [INPUT] → 1

Square root :

R9 [INPUT] → 3

NOTE
1 For + - * /, more than two terms cannot be handled at once. The third and subsequent terms are neglected. So, 1 + 2 + 3 will be calculated as 1 +2.
2 The calculation of SIN, COS, TAN and Square root can be done as only independent calculations. The second and subsequent terms, if specified, will be neglected. C60 + S60 will be calculated as C60 only.
Calculation based on previously input data is also possible.

(2) Notes

NOTE
1 No more than thirty figure blocks can be creating for turning figure.
2 During creating of turning figure, if the screen is changed to another ones by the user pressing the likes of a function button, the active screen displayed in creating screen of turning figure will be forced to close.
3 When editing the turning figures, keep the following in mind.
(1) The turning figures that was operated one or more times cannot be edited.
(2) After creating the turning figures, another turning figure cannot be newly entered.

4.2 "INPUTING DIRECTLY END POINT" TYPE

In this programming method, an operator inputs directly the end point of figure.

4.2.1 Start-up Method

Press soft key [2] through [6] on the page 2/2 of "Bar Machining Figure Menu" screen, select data input screen for defining figure.

[2] [3] [4] [5] [6]

[2] : Select Start Point data input screen
[3] : Select Line data input screen
[4] : Select Arc (CW) data input screen
[5] : Select Arc (CCW) data input screen
[6] : Select Figure End data input screen

4.2.2 Method of Creating Figure

(1) On the figure data input screen, enter the figure data such as end point and radius.
(2) After inputting data, press the soft key [OK].
Refer to section "3.2.2 Figure Block" about the details of each figure data.
(3) After completing enter figure data, press the soft key [6] for displaying Figure End screen.
(4) Press the soft key [OK] without inputting data.

Figure block has to be specified as follows.
In beginning block of figure, G1400(Start Point) has to be specified. In last block of figure, 1406(Figure End) has to be specified.

V. CONTOUR PROGRAMMING

1 OPERATING METHOD

Follow the operating method below to create a contour program.

(1) Calling the contour programming screen
(2) Selecting the method of editing a contour program
(3) Creating a contour program
(4) Checking a contour program
(5) Conversion to NC programs

These operations are described below.

Chapter 1, "OPERATING METHOD", consists of the following sections:

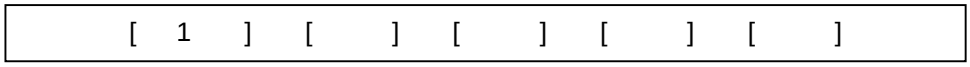
1.1 CALLING CONTOUR PROGRAMMING SCREEN

(1) When a program is created with G01/G02/G03

Press soft key [CONTUR] on page 2 of the main menu screen.

(2) When the figure block of a turning cycle is entered

Press soft key [1] (contour figure) on the turning cycle figure menu screen.



In both cases, the contour program initial screen will be displayed.

NOTE

When you are going to enter figure blocks for the Stock removal cycle, you can either enter them directly immediately after the machining type block, or enter them as a sub program.
The latter is useful when the same contour figure is used both for roughing and finishing. First, you can enter the figure blocks as the sub program for roughing, and then you need only enter a sub program that calls a block such as M98 Pxxxx.
For details on the operation, refer to Section 1.5 “CONVERT TO NC PROGRAM”.

1.2 SELECTING OF METHOD TO EDIT OF CONTOUR PROGRAM

Pressing soft key [CONTUR] (or soft key [1]) causes the initial screen for contour programming to be displayed.
After the contour programming operation screen, the next screen appears to allow the user to select whether a new program is to be created, or whether an existing one is to be edited.

When you want to create a new program, press soft key [NEW]. To work with an existing program, press soft key [EDIT].

```
SELECT METHOD TO EDIT CONTOUR PROGRAM

  CONTOUR PROGRAM IS EXISTING
  [NEW]:    CREATE NEW PROGRAM.
  [EDIT]:   EDIT EXISTED PROGRAM.



        [ NEW  ]    [ EDIT ]    [      ]    [      ]    [      ]
```

NOTE
In contour programming, a temporary working program must be allocated to hold a temporary NC program. A program number of this temporary program should be set in parameter No.9330 using a non-zero value. In this manual, all explanations assume that 9999 is set.

1.3 METHOD OF CREATING A CONTOUR PROGRAM

(1) Start point

As the data item screen for the start point will be displayed, enter the start point.

(2) Select type of contour figure

During contour programming, the following contour figure selecting or other operating soft-key will be displayed. In this screen operations for contour figure entering can be done.

```
[ LINE ]  [ARC ↻]  [ARC ↺]  [CORNER]  [CHAMF.]
```

By pushing right end soft-key, next soft-key page will be displayed.

```
[MODIFY]  [RECALC]  [GRAPH ]  [NC CNV]  [ STOP ]
```

[LINE] : Select a LINE
[ARC ↻] : Select a CW arc
[ARC ↺] : Select a CCW arc
[CORNER] : Select a corner-R
[CHAMF.] : Select a chamfering

[MODIFY] : A data entering menu screen for each figure is displayed, and can be used for modifying a data that was previously entered.
[RECALC] : Calculation for whole contour figures is done again, and must be done after modifying a part of contour figures or entering new figure.
[GRAPH] : Figure drawing screen is displayed, and can be used for checking the entered figure. Enlarging, scale-down and others are available.
[NC CNV] : Convert entered contour figures into NC motion program.
After finishing conversion, a contour programming will be finished and exit to former screen.
[STOP] : Stop a contour programming and after replying the prompt of it, it can exit to former screen. NC program will be never made.

NOTE
During a contour programming, only line / arc / chamfering / corner-R on an XY-plane or XZ-plane can be entered.

(3) Modify contour figures

There are two ways of modifying contour figure data that has already been fixed and stored.

(a) Method 1

Use a contour figure data screen
On the contour program list screen, position the cursor to the
figure block that is to be modified, and then press [MODIFY]. The figure data screen corresponding to the selected figure is displayed, allowing you to enter new data. Enter the required new data and then press [OK]. Then, press [RECALC] to calculate all of the contour figures by using the newly entered data.

(b) Method 2

Modify the data directly on a contour program list screen
On the contour program list screen, position the cursor to the data that is to be modified, enter a new value, and then press 

key.
Then, press [RECALC] to calculate all of the contour figures by using the newly entered data.

NOTE
To clear a data entry, press 

key and then 

key.

(4) Insert a new contour figure

Position the cursor to the figure block immediately before the position where a new figure should be inserted. Then, using the procedure described in examples 2 and 3, enter new figure block.
Then, press [RECALC] to calculate all of the contour figures using the newly entered data.

(5) Delete a contour figure

Position the cursor to the head of the figure block or figure symbol to be deleted, and then press 

key. The prompt "ARE YOU SURE TO DELETE BLOCK?" will appear. Press [YES] to delete the figure. Press [NO] to abandon the deletion.
Then, press [RECALC] to calculate all of the contour figures using the newly entered data.

(6) Change a contour figure

To change the figure type of a previously input contour figure, first delete the old figure block, and then insert a new figure block.

1.4 CHECKING CONTOUR PROGRAMS

Entered contour figures can be checked on the screen by means of operations such as zooming-in, zooming-out, and so on.
On the program list screen, press [GRAPH]. The graphic drawing screen shown below will appear. A drawing scale appears at the bottom of the screen.

[LARGE] [SMALL] [AUTO] [REAL] [RETURN]

Pressing the right-end soft-key causes the next soft-key page to appear.

[←] [→] [↑] [↓] [CENTER]

[LARGE] : Double a scale factor.
[SMALL] : Make a scale factor half.
[AUTO] : Decide a scale factor automatically in order to draw a whole part within a screen.
[REAL] : Draw contour figures in real scale.
[RETURN] : Return to a contour program list screen.
[←] [→] [↑] [↓] : Move a view point to each direction. Cursor key can be used also to move.
[CENTER] : Move a drawing figure at the center of a screen.

1.5 CONVERT TO NC PROGRAM

Entered contour figures can be converted to NC programs in the form of G-code.
Press [NC CNV]. The following screen appears.

CONFIRM CONVERSION OF NC PROGRAM

YOU CAN SELECT NC CONVERSION TYPE

PUSH [YES] WITH NO DATA INPUT
→ NC PROGRAM WILL BE ENTERED TO ACTUAL PROGRAM

ENTER SUB PROGRAM NUMBER, THEN PUSH [YES]
→ NC PROGRAM WILL BE ENTERED AS A SUB PROGRAM

[YES] [NO] [ ] [ ] [ ]

By following the message displayed on the screen, press [YES] immediately or press it after entering a sub program number.
If you press [YES] without entering a sub program number, the contour figures will be entered immediately after the cursor before the start of Contour programming.
If you press [YES] after entering a new sub program number, the contour figure will be entered into the newly made sub program, and a sub-program calling block such as M98 Pxxxx will be entered immediately after the cursor before the start of Contour programming.
By pressing [NO], you can abandon the conversion operation.
Contour figures can be converted to the following G-code programs.

Type of Figure	Symbol	G-code
Start point	●	G00 or G01
Line	→	G01
Arc (CW)		G02
Arc (CCW)		G03
Corner R	R	G02 or G03
Chamfering	C	G01
Cutter compensation		G41 or G42
Cutter compensation cancel		G40

NOTE
1 Converted NC program blocks are stored immediately after the block to which the cursor was positioned.
After a return to these previous screens, the cursor will be positioned to the head of the NC program that was newly stored after conversion.
2 If there are any figures for which the end points are pending, NC program conversion will be done for the contour figure immediately before the pending block.
3 A value of up to eight digits can be entered for the axis command (X/Y (Z)) of a converted NC program, and the value must always have a decimal point. The decimal digits conform to the minimum setting units, with any lower digits being rounded off.

(1) IS-B form

	Least input increment	Least command increment	Max.
mm	0.001 mm	0.001 mm	±99999.999 mm
inch	0.0001 inch	0.0001 inch	±9999.9999 inch

NOTE
1 By setting bit 0 (IJR) of parameter No.9341 to 1, address "R" is output as the radius data of arc n. By setting it to 0, data "I" and "J" are output as center point coordinates.
2 Even if data having the same value is output continuously for a given address, they will never be canceled.

2 CONTOUR FIGURE DATA

This chapter details contour figure data to be entered on the contour figure data screen.
The contour figure data of a start point and straight line is described in detail in the previous chapter. See the previous chapter if necessary.

Chapter 2, "CONTOUR FIGURE DATA", consists of the following sections:

2.1 START POINT

Data item	Comment
START POINT X	X coordinate of a contour figure start point
START POINT Y / Z	Y/Z coordinate of a contour figure start point
FEED METHOD	Type of motion toward a start point (No G-code/G00/G01)

The following soft keys are displayed.
[AUX.] : Call auxiliary calculating screen. The result of it will be set into start point coordinate data.
[OK] : Fix start point data and store into memory.
[EXIT] : Cancel start point entering and exit from contour programming.

NOTE
By setting a parameter No.9342#2(STP) to 1, data comment of start point can be changed to "APPROACH POINT".

2.2 LINE

If you select a line, the line screen is displayed, allowing you to enter all the figure data written on a drawing.

Even though the end point coordinate value does not appear in a drawing, it can be determined by calculating the cross point coordinate between this figure and that to be entered next.

Data item	Comment
START POINT X	X coordinate of a contour figure start point
START POINT Y / Z	Y/Z coordinate of a contour figure start point
ANGLE A	Angle of line from +Z-axis. A positive angle is counter clockwise direction.
TOUCH STATE	Select from the following soft-key menu whether tangential or not between neighbor figure. [NO] : Not tangential [LAST] :Tangential with the preceding figure

The following soft keys are displayed.

[AUX.] : Call auxiliary calculating screen. The result of it will be set into end point coordinate or angle data.
[OK] : Fix line figure data and store into memory.
[CANCEL] : Cancel line figure data entering and return to the contour program list screen.

2.3 ARC

Data item	Comment
END POINT X	X coordinate of an arc end point
END POINT Y/Z	Y/Z coordinate of an arc end point
RADIUS R	Radius of an arc, but plus value only
CENTER I	X coordinate of an arc center
CENTER J/K	Y/Z coordinate of an arc center
TOUCH STATE	Select from the following soft-key menu whether tangential or not between neighbor figure. [NO] : Not tangential [LAST] :Tangential with the preceding figure
FEEDRATE	Feedrate under G01 selecting

NOTE
1 The feedrate data item is displayed when parameter No.9341#3(FCD) is set to 1.
2 When entering the coordinates of an arc end point, do not round off the calculated value.

The following soft keys are displayed.
[AUX.] : Call auxiliary calculating screen. The result of it will be set into start point coordinate data.
[OK] : Fix arc figure data and store into memory.
[CANCEL]: Cancel arc figure data entering and return to the contour program list screen.

2.4 CORNER R

Data item	Comment
RADIUS R	Radius of a corner R, but plus value only
FEEDRATE	Feedrate

NOTE
The feedrate data item is displayed when parameter No.9341#3(FCD) is set to 1.

The following soft keys are displayed.
[OK] : Fix corner R figure data and store into memory.
[CANCEL] : Cancel corner R figure data entering and return to the contour program list screen.

2.5 CHAMFERING

Data item	Comment
CAMFER C	Chamfering amount, but plus value only
FEEDRATE	Feedrate

NOTE
The feedrate data item is displayed when parameter No.9341#3(FCD) is set to 1.

The following soft keys are displayed.
[OK] : Fix chamfering figure data and store into memory.
[CANCEL] : Cancel chamfering figure data entering and return to the contour program list screen.

2.6 SELECT CROSSING POINT OF FIGURE

During the calculation of a contour figure, such as between a line and an arc for example, there may be cases in which two or more cross points or figures are possible. In this case, the screen for selecting a cross point or figure appears.

The following soft keys are displayed.
[PREV.]/[NEXT] : Cross point or figure should be selected is changed. The active one blinks among some figures should be selected.
[OK] : Select an active figure, blinking, finally.
[CANCEL] : Cancel a selecting operation. And then, store the actual figure into memory still keeping it in pending.

3 CONTOUR CALCULATION

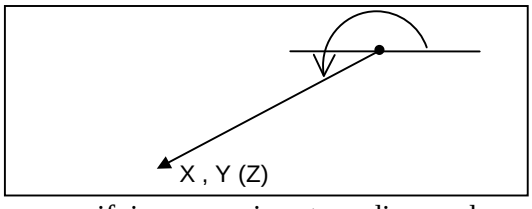
This chapter explains the details of contour calculations, such as those for cross points or tangential points, that are supported by contour programming.
A figure or part of a contour for which an end point has not yet been determined is said to be in the pending state. A pending figure is indicated by a dotted line.
On the screen for entering contour figure data, more data input items than required will appear. These data items are used to calculate the cross points with the immediately preceding pending figure block, and also to calculate the end point.
Ten successive figure blocks can be specified as pending blocks.

NOTE
It is impossible to enter successive straight lines with only their angles specified. (They are not assumed to be pending blocks.)

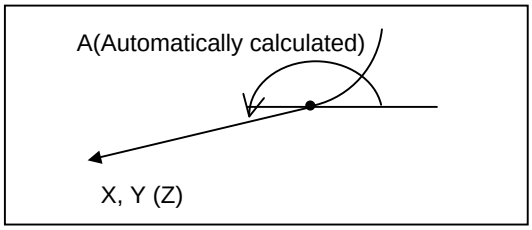
Chapter 3, "CONTOUR CALCULATION", consists of the following sections:

3.1 LINE

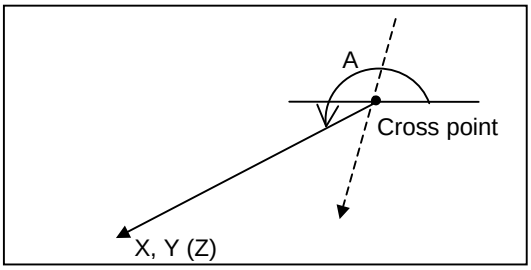
(1) When the preceding figure is not pending
  (a) Only X is inputted
    → This line is determined as a vertical line.
  (b) Only Y (Z) is inputted
    → This line is determined as a horizontal line.
  (c) A and either X or Y (Z) are inputted
    → The end point that is not inputted is calculated.



(2) When the preceding figure specifying an arc is not pending, and "TOUCH LAST" is specified in the line.
  (a) Either X or Y (Z) is inputted
    → The angle A is calculated automatically, and an end point is determined.
    If neither X nor Y (Z) is inputted, this line will be pending.

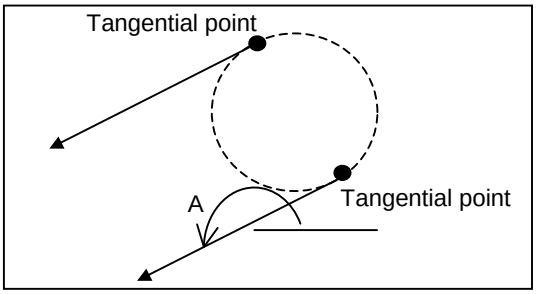


(3) When the preceding figure is pending, and "TOUCH LAST" is specified in the line.
(a) Both X and Y (Z), and A are inputted
→ The cross point between the preceding figure is calculated.

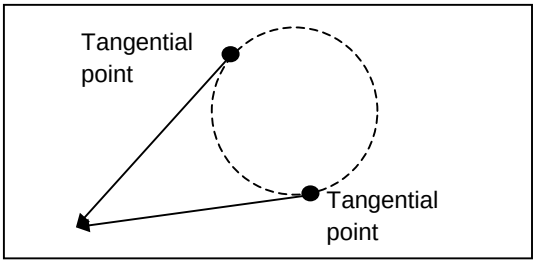


When the preceding figure is an arc, the cross point selection screen is displayed, so select a necessary one.

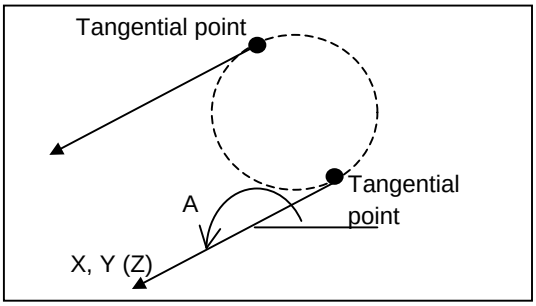
(4) When the preceding figure is pending arc, and "TOUCH LAST" is specified in the line.
It is assumed that the radius and the center point coordinate (I,K) of arc have already inputted.
(a) Only A is inputted
→ The tangential point selection screen is displayed, so select a necessary one.
This line will be pending.



(b) Both X and Y (Z) are inputted
→ The tangential point selection screen is displayed, so select a necessary one.
This line will be determined.



(c) A and either X or Y (Z) are inputted
→ The tangential point selection screen is displayed, so select a necessary one.
This line will be determined.



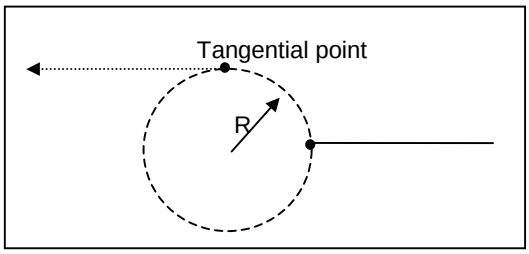
If the positional relationship between the tangential point and the line is such that an inputted A conflicts with the inputted X or Y (Z), a warning message is displayed to indicate that invalid data has been inputted.

3.2 ARC

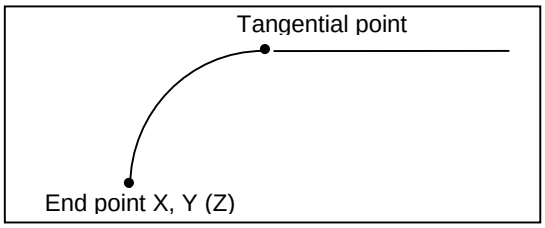
(1) When the preceding figure is not pending, and "TOUCH LAST" is not specified in the arc.
  (a) I and J (K) are inputted.
    → This arc will be pending.
  (b) X, Y (Z), and R are inputted.
    → A short path arc is uniquely defined.
  (c) X, Z, I, and J (K) are inputted.
    → A short path arc is uniquely defined.

NOTE
If the distance (radius) between the start point and center differs from the end point and center, the figure is displayed based on the actual form, and the actual figure will not be machined correctly.

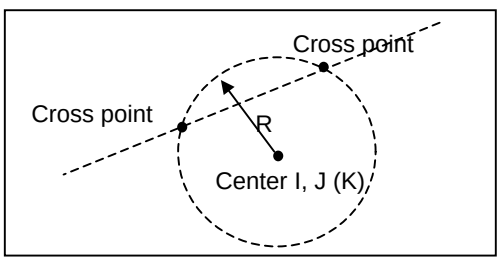
  (d) Only R is inputted
    → By specifying "TOUCH LAST" and inputting a line with A=0 degree and Y coordinate as an immediately after figure, this arc can be determined.



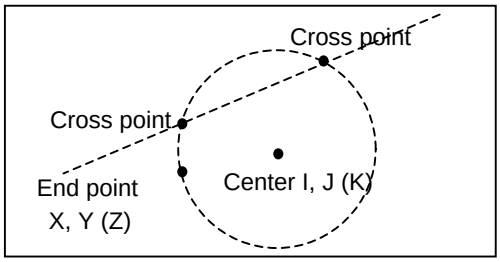
(2) When the preceding figure is not pending, and "TOUCH LAST" is specified in the arc.
  (a) X and Y (Z) are inputted.
    → The radius is automatically calculated and this arc will be determined.



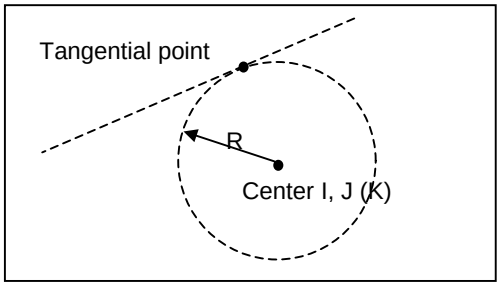
(3) When the preceding figure is pending (foe which the start point has been determined), and "TOUCH LAST" is not specified in this arc
  (a) R, I, and J (K) are inputted.
    → The cross point selection screen is displayed, so select a necessary one. This arc will be pending.



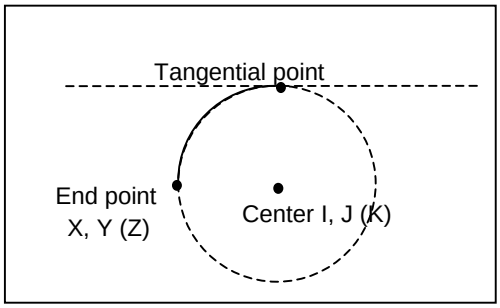
(b) X, Y (Z), I, and K are inputted.
→ The cross point selection screen is displayed, so select a necessary one. This arc will be determined.



(4) When the preceding figure is pending (for which the start point has been determined), and "TOUCH LAST" is specified in the arc
(a) R, I, and J (K) are inputted.
→ The tangential point is calculated, and this arc will be pending.

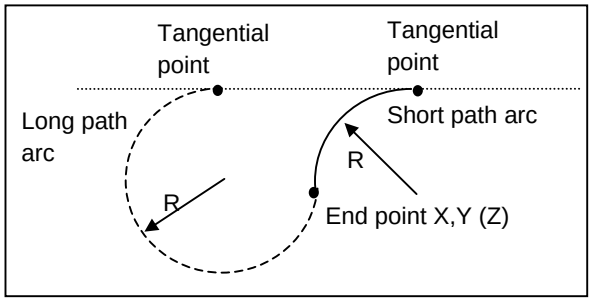


(b) X, Y (Z), I , and K are inputted.
→ The tangential point is calculated, and this arc will be determined.



NOTE
If the distance (radius) between the start point and center differs from the end point and center, the figure is displayed based on the actual form, and the actual figure will not be machined correctly.

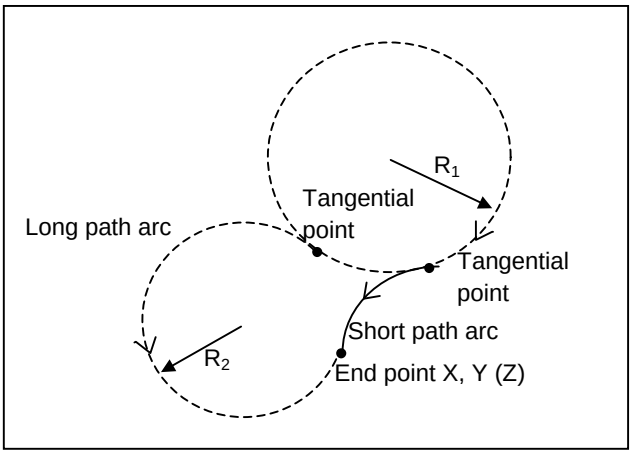
(c) R, X, and Y (Z) are inputted.
→ The tangential point is calculated and a short path arc is uniquely defined.



(5) When the preceding figure "arc" is pending (for which the start point has been determined and only R is to be inputted), and "TOUCH LAST" is specified in the arc.

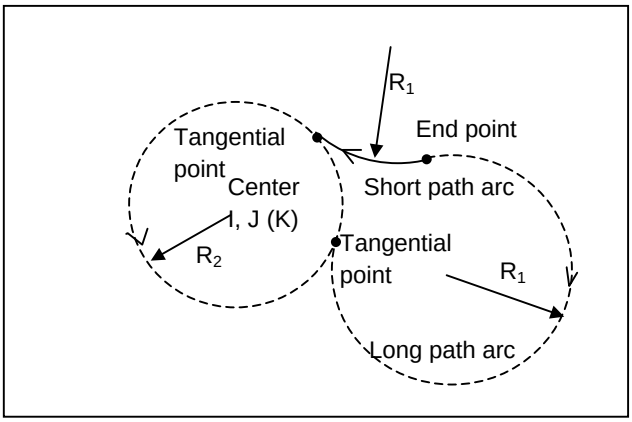
(a) R, X, and Y (Z) are inputted.

→ The tangential point is calculated and a short path arc is uniquely defined.

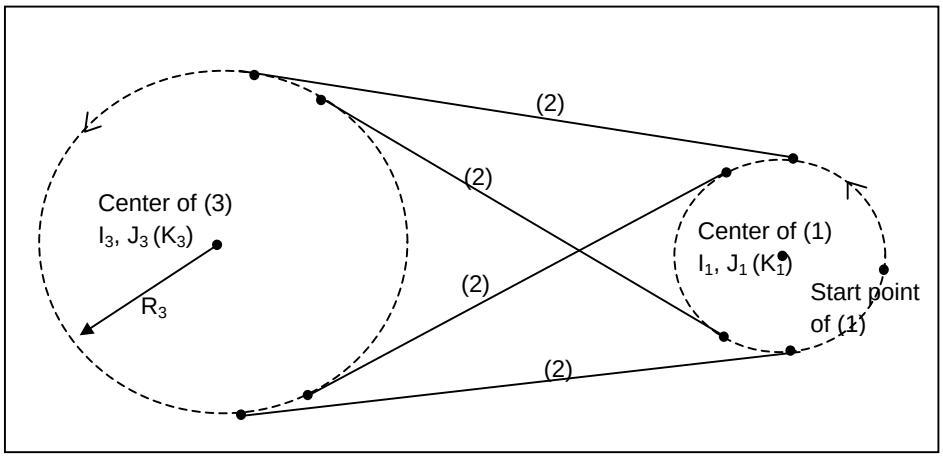


(b) R, X, and Y (Z) are inputted.

→ The tangential point is calculated and a short path arc is uniquely defined.



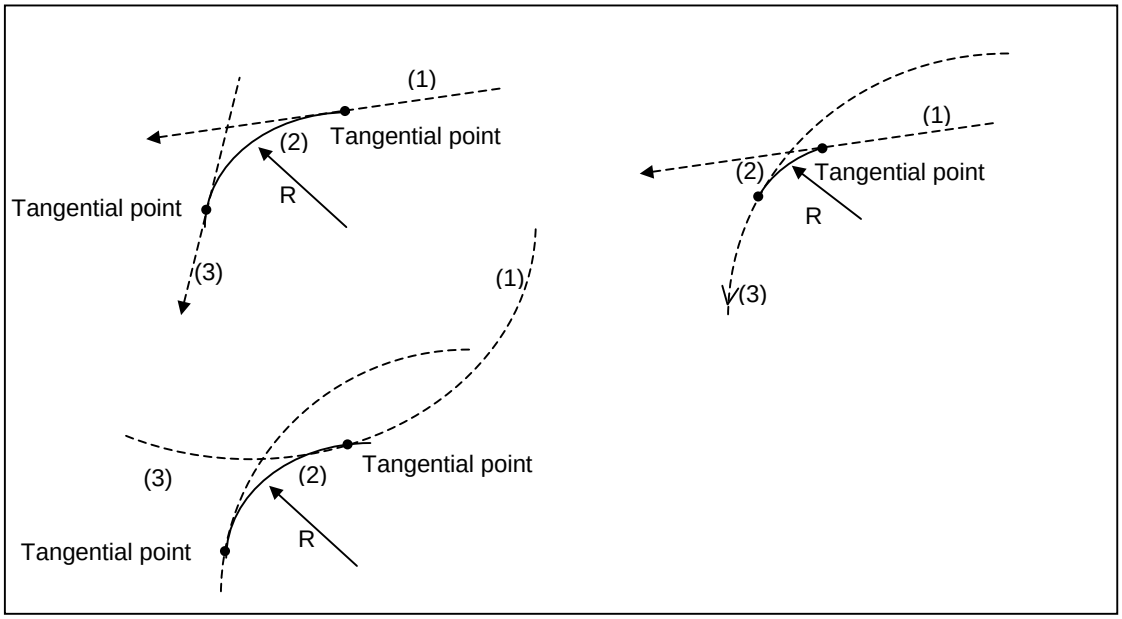
3.3 LINE TANGENTIAL TO TWO ARCS



By inputting three successive figures as follows, line (2) that is tangential to two arcs can be specified as shown in the above drawing. The end points of (1) and (2) are determined, while (3) is left pending.
Among the above four possible lines, depending on the direction of the two arcs, the line that makes the smoothest connection to the arcs is automatically selected.

Arc (1) :
I and J (K) are inputted. (A start point is determined. This arc is pending.)
Line (2) :
Only "TOUCH LAST" is inputted.
Arc (3) :
R, I and J (K) are inputted.

3.4 ARC THAT CONTACTS TO BOTH CROSSING LINES AND ARCS

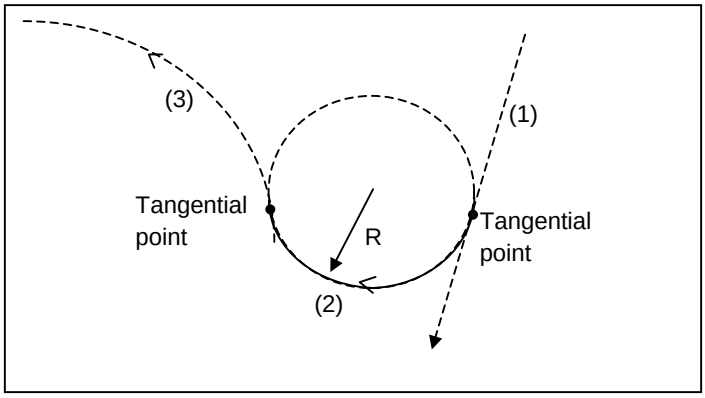


By inputting three successive figures as follows, arc (2) that is tangential to two lines or arcs can be specified as shown in the above drawing. The end points of (1) and (2) are determined, while (3) is left pending. When (3) is a line, it is determined.

Line (1) or Arc (1) :
Line that is pending (for which A is inputted and the start point has been determined), or Arc that is pending (for which I and J (K) are inputted and the start point has been determined)
Arc (2) :
R and "TOUCH LAST" are inputted.
Line (3) or Arc (3) :
Line with A, X and, or Arc with R, I, J (K) and "TOUCH LAST"

When either figure (1) or (3) is an arc, or when both of them are arcs, the selection screen for multiple arcs is displayed, so select a suitable one.

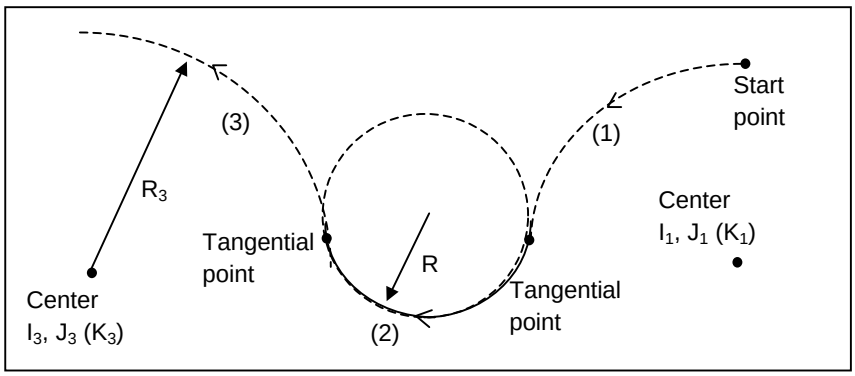
3.5 ARC THAT CONTACTS TO UNCROSSING LINE AND ARC



By inputting three successive figures as follows, arc (2) that is tangential to line (1) and arc (3) that do not cross, can be specified as shown in the above drawing. The end points of (1) and (2) are determined, while (3) is left pending.
Among the multiple possible arcs shown above, the arc that makes the smoothest connection to line (1) and arc (3) is automatically selected. But, even though a "Long path arc" and "Short path arc" remain, select whichever is necessary from the selection screen.

Line (1) : Line that is pending (for which A is inputted and the start point has been determined)
Arc (2) : R and "TOUCH LAST" are inputted.
Arc (3) : Arc with R, I and J (K)

3.6 ARC THAT CONTACTS TO UNCROSSING 2 ARCS



By inputting three successive figures as follows, arc (2) that is tangential to arcs (1) and (3) that do not cross can be specified as shown in the above drawing.
The end points of (1) and (2) are determined, while (3) is left pending.
Among the multiple possible arcs shown above, the arc that makes the smoothest connection to arcs (1) and (3) is automatically selected. But, even though a "Long path arc" and "Short path arc" remain, select whichever is necessary from the selection screen.

Arc (1) : Arc with I and J (K), and it is pending (for which the start point has been determined)
Arc (2) : R and "TOUCH LAST" are inputted.
Arc (3) : Arc with R, I, and J (K)

4 AUXILIARY CALCULATION

This chapter explains the details of the auxiliary calculation.
By using this auxiliary calculation, the coordinates of a point or the angle of a line can be determined. Furthermore, the form of a contour, such as the amount of shift from the original form for a tool radius can be entered easily.

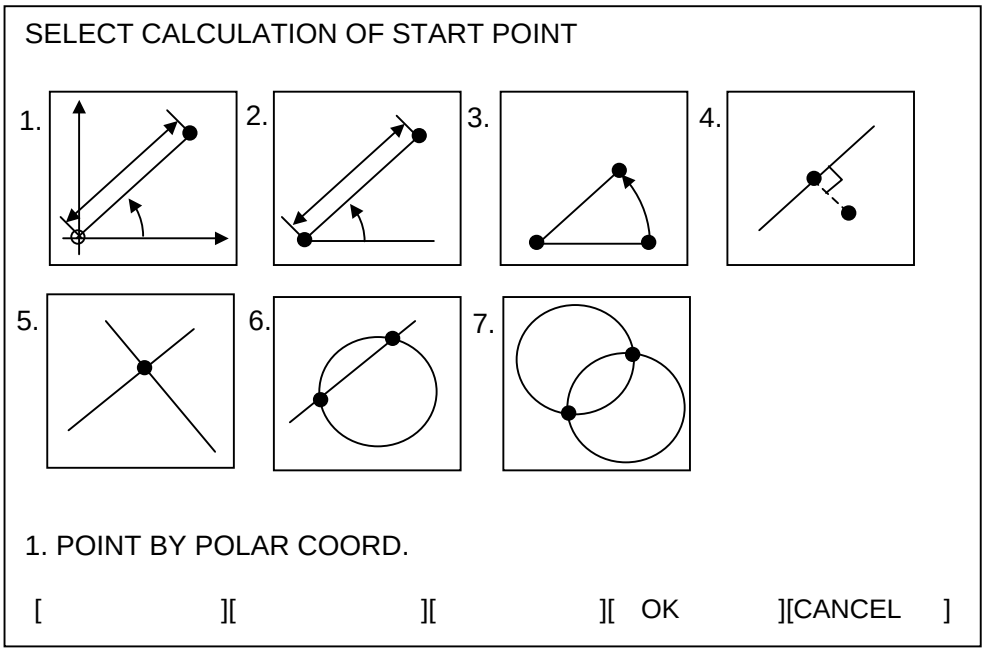
(1) Data items in which auxiliary calculation can be used
  (a) Start point
    - Coordinate (X, Y (Z)) of start point
  (b) Line
    - Coordinate (X, Y (Z)) of end point
    - Angle of a line (A)
  (c) Arc
    - Coordinate (X, Y (Z)) of end point
    - Coordinate (I, J(K)) of center
    - Specifying an arc

(2) Calculation type available in auxiliary calculation
  (a) Calculation of coordinate
    - A point specified by polar coordinate
    - A point specified by a point, angle and distance
    - A point specified by rotating a point
    - Neighbor point of a line
    - Cross point between 2 lines
    - Cross point between line and arc
    - Cross point between 2 arcs
  (b) Calculation of angle
    - Angle of the line passes 2 points
    - Angle of a line rectangular to the line passes 2 points
  (c) Specify an arc (Center and radius)
    - An arc passes 1 point and its center coordinate has been determined
    - An arc passes 2 points and its radius has been determined
    - An arc passes 3 points

4.1 START POINT

(1) Selecting type of calculation

On the data-entry screen for a start point, press [AUX.]. The following calculation type menu screen will appear.
By pressing the cursor key, scroll through the menu comments until you find the one to select.



The following soft keys are displayed.
[OK] : Use the active calculation type
[CANCEL] : Cancel selecting operations and return to the previous screen

NOTE
When bit 5 (AUX) of parameter No.9305 is set to 1, the menu for auxiliary calculation is displayed as shown above. By setting it to 0, the menu is displayed as a list of comments for each calculation type.
This parameter is available for other auxiliary calculation menus.

(2) Entering data for calculation

(a) A point specified by polar coordinate

Data item	Comment
DIST. D	Distance between the point and work coordinate origin
ANGLE A	Angle of line from +Z-axis. A positive angle is counter clockwise direction.

(b) A point specified by a point, angle and distance

Data item	Comment
POINT X	X coordinate of the base point
POINT Y / Z	Y or Z coordinate of the base point
DIST. D	Distance between the point and the base point
ANGLE A	Angle of line from +Z-axis. A positive angle is counter clockwise direction.

(c) A point specified by rotating a point

Data item	Comment
POINT X	X coordinate of the base point
POINT Y / Z	Y or Z coordinate of the base point

Data item	Comment
CENTER I	X coordinate of the rotating center
CENTER J / K	Y or Z coordinate of the rotating center
ANGLE A	Angle of line from +Z-axis. A positive angle is counter clockwise direction.

(d) Neighbor point of a line

You can determine the nearest point to the line. Furthermore, you can determine the nearest point to the line that has been moved through a specific distance.
This can be useful for determining the end point of an approaching motion from a certain point close to the line.

(1) When specifying a line with one point and an angle

Data item	Comment
BASE POINT X	X coordinate of the base point positioned apart from a line
BASE POINT Y / Z	Y or Z coordinate of the base point positioned apart from a line
PASS POINT U	X coordinate of a certain point on the line
PASS POINT V / W	Y or Z coordinate of a certain point on the line
ANGLE A	Angle of line from +X or +Z-axis. A positive angle is counter clockwise direction.
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

(2) When specifying a line with two points
By pressing soft key [XY,XY] or [XZ,XZ], you can specify a line with two passing points.
By pressing soft key [XY, A] or [XZ, A], you can select the above type by using an angle again.

Data item	Comment
BASE POINT X	X coordinate of the base point positioned apart from a line
BASE POINT Y / Z	Y or Z coordinate of the base point positioned apart from a line
PASS POINT U	X coordinate of the 1st passing point on the line
PASS POINT V / W	Y or Z coordinate of the 1st passing point on the line
PASS POINT P	X coordinate of the 2nd passing point on the line
PASS POINT Q	Y or Z coordinate of the 2nd passing point on the line
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting distance
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

(e) Cross point between 2 lines

The cross point of two lines can be calculated. Furthermore, this calculation can also be done for a line that is shifted from its original position by some distance.
This can be useful when entering a tool path that is shifted from the original figure by an amount equal to the tool radius.

(1) When specifying a line with one point and an angle
Initially, the following data items are displayed for Line-1.

Data item	Comment
PASS POINT X	X coordinate of a certain point on the line
BASE POINT Y / Z	Y or Z coordinate of a certain point on the line
ANGLE A	Angle of line from +X or +Z-axis. A positive angle is counter clockwise direction.
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting distance
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

By pressing [NEXT] after entering the necessary data, a similar data item screen for Line-2 is displayed. By pressing [PREV.], you can return to the previous screen for Line-1.

(2) When specifying a line with two points
By pressing soft key [XY,XY] or [XZ,XZ], you can specify a line with two passing points.
By pressing soft key [XY, A] or [XZ, A], you can select the above type by using an angle again.

The following data items are displayed respectively for Line-1 and Line-2.

Data item	Comment
PASS POINT X	X coordinate of the 1st passing point on the line
PASS POINT Y / Z	Y or Z coordinate of the 1st passing point on the line
PASS POINT U	X coordinate of the 2nd passing point on the line
PASS POINT V / W	Y or Z coordinate of the 2nd passing point on the line
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

(f) Cross point between line and arc

The cross point between a line and arc can be calculated. For the line, one that has been shifted through some distance can be used for the calculation.
This is useful for entering a tool path that has been shifted from the original figure by an amount equal to the tool radius.

(1) When specifying a line with one point and an angle
Initially, the following data items are displayed for the Line.

Data item	Comment
PASS POINT X	X coordinate of the 1st passing point on the line
PASS POINT Y / Z	Y or Z coordinate of the 1st passing point on the line
ANGLE A	Angle of line from +X or +Z-axis. A positive angle is counter clockwise direction.
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

By pressing [NEXT] after entering the necessary data, the following screen for the Arc is displayed.
By pressing [PREV.], you can return to the previous screen for the Line.
By pressing [NEXT] after entering the necessary data, a similar data item screen for Line-2 is displayed. By pressing [PREV.], you can return to the previous screen for Line-1.

Data item	Comment
CENTER I	X coordinate of an arc center
CENTER J / K	Y or Z coordinate of an arc center
RADIUS R	Radius of an arc, but plus value only
SELECT	Select necessary cross point from two possible points by arrow soft-keys

(2) When specifying a line with two points
By pressing [XZ,XZ], you can specify a line with two passing points.
By pressing [XZ, A], you can select the above type using an angle again.

Data item	Comment
PASS POINT X	X coordinate of the 1st passing point on the line
PASS POINT Y / Z	Y or Z coordinate of the 1st passing point on the line
PASS POINT U	X coordinate of the 2nd passing point on the line
PASS POINT V / W	Y or Z coordinate of the 2nd passing point on the line
SHIFT DIST. D	When the line should be specified by shifting an original line, enter the shifting distance
SHIFT DIRC.	Select a shifting direction by arrow soft-keys

Data screen for the arc is displayed quite similar to the case of (1).

(g) Cross point between 2 arcs

On the screen as shown below, data for two arcs can be entered and the cross point between them can be calculated.

Data item	Comment
CENTER X1	X coordinate of an arc-1 center
CENTER Y1 / Z1	Y or Z coordinate of an arc-1 center
RADIUS R1	Radius of an arc-1, but plus value only
CENTER X2	X coordinate of an arc-2 center

Data item	Comment
CENTER Y2 / Z2	Y or Z coordinate of an arc-2 center
RADIUS R2	Radius of an arc-2, but plus value only
SELECT	Select necessary cross point from two possible points by arrow soft-keys

(3) Execution of auxiliary calculation

After entering all of the necessary data for each of the above calculation types, press [OK]. The auxiliary calculation is done, and then the result is entered into the coordinate data item (X, Z) of the start point.
By pressing [CANCEL], you can return to the auxiliary calculation menu screen.

4.2 LINE

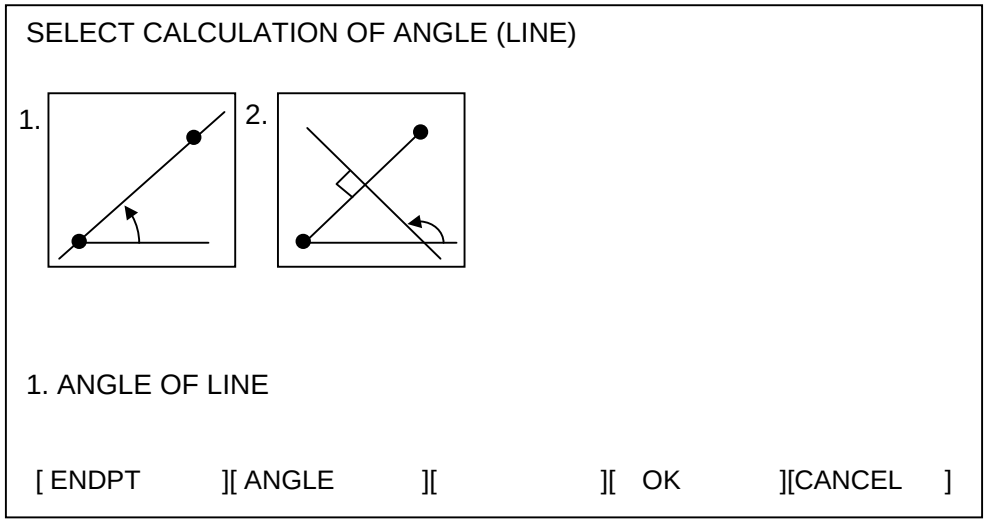
As part of the auxiliary calculation for a line, the end point coordinate and angle can be calculated.
The following soft-keys are displayed on the auxiliary calculation menu screen.

[ENDPNT] : Menu screen for end point calculation is displayed
[ANGLE] : Menu screen for angle calculation is displayed

For the end point calculation, these items are similar to those for the start point, so refer to the preceding explanation.

(1) Selecting type of calculation

On the menu screen for the auxiliary calculation of a line, press [ANGLE]. The following menu screen for angle calculation appears.
By pressing the cursor key, you can scroll through the menu comments until you find the type that you want to select.



(2) Entering data for calculation

(a) Angle of the line passes 2 points

Data item	Comment
POINT X	X coordinate of the 1st passing point on the line
POINT Y / Z	Y or Z coordinate of the 1st passing point on the line
POINT U	X coordinate of the 2nd passing point on the line, this should be apart from the 1st point
POINT V / W	Y or Z coordinate of the 2nd passing point on the line, this should be apart from the 1st point

(b) Angle of a line rectangular to the line passes 2 points

The angle of a line that is rectangular to a line and which passes through two points can be calculated.

Data item	Comment
POINT X	X coordinate of the 1st passing point on the line
POINT Y / Z	Y or Z coordinate of the 1st passing point on the line
POINT U	X coordinate of the 2nd passing point on the line, this should be apart from the 1st point
POINT V / W	Y or Z coordinate of the 2nd passing point on the line, this should be apart from the 1st point

(3) Execution of auxiliary calculation

After entering all of the necessary data for each of the above calculation types, press [OK]. The auxiliary calculation is performed, after which the result is entered into the end point coordinate (X, Z) or angle (A) of the line.
By pressing [CANCEL], you can return to the auxiliary calculation menu screen.

4.3 ARC

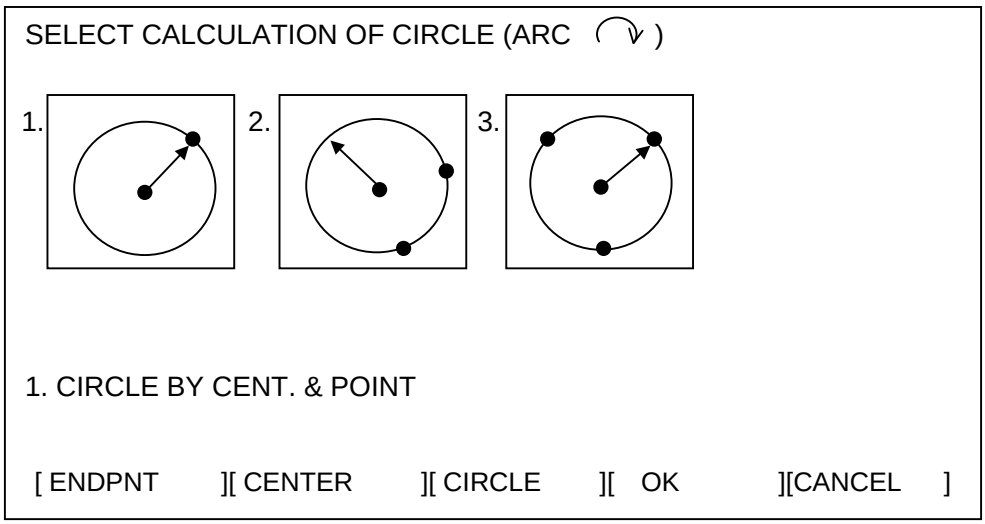
As part of the auxiliary calculation for an arc, the end point coordinate and center coordinate can be calculated. Furthermore, the arc itself can be specified by entering the likes of three passing points.
The following soft-keys are displayed on the auxiliary calculation menu screen.

[ENDPNT] : Menu screen for end point calculation is displayed
[CENTER] : Menu screen for center point calculation is displayed
[CIRCLE] : Menu screen for circle specifying is displayed

For the end point and center point calculation, they are quite similar to those for the start point case, so refer to the preceding explanation.

(1) Selecting type of calculation

On the menu screen for auxiliary calculation of an arc, press [CIRCLE]. The following menu screen for specifying a circle is displayed.
By pressing the cursor keys, you can scroll through the menu items until you find the item that you want to select. The currently selected item is highlighted in yellow.



(2) Entering data for calculation

(a) An arc passes 1 point and its center coordinate has been determined

Data item	Comment
POINT X	X coordinate of a certain point on the arc
POINT Y / Z	Y or Z coordinate of a certain point on the arc
CENTER I	X coordinate of an arc center
CENTER J / K	Y or Z coordinate of an arc center

(b) An arc passes 2 points and its radius has been determined

Data item	Comment
POINT X	X coordinate of the 1st passing point on the arc
POINT Y / Z	Y or Z coordinate of the 1st passing point on the arc
POINT U	X coordinate of the 2nd passing point on the arc, this should be apart from other points
POINT V /W	Y or Z coordinate of the 2nd passing point on the arc, this should be apart from other points
RADIUS R	Radius of the arc, but plus value only
SELECT	Select necessary one from two possible arcs by arrow soft-keys

(c) An arc passes 3 points

Data item	Comment
POINT X	X coordinate of the 1st passing point on the arc
POINT Y / Z	Y or Z coordinate of the 1st passing point on the arc
POINT U	X coordinate of the 2nd passing point on the arc, this should be apart from other points
POINT V / W	Y or Z coordinate of the 2nd passing point on the arc, this should be apart from other points
POINT P	X coordinate of the 3rd passing point on the arc, this should be apart from other points
POINT Q	Y or Z coordinate of the 3rd passing point on the arc, this should be apart from other points

(3) Execution of auxiliary calculation

After entering all the necessary data for each of the above calculation types, press [OK]. The auxiliary calculation is performed, after which the result is entered into the end point coordinate (X, Z) or center point coordinate (I, K) of an arc.
When circle specification is selected by [CIRCLE], the radius and center point coordinate are calculated and the results are entered into these data items.

By pressing [CANCEL], you can return to the auxiliary calculation menu screen.

4.4 OTHERS

4.4.1 Calculation of Inputting Data

When entering data for the items on the contour programming screen, calculation as shown below can be made as in a desktop calculator.

Addition :

10+10  → 20

Subtraction :

10-10  → 0

Multiplication :

10*10  → 100

Division :

10/10  → 1

SIN :

S30  → 0.5

COS :

C60  → 0.5

TAN :

T45  → 1

Square root :

R9  → 3

NOTE

1 For + - * /, more than two terms cannot be handled at once. The third and subsequent terms are neglected. So, 1 + 2 + 3 will be calculated as 1 +2.
2 The calculation of SIN, COS, TAN and Square root can be done as only independent calculations. The second and subsequent terms, if specified, will be neglected. C60 + S60 will be calculated as C60 only.
Calculation based on previously input data is also possible.

4.4.2 Notes

NOTE

1 No more than forty figures can be entered for a contour program.
2 During contour program operation, if the screen is changed to another ones by the user pressing the likes of a function button, the active screen displayed in contour programming will be forced to close.
3 When editing a contour program, keep the following in mind.
(1) A contour program that was operated one or more times cannot be edited.
(2) After creating a contour program, another contour figure cannot be newly entered.

APPENDIX

A ALARMS

If one or more of the set of the parameters or inputted programs are not correct when an attempt is made to execute that program, the following P/S alarms are raised.
When an alarm other than the following P/S alarms is raised, refer to the relevant NC operator's manual.

Appendix A, "ALARMS", consists of the following sections:

A.1 MACHINING CENTER SYSTEM ALARMS

Alarm No.	Description	
3001	Cause	Necessary data is not entered. Or entered data is invalid.
	Action	Display the block data of pop-up window, at which is occurred the alarm, and enter the correct data after confirming it.
	Reference	All cycle machining other than hole machining in "3. INPUT ITEM OF EACH CYCLE" of III MILLING CYCLE
3002	Cause	The offset data corresponding to the specified D code is 0 or less.
	Action	Confirm the D code, at which is occurred the alarm and enter the correct data to the offset table.
	Reference	All cycle machining other than hole machining in "3. INPUT ITEM OF EACH CYCLE" of III MILLING CYCLE
3004	Cause	Machining is impossible because the cutter diameter is too large.
	Action	Confirm the tool or block data, at which is occurred the alarm and select the tool of smaller radius than the last time.
	Reference	Pocketing in "3. INPUT ITEM OF EACH CYCLE" of III MILLING CYCLE
3005	Cause	The tool interferes with the opposite edge because the length of approach is too long.
	Action	Confirm the approach data, at which is occurred the alarm and enter the correct data to the approach.
	Reference	Pocketing in "3. INPUT ITEM OF EACH CYCLE" of III MILLING CYCLE
3006	Cause	Corner R interferes with the opposite one because the radius of corner R is too large.
	Action	Confirm the radius of corner R, at which is occurred the alarm and enter the correct data to the radius.
	Reference	Pocketing for corner R in "3. INPUT ITEM OF EACH CYCLE" of III MILLING CYCLE
3008	Cause	Corner R machining can not be performed because the cutter diameter is larger than corner R.
	Action	Confirm the tool or block data, at which is occurred the alarm and select the tool of smaller radius than the last time.
	Reference	Pocketing for corner R in "3. INPUT ITEM OF EACH CYCLE" of III MILLING CYCLE
3012	Cause	The chamfering tool interferes with the bottom surface (Z point) in chamfering.
	Action	Confirm the block data or block data related to chamfer tool, at which is occurred the alarm and enter the correct data to it.
	Reference	Pocketing of "3. INPUT ITEM OF EACH CYCLE" of III MILLING CYCLE
3013	Cause	The angle at which the chamfering tool is placed is not specified.
	Action	Confirm the block data related to chamfer tool, at which is occurred the alarm and enter the correct data to it.
	Reference	Pocketing in "3. INPUT ITEM OF EACH CYCLE" of III MILLING CYCLE

Alarm No.	Description	
3022	Cause	Tool diameter is 0 or it is not specified.
	Action	Input correct data.
	Reference	“3. INPUT ITEM OF EACH CYCLE” of III MILLING CYCLE
3025	Cause	Machinining block and figure block are not specified.
	Action	In Machining cycle, machining block and the figure block has to be specified.
	Reference	“2.1 NEWLY CREATING A MACHINING CYCLE” of III MILLING CYCLE
3026	Cause	Machining type is not correct in G1055.
	Action	Specify the machining type except 2 and 4.
	Reference	3.4 GROOVE MACHINING. Of III MILLING CYCLE

A.2 LATHE SYSTEM ALARMS

Alarm No.	Description	
3001	Cause	The value of canned cycle data is incorrect. For example, a negative value is entered for an item that must be positive.
	Action	Display the input data screen of the block in which this alarm occurred and input correct data if the value of the entered data is incorrect.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE
3002	Cause	The figure block of the cycle machining specified in the machining type block cannot be found.
	Action	Enter the figure data of the cycle machining specified in the machining type block.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE
3005	Cause	The feedrate or feed amount is not specified.
	Action	Display the input data screen of the block in which this alarm occurred and input correct data if the value of the entered data is incorrect.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE
3006	Cause	Check the cutting amount.
	Action	Display the input data screen of the block in which this alarm occurred and input correct data if the value of the entered data is incorrect.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE
3015	Cause	Because the finishing allowance is too large or tool nose R radius is too large, the rough cutting cannot be executeed in turning or trapezoidal grooving.
	Action	Reduce finishing allowance or change tool with smaller nose R radius.
	Reference	IV Turnning cycle
3016	Cause	Tool nose R radius is too large, the finnishing cannot be executed in turning or trapezoidal grooving.
	Action	Change tool with smaller nose R radius.
	Reference	IV Turnning cycle
3020	Cause	Cutting in bar machining is impossible because of the relationship between the tool angle, cutting edge angle, and clearance angle (parameter No. 9371) for the tool used. This alarm is raised when the sum of the above three angles is less than or equal to 90 degrees or is greater than or equal to 180 degrees.
	Action	Check these three angles value, and if necessary change the tool.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE
3022	Cause	Machining figure is not correct in turning.
	Action	Input correct figure.
	Reference	IV Turnning cycle
3025	Cause	A correct tool path cannot be calculated in bar machining. This alarm is raised when there is a error in the result of internal calculation (for example, when the value under a square root sign is negative because of an error in calculation)
	Action	Check the figure data of the Bar machining and specify smaller finish amount or use a tool with smaller nose radius.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE

Alarm No.	Description	
3026	Cause	The groove width or tool used is incorrect in grooving. This alarm is raised when the width of the tool is used is larger than the groove width excluding finishing amount.
	Action	Check the groove width finishing amount and tool width.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE
3028	Cause	Cutting trapezoidal grooving is impossible because of the relationship of the groove bottom width and the tool width. This alarm is raised when the cutting edge width of the tool used is larger than the groove bottom width excluding finishing amount.
	Action	Check the groove width finishing amount and tool width.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE
3029	Cause	The start point for threading coincides with the end point.
	Action	Check the figure data for the threading to this alarm, and enter correct figure data.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE
3035	Cause	The number of figure blocks exceeded the maximum block number.
	Action	Change the setting so that the number of figure blocks does not exceed the maximum block number.
	Reference	V CONTOUR PROGRAMMING
3036	Cause	The groove finishing machining block (G1133/G1135) was combined with the normal groove figure (G1460) during command execution.
	Action	To perform only roughing or finishing with the finishing allowance left in a normal groove, set bit 4 of parameter No. 9302 to 0.
	Reference	“3. INPUT ITEM OF EACH CYCLE” in IV TURNING CYCLE

B PARAMETERS

> ⚠ **WARNING**
> Be sure to use the parameter settings made by the machine tool builder. Otherwise, a machining cycle may not operate correctly.
> If a machining cycle does not operate correctly, the tool may interfere with the workpiece or perform forced machining, causing breakage of the tool or machine or serious injury.

Appendix B, "PARAMETERS" consists of the following sections.

B.1 PARAMETERS COMMON TO THE MACHINING CENTER AND LATHE SYSTEM

9048	EXEPNO

EXEPNO P-CODE macro number of the execution macro
When using MANUAL GUIDE 0*i*, be sure to set this parameter to 94 for the M series or 96 for the T series.
[Valid data range] 94 or 96
[Unit of data] 1

9049	CAPPNO

CAPPNO P-CODE macro number of the conversation macro
When using MANUAL GUIDE 0*i*, be sure to set this parameter to 94 for the M series or 96 for the T series.
[Valid data range] 94 or 96
[Unit of data] 1

9050	AUXPNO

AUXPNO P-CODE macro number of the auxiliary macro
When using MANUAL GUIDE 0*i*, be sure to set this parameter to 94 for the M series or 96 for the T series.
[Valid data range] 94 or 96
[Unit of data] 1

9071	EX2PNO

AUXPNO P-CODE macro number of the second execution macro
When using MANUAL GUIDE 0*i*, be sure to set this parameter to 93 for the M series or 95 for the T series.
[Valid data range] 93 or 95
[Unit of data] 1

	#7	#6	#5	#4	#3	#2	#1	#0
9300			PD2			PD1		

PD1 = 0 : Does not change the process data screen (MACHINING CENTER SYSTEM)
= 1 : Changes the process data screen (MACHINING CENTER SYSTEM)
PD2 = 0 : Does not change the process data screen (LATHE SYSTEM)
= 1 : Changes the process data screen (LATHE SYSTEM)

	#7	#6	#5	#4	#3	#2	#1	#0
9301	COL				CS3	CS2	CS1	CS0

COL= 0 : M7 is Flood and M8 is Mist on the process data input screen.
= 1 : M7 is Mist and M8 is Flood on the process data input screen.

CS1,CS0 = 0,1 : When CNC is powered on, CUSTOM screen(Conversational Macro screen1) of the 1st path is displayed.
= 1,0 : When CNC is powered on, CUSTOM screen(Conversational Macro screen2) of the 1st path is displayed.
= 1,1 : When CNC is powered on, CUSTOM screen(Conversational Macro screen3) of the 1st path is displayed.
= 0,0 : When CNC is powered on, POSITION screen(Current Position screen) of the 1st path is displayed.
CS3,CS2 = 0,1 : When CNC is powered on, CUSTOM screen(Conversational Macro screen1) of the 2nd path is displayed.
= 1,0 : When CNC is powered on, CUSTOM screen(Conversational Macro screen2) of the 2nd path is displayed.
= 1,1 : When CNC is powered on, CUSTOM screen(Conversational Macro screen3) of the 2nd path is displayed.
= 0,0 : When CNC is powered on, POSITION screen(Current Position screen) of the 2nd path is displayed.

NOTE
In case of setting the parameter No.9301#0<CS0>～#3<CS3>, set also the parameter No.8650#1<CNA> to 1 (When CNC alaram occurs during User screen by C-language executor is displayed, Alarm screen is not displayed independent of the parameter No.3111#7<NPA> setting).

	#7	#6	#5	#4	#3	#2	#1	#0
9304	M99	CMP	DCD	G41	FCD		RAD	IJR

IJR = 0 : The arc command in the I/J format is output in NC program conversion.
= 1 : The arc command in the R format is output.
RAD = 0 : The unit of angle data becomes degree.
= 1 : The unit of angle data becomes radian.
FCD = 0 : Feedrate data cannot be entered.
= 1 : Feedrate data can be entered.
G41 = 0 : Cutter compensation data cannot be entered.
= 1 : Cutter compensation data can be entered.
DCD = 0 : A compensation number can be entered when G41 is 1.
= 1 : A compensation number cannot be entered when G41 is 1.
CMP = 0 : The start point screen is displayed first.
= 1 : The compensation data screen is displayed first.

M99 = 0 : A converted NC program is not followed by M99.
= 1 : A converted NC program is followed by M99.

	#7	#6	#5	#4	#3	#2	#1	#0
9305			AUX			STP	KEY	COL

COL = 0 : The standard color is used for guidance drawing.
= 1 : The color set by parameters No. 9400 to 9409 is used for guidance drawing.
KEY = 0 : The up, down, left, and right cursor keys are provided as MDI keys.
= 1 : Only the up and down cursor keys are provided as MDI keys.

- The up, down, left, and right cursor keys are displayed as soft keys on page 3 of the contour program list screen.
- The cursor on the contour program list screen moves right with the down cursor key or left with the up cursor key.

STP = 0 : [START POINT] is displayed in the start point data window.
= 1 : [APPROACH POINT] is displayed in the start point data window.
AUX = 0 : The auxiliary menu is displayed as a comment list.
= 1 : The auxiliary menu is displayed as a drawing.

	#7	#6	#5	#4	#3	#2	#1	#0
9316			TG2				M99	

M99 = 0 : An subprogram is not followed by M99 during output.
= 1 : An subprogram is followed by M99 during output.
TG2 = 0 : Does not display adjacent condition soft key [NEXT].
= 1 : Displays adjacent condition soft key [NEXT].

9358	TMPPNO

TMPPNO Program number used as a temporary program saving area for NC program conversion. When this parameter is 0, a warning message appears and selection of the MANUAL GUIDE 0*i* screen is disabled.

9400	CNTCL0

CNTCL0 Number indicating the color of the guide drawing of the contour programming screen (color of the contour)
[Valid data range] 0 to 255
[Unit of data] 1

9401	CNTCL1

CNTCL1 Number indicating the color of the guide drawing of the contour programming screen (color of the end point)
[Valid data range] 0 to 255
[Unit of data] 1

9402	CNTCL2

CNTCL2 Number indicating the color of the guide drawing of the contour programming screen (color of the center point)
[Valid data range] 0 to 255
[Unit of data] 1

9403	CNTCL3

CNTCL3 Number indicating the color of the guide drawing of the contour programming screen (color of the drawing of calculation results)

[Valid data range] 0 to 255

[Unit of data] 1

9404	CNTCL4

CNTCL4 Number indicating the color of the guide drawing of the contour programming screen (color of the pass points)

[Valid data range] 0 to 255

[Unit of data] 1

9405	CNTCL5

CNTCL5 Number indicating the color of the guide drawing of the contour programming screen (color of the reference position/reference line)

[Valid data range] 0 to 255

[Unit of data] 1

9406	CNTCL6

CNTCL6 Number indicating the color of the guide drawing of the contour programming screen (color of dimension lines)

[Valid data range] 0 to 255

[Unit of data] 1

9407	CNTCL7

CNTCL7 Number indicating the color of the guide drawing of the contour programming screen (color of auxiliary lines)

[Valid data range] 0 to 255

[Unit of data] 1

9408	CNTCL8

CNTCL8 Number indicating the color of the guide drawing of the contour programming screen (color of shift lines)

[Valid data range] 0 to 255

[Unit of data] 1

9409	CNTCL9

CNTCL9 Number indicating the color of the guide drawing of the contour programming screen (the color of the other line of the intersection of two elements)

[Valid data range] 0 to 255

[Unit of data] 1

B.2 LATHE SYSTEM PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
9302				GRV				

GRV = 0 : Does not distinguish roughing and finishing in normal grooving.
= 1 : Distinguishes roughing and finishing in normal grooving.

	#7	#6	#5	#4	#3	#2	#1	#0
9303						BKT		

BKT = 0 : Disables the back tool post (X negative area machining).
= 1 : Enables the back tool post (X negative area machining).

	#7	#6	#5	#4	#3	#2	#1	#0
9306				ING	EFV			

ING = 0 : Disables inner surface grooving.
= 1 : Enables inner surface grooving.
EFV = 0 : Disables the feedrate override and cutting amount override of the end face in concurrent end facing (roughing).
= 1 : Enables the feedrate override and cutting amount override of the end face in concurrent end facing (roughing)

Related parameters: P9359, P9360

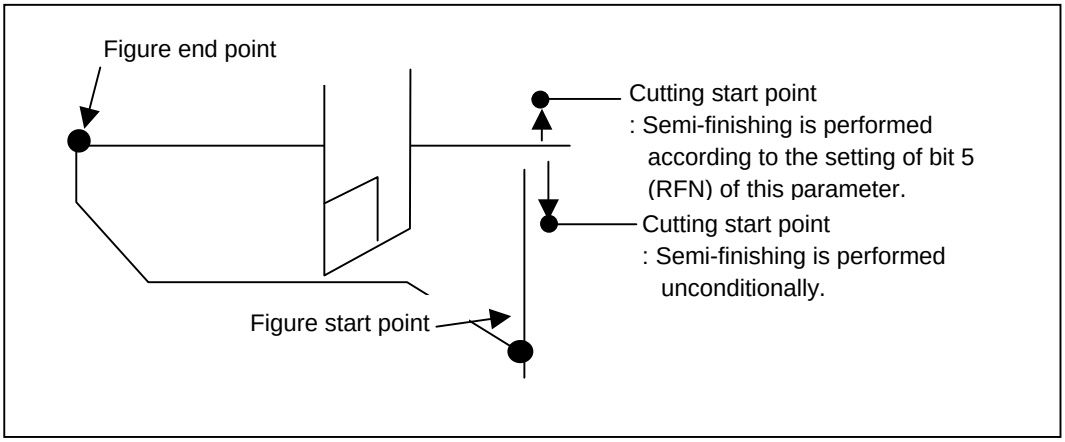
	#7	#6	#5	#4	#3	#2	#1	#0
9310				MTA	NCR		SGT	SFC

SFC = 0 : Performs cutter radius compensation using only the tool tip radius for an arc created in bar machining. The finishing allowance is used as a compensation shift amount.
= 1 : Both the tool tip radius and the finishing allowance are used in the above compensation.
SGT = 0 : Disables cutting edge compensation.
= 1 : Enables cutting edge compensation.
NCR = 0 : G41 and G42 are used in a bar machining finishing cycle.
= 1 : Neither G41 nor G42 is used.
MTA = 0 : Performs multiple types of threading by shifting the start point of each screw.
= 1 : Performs multiple types of threading by specifying angle (Q).

	#7	#6	#5	#4	#3	#2	#1	#0
9312			RFN					

RFN = 0 : Always performs semi-finishing.
= 1 : Does not perform semi-finishing.

NOTE
As shown below, semi-finishing is performed according to the start point and end point of cutting of the entered figure regardless of the setting of bit 5 (RFN) of this parameter.



9350	STGECF

STFECF Override value of cutting feed at the start of cutting in reaming
[Valid data range] 0 to 255
[Unit of data] 1%

9351	RIMKLN

RIMKLN Automatic determination value of the cutting lip length at the start of cutting in reaming
When no cutting lip length (Q) is specified in a reaming command (G1103) block, this parameter is used.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9353	S1TTMN

S1TTMN M-code output before normal tapping in turning
[Valid data range] 0 to 999
[Unit of data] 1%
When this parameter is set to 0, nothing is output.

9354	S1TTMR

S1TTMR M-code output before reverse tapping in turning
[Valid data range] 0 to 999
[Unit of data] 1%
When using reverse tapping in turning, set this parameter to an non-zero M code value.

9359	FFROVD

FFAOVD Feedrate override during concurrent (rough) end facing
When this parameter is set to 0, 100% is assumed.
[Valid data range] 0 to 255
[Unit of data] 1%
[Related parameter] No. 9306#3

9360	FFAOVD

FFAOVD Cutting override during concurrent (rough) end facing
When this parameter is set to 0, 100% is assumed.
[Valid data range] 0 to 255

[Unit of data] 1%
[Related parameter] No. 9306#3

9361	MBF1EX

MBF1EX M code before execution of cycle cutting
When this parameter is set to 0, nothing is output.
[Valid data range] 0 to 255
[Unit of data] 1%

9362	CLRNEX

CLRNEK Clearance used for finishing of unnecked part
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9363	ESCNEX

ESCNEK Escape amount used for finishing of unnecked part
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9364	SFCLRX

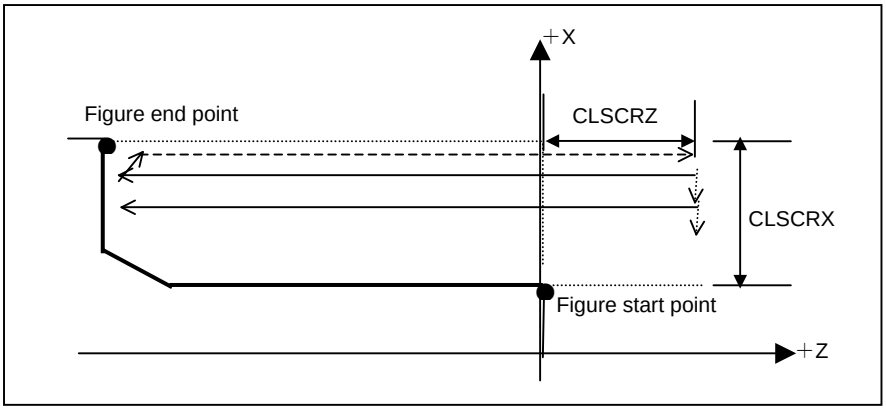
SFCLRX Clearance (diameter value) in the X-axis direction in bar machining
Set this parameter to a value equal to or greater than the tool-nose radius compensation.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9365	SFCLRZ

SFCLRZ Clearance in the Z-axis direction in bar machining
Set this parameter to a value equal to or greater than the tool-nose radius compensation.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9366	ZEROPT

ZEROPT = 0: The program zero point is located on the workpiece end face.
= 1: The program zero point is located on the chuck end face.



9369	RELFX

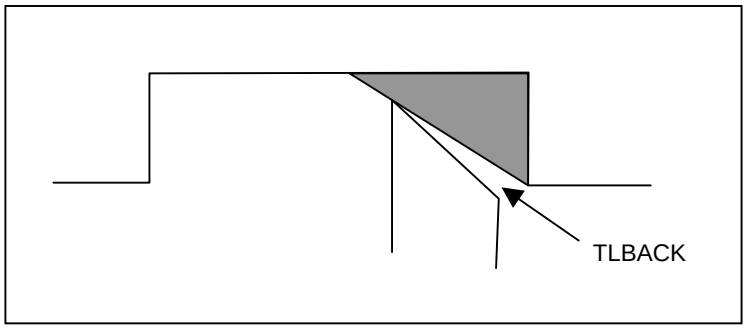
RELFX X component (diameter value) of escape amount from the cutting surface in in-feed machining of outer/inner bar machining
When no escape amount (E) is specified in an outer/inner bar machining command (G112x) block, this parameter is used.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9370	RELFZ

RELFZ Z component (radius value) of escape amount from the cutting surface in in-feed machining of outer/inner bar machining
When no escape amount (E) is specified in an outer/inner bar machining command (G112x) block, this parameter is used.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9371	TLBACK

TLBACK Angle between the tool back and the workpiece surface in intermediate cutting
[Valid data range] 0 to 180
[Unit of data] 1 degree



9372	PCOVR1

PCOVR1 Feed amount override when the tool cutting angle is greater than 90° and equal to or less than 135°

9373	PCOVR2

PCOVR2 Feed amount override when the tool cutting angle is greater than 135° and less than 180°

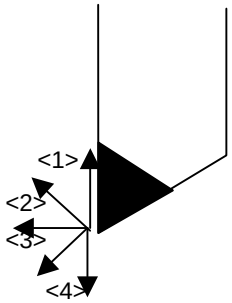
9374	PCOVR3

PCOVR3 Feed amount override when the tool cutting angle is greater than 180° and equal to or less than 225°

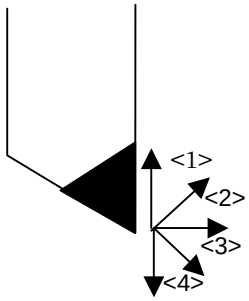
9375	PCOVR4

PCOVR4 Feed amount override when the tool cutting angle is greater than 225° and less than 270°
[Valid data range] 0 to 20
[Unit of data] 10%

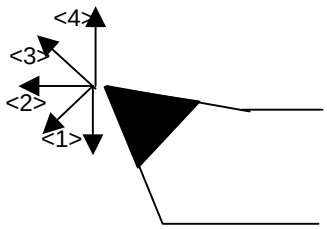
NOTE
When the tool cutting angle is 90°, 180°, or 270°, override is invalid.
When any of parameters No. 9372 to 9375 is set to 0, override is invalid.



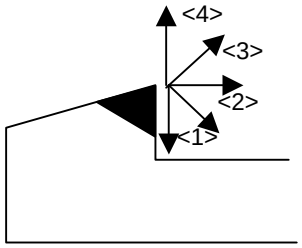
Outer surface (forward direction)



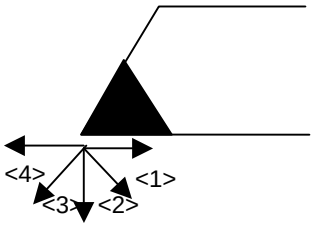
Outer surface (reverse direction)



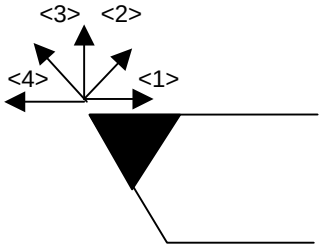
Inner surface (forward direction)



Inner surface (reverse direction)



End face (forward direction)



End face (reverse direction)

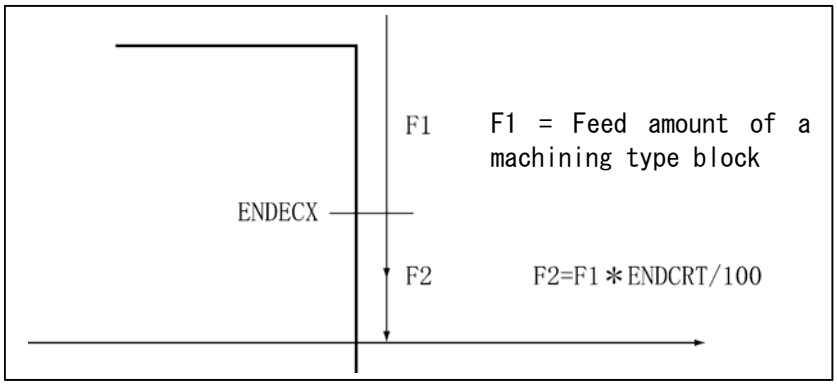
<1>: 90° < Tool cutting angle ≦ 135°
<2>: 135° < Tool cutting angle < 180°
<3>: 180° < Tool cutting angle ≦ 225°
<4>: 225° < Tool cutting angle < 270°

9377	ENDECX

ENDECX X coordinate for reduction of a feed amount in concurrent (rough) end facing
This parameter is valid when ENDECX is greater than (figure start point X - tool nose diameter) in outer surface rough machining.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9378	ENDCRT

ENDCRT Feed amount override near the center in concurrent (rough) end facing
This parameter is valid when the end face cutting allowance is specified.
When this parameter is set to 0, 100% is assumed.
[Valid data range] 0 to 100
[Unit of data] %



9379	CUTANG

CUTANG Cutting edge angle
When no cutting edge angle (V) is specified in a bar machining command (G112x) block, this parameter is used.
[Valid data range] 0 to 1800
[Unit of data] 0.1 degree

9380	TLNANG

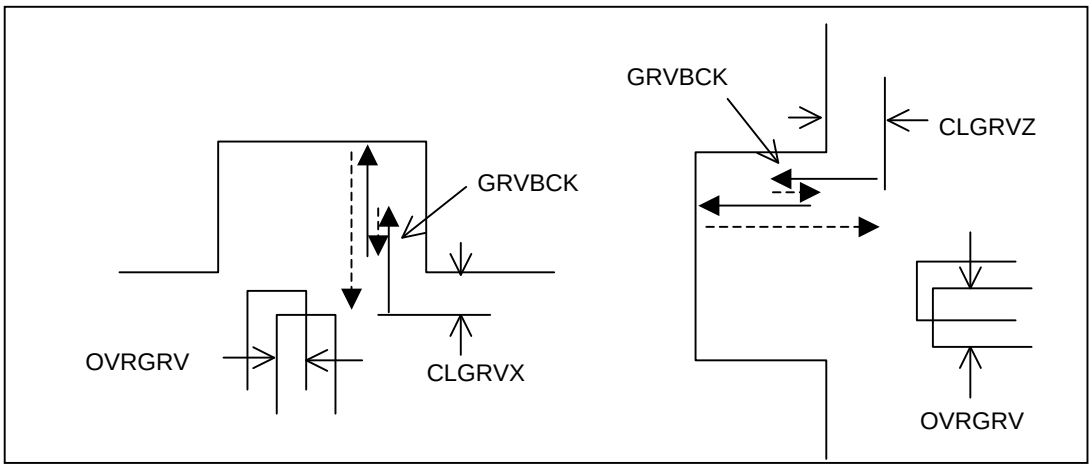
TLNANG Tool angle
When no tool angle (A) is specified in a bar machining command (G112x) block, this parameter is used.
[Valid data range] 0 to 1800
[Unit of data] 0.1 degree

9381	CLGRVX

CLGRVX Clearance (diameter value) in the X-axis direction in outer groove machining and inner groove machining
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9382	CLGRVZ

CLGRVZ Clearance (radius value) in the Z-axis direction in end face grooving
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch



9385	GRVBCK

GRVBCK Pecking return amount (radius value) of grooving
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9386	OVLGRV

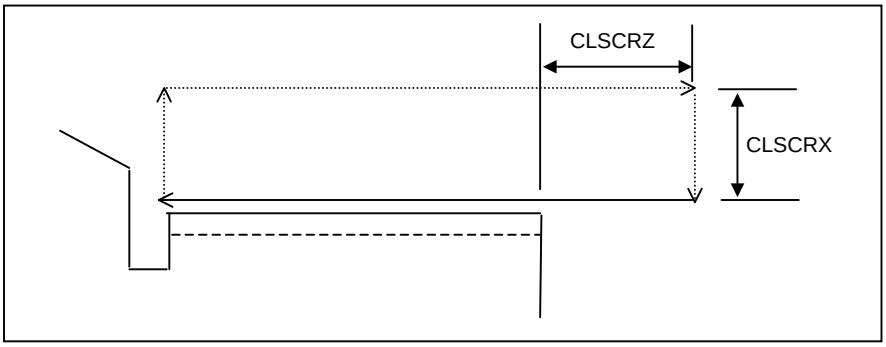
OVLGRV Amount of overlap between grooves in grooving (percentage relative to the tool width)
[Valid data range] 0 to 100
[Unit of data] %

9387	CLSCRX

CLSCRX Clearance in the X-axis direction in threading (diameter value)
When clearance X (C) is not specified in a threading command (G114x) block, this parameter is used.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9388	CLSCRZ

CLSCRZ Clearance in the Z-axis direction in threading
When clearance Z (M) is not specified in a threading command (G114x) block, this parameter is used.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch



9389	TRDMIN

TRDMIN Minimum cutting amount (radius value) in threading
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9390	TRDFNX

TRDFNX Finishing allowance in the X-axis direction in threading (diameter value)
When finishing allowance X (H) is not specified in a threading command (G114x) block, this parameter is used.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9393	DRLDEC

DRLDEC Cutting decrement amount (radius value) in drilling (peck drilling, high speed peck drilling)
The cutting amount (Q) specified in a drilling command (G1101) block is decremented by DRLDEC.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9394	DRLRET

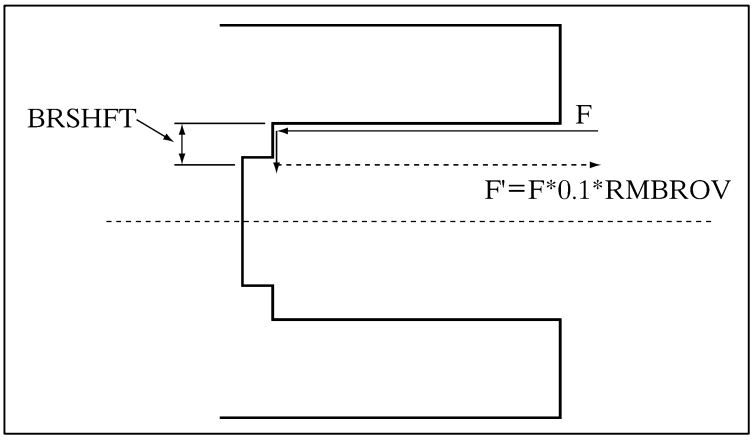
DRLRET Retract amount (radius value) in drilling (peck drilling, high speed peck drilling)
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9395	DRLMIN

DRLMIN Minimum cutting amount (radius value) in drilling (peck drilling, high speed peck drilling)
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch

9396	BRSHFT

BRSHFT Initial setting of a shift amount during a return in boring
When the shift amount during a return is not specified in a boring command (G1104) block, this parameter is used.
[Valid data range] 0 to 99999999
[Unit of data] 0.001mm/0.0001inch



9397	RMROV

RMROV Feed amount (feedrate) override during a return in reaming
F'=F*RMROV/100
In reaming, a return is made at this speed F'
When this parameter is set to 0, an override of 100% is assumed.
[Valid data range] 0 to 200
[Unit of data] 1%

C LINKING MACRO CREATED BY MACHINE TOOL BUILDER WITH MANUAL GUIDE 0*i* MACRO

A macro program created by the machine tool builder can be created and integrated as a user module (MEM file) that differs from a software module for MANUAL GUIDE 0*i*.
After creating a user module, the machine tool builder is to write the module to the FROM with the boot system.

The method of creating a user module is described below.

C.1 CREATING A MACRO PROGRAM

The machine tool builder uses the divided module function of the macro executor to create a macro program.
Specifically, create the macro program of the second or third module in the same way as in the 0*i* -TD/MD.
Set the P-code macro numbers of a macro program created by the machine tool builder in the following parameters.
(Up to two P-code macro numbers can be used.)

9055	MD2PNO

MD2PNO P-CODE macro number of the second module
[Valid data range] 1 to 4
[Unit of data] 1

9056	MD3PNO

MD2PNO P-CODE macro number of the third module
[Valid data range] 1 to 4
[Unit of data] 1

C.2 RESTRICTIONS ON MACRO PROGRAM NUMBERS

The machine tool builder can use only the following macro program numbers to create a macro program,.

(1) Macro program numbers allowed for the M series

O1300 to O1989
O2000 to O2999
O3300 to O4999
O5100 to O5389
O5400 to O5999
O6900 to O7199
O8700 to O8899
O9001 to O9009
O9011 to O9029
O9060 to O9119
O9641 to O9699

O9920 to O9996

(2) Macro program numbers allowed for the T series

O9001 to O9009
O9011 to O9029
O9060 to O9569
O9920 to O9996

C.3 RESTRICTIONS ON MACRO VARIABLES

There are no restrictions on macro variables.

C.4 CUSTOM SOFTWARE CAPACITY

When creating a macro program, the machine tool builder needs to purchase the custom software capacity option for the macro program.

D CUSTOMIZATION OF MANUAL GUIDE 0*i* BY THE MACHINE TOOL BUILDER

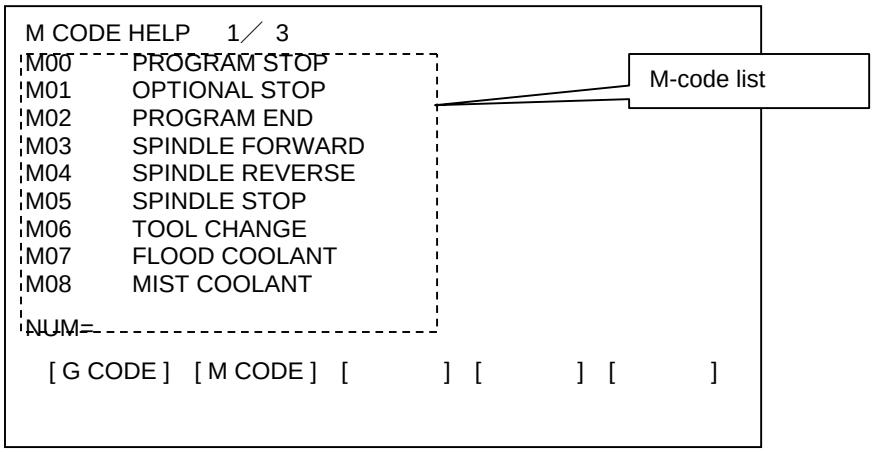
This chapter is the auxiliary explanation on the customization of “M code support function”.

D.1 AUXILIARY EXPLANATION ON THE CUATOMIZATION OF “M CODE SUPPORT FUNCTION”

It is possible to customize the following in “M code support function”.
(1) Customizing M code help screen
(2) Customizing M code text screen

D.1.1 Customizing M Code Help Screen

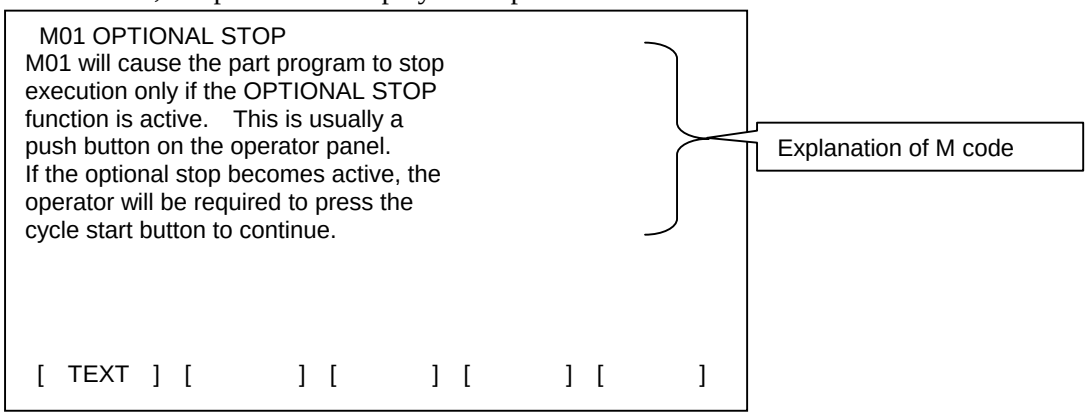
On the M-code help screen, it’s possible to display user’s own M-code list.



（Example of Mcode Help screen ）

D.1.2 Customizing M Code Text Screen

On the M code Help screen, entering number of M code to “NUM=” and depressing [INPUT] key, M Code Text screen is displayed.
On the M Code Text screen, it’s possible to display the explanation of user’s own M code.



（Example of Mcode Text screen）

D.2 AUXILIARY EXPLANATION ON THE CUSTOMIZING METHOD OF "M CODE SUPPORT FUNCTION"

This section is the auxiliary explanation on the customizing method of "M Code Support Function" for MANUAL GUIDE 0i.
This customization is done by making conversational macro program.

NOTE
1 There is restriction on the macro program number which MTB can use. Refer to section "C.2 RESTRICTIONS ON MACRO PROGRAM NUMBER".
2 On the specifications of system variables (#8xxx) and command of screen display such as G243, refer to "FANUC Series 0i/0i-Mate MODEL D Macro Executor PROGRAMMING MANUAL"(B-64303EN-2)

D.2.1 Customizing Method of M Code Help Screen

D.2.1.1 Displaying Method of M Code List

Making macro programs for displaying user's own M code list

(1) Example of macro program for displaying the 1st page of M code list

```
O 8917;
    M98 Pxxxx          <-- Changing language for display (See D.2.1.3)
    G240 P 6 L0C0      <-- Specify color of display message
    G243 X1 Y1 P1168
    G243 X1 Y2 P1169
    G243 X1 Y3 P1170
    G243 X1 Y4 P1171   <-- (a) Specify display position "X_Y_" of M code.
    G243 X1 Y5 P1172       (b) As "Pnnnn", specify sequence number of message
    G243 X1 Y6 P1173           defined in the other program.（See D.2.1.2）
    G243 X1 Y7 P1174
    G243 X1 Y8 P1175
    G243 X1 Y9 P1176
    M98 P8959          <-- Make sure program this line
    M99
```

(2) Example of macro program for displaying the 2nd page of M code list

```
O 8918;
    M98 Pxxxx
    G240 P 6 L0C0
    G243 X1 Y1 P1177
    G243 X1 Y2 P1178
    G243 X1 Y3 P1179
    G243 X1 Y4 P1180
    G243 X1 Y5 P1181
    G243 X1 Y6 P1182
    G243 X1 Y7 P1183
    G243 X1 Y8 P1184
    G243 X1 Y9 P1185
    M98 P8959
    M99
```

(3) Example of macro program for displaying the 3rd page of M code list

```
O 8919;
        M98 Pxxxx
        G240 P 6 L0C0
        G243 X1 Y1 P1186
        G243 X1 Y2 P1187
        G243 X1 Y3 P1188
        G243 X1 Y4 P1189
        M98 P8959
        M99
```

NOTE
1 Number of M-code which can be displayed on one page of M-code list is within 9.
2 Number of page of M-code list is within 3 pages.
3 M-code number which can be specified is M00 through M99.

D.2.1.2 Defining Message to Display on the M Code List

Message displayed on the M-code list is defined by macro program.
Decide the macro program number by MTB.

(1) Example of Japanese message

```
Oyyyy;
        N1168 (' M00 ﾌﾟﾛｸﾞﾗﾑｽﾄｯﾌﾟ')
        N1169 (' M01 ｵﾌﾟｼｮﾅﾙｽﾄｯﾌﾟ')
        N1170 (' M02 ﾌﾟﾛｸﾞﾗﾑｴﾝﾄﾞ')
        N1171 (' M03 主軸正転')
        N1172 (' M04 主軸逆転')
        N1173 (' M05 主軸停止')
        N1174 (' M06 工具交換')
        N1175 (' M07 液ｸｰﾗﾝﾄ')
        N1176 (' M08 ﾐｽﾄｸｰﾗﾝﾄ')
        N1177 (' M09 ｸｰﾗﾝﾄｵﾌﾞ')
        N1178 (' M19 主軸ｵﾘｴﾝﾃｰｼｮﾝ')
        N1179 (' M30 ﾌﾟﾛｸﾞﾗﾑｽﾄｯﾌﾟ とﾘﾜｲﾝﾄﾞ')
        N1180 (' M48 主軸速度ｵｰﾊﾞｰﾗｲﾄﾞ有効')
        N1181 (' M49 主軸速度ｵｰﾊﾞｰﾗｲﾄﾞ 100%')
        N1182 (' M67 ﾓｰﾀｰ 1 起動')
        N1183 (' M68 ﾓｰﾀｰ 2 起動')
        N1184 (' M69 ﾓｰﾀｰ 3 起動')
        N1185 (' M77 ﾓｰﾀｰ 1 停止')
        N1186 (' M78 ﾓｰﾀｰ 2 停止')
        N1187 (' M79 ﾓｰﾀｰ 3 停止')
        N1188 (' M98 ｻﾌﾞﾌﾟﾛｸﾞﾗﾑ呼び出し')
        N1189 (' M99 ｻﾌﾞﾌﾟﾛｸﾞﾗﾑ戻り')
        N9999 M99
```

Inside of "( )" belong the sequence number Nxxxx, define message displayed on the 1st page of M-code list

Inside of "( )" belong the sequence number Nxxxx, define message displayed on the 2nd page of M-code list

Inside of "( )" belong the sequence number Nxxxx, define message displayed on the 3rd page of M-code list

* About the example of message that Manual Guide 0i displays in each language, refer to the section "D.2.4 Ample of Messages in Each Language"

D.2.1.3 Changing Message according to the selected display language

Making macro program for changing display language of message.
According to the display language, set the macro program number of defining message corresponding to the display language to the macro system valiable #8509.
Decide the macro program number by MTB.

(1) Example of macro program for changing display language of the message

```
Oxxxx;
        #8509 = 8700              /* Set the macro program number of defining English message.
N100    IF [P3281 EQ 1] THEN      /* When Japanese is selected,
          #8509 = 8701            /* set the macro program number of defining Japanese message.
          GOTO 999
        ENDIF
        IF [P3281 EQ 4] THEN      /* Whe Chinese (Traditional character) is selected,
          #8509 = 8702            /* set the macro program number of defining Chinese message.
          GOTO 999
        ENDIF
N101    IF [P3281 EQ 2] THEN      /* When Germany is selected,
          #8509 = 8703            /* set the macro program number of defining German message.
          GOTO 999
        ENDIF
        IF [P3281 EQ 3] THEN      /* When French is selected,
          #8509 = 8704            /* set the macro program number of defining French message.
          GOTO 999
        ENDIF
N102    IF [P3281 EQ 5] THEN      /* When Italian is selected,
          #8509 = 8705            /* set the macro program number of defining Italian message.
          GOTO 999                /*
        ENDIF
        IF [P3281 EQ 7] THEN      /* When Spanish is selected,
          #8509 = 8706            /* set the macro program number of defining Spanish message.
          GOTO 999                /*
        ENDIF
N103    IF [P3281 EQ 13] THEN     /* When Sweidish is selected,
          #8509 = 8707            /* set the macro program number of defining Swedish message.
          GOTO 999                /*
        ENDIF
        IF [P3281 EQ 10] THEN     /* When Portigue is selected,
          #8509 = 8708            /* set the macro program number of defining Portigue message.
          GOTO 999                /*
        ENDIF
N104    IF [P3281 EQ 14] THEN     /* When Czech is selected,
          #8509 = 8709            /* set the macro program number of defining Czech message.
          GOTO 999                /*
        ENDIF
        IF [P3281 EQ 11] THEN     /* When Polish is selected,
          #8509 = 8710            /* set the macro program number of defining Polish message.
          GOTO 999                /*
        ENDIF
N999    M99
```

D.2.1.4 Displaying Method of M Code Text Screen from M Code Help Screen

Making macro program O8975 for displaying M-code text screen from M-code help screen.

(1) Example of macro program O8975

```
O8975;
        #10 = #8503
        IF [#10 EQ 0]          GOTO 1000
        IF [#10 EQ 1]          GOTO 1000
        IF [#10 EQ 2]          GOTO 1000
        IF [#10 EQ 3]          GOTO 1000
        IF [#10 EQ 4]          GOTO 1000
        IF [#10 EQ 5]          GOTO 1000
        IF [#10 EQ 6]          GOTO 1000
        IF [#10 EQ 7]          GOTO 1000
        IF [#10 EQ 8]          GOTO 1000
        IF [#10 EQ 9]          GOTO 1000
        IF [#10 EQ 19]         GOTO 1000
        IF [#10 EQ 30]         GOTO 1000
        IF [#10 EQ 48]         GOTO 1000
        IF [#10 EQ 49]         GOTO 1000
        IF [#10 EQ 67]         GOTO 1000
        IF [#10 EQ 68 || #10 EQ 69] THEN
          #10 = 67
          GOTO 1000
        ENDIF
        IF [#10 EQ 77]         GOTO 1000
        IF [#10 EQ 78 || #10 EQ 79] THEN
          #10 = 77
          GOTO 1000
          ENDIF
        IF [#10 EQ 98]         GOTO 1000
        IF [#10 EQ 99]         GOTO 1000
          GOTO 9000
N1000   #96832 = #10
        M99
N9000   #96832 = -1
        M99
```

Read key-inned M-code number

Check key-inned M-code number.
If the M-code number is available, go to the N1000.
If it is not available, go to the N9000.

If key-inned M-code number is available, set the M-code number to the P-code valiable #96832.

If key-inned M-code number is not available, set "-1" to the P-code valiable #96832.

D.2.2 Customizing Method of M Code Text Screen

D.2.2.1 Method od Displaying Title of M Code Explanation

Making macro program for displaying the title of M-code explanation and the others.
Use O86xx as the macro program number. And specify the M-code number to "xx".
But, in case of specifying M67～M69, specify 67 as "xx". In case of specifying M77～M79, specify 77 as "xx".

(1) Example of macro program for displaying the title of M00 explanation

```
O8600;
      M98 Pxxxx          <-- Change display language of message (see D.2.1.3).
      G240 P 7 L0C0      <-- Specify color of the title of message.
      G243 X0 Y0 P0971   <-- Specify "X0,Y0" as display position of title of M00 explanation.
                             Specify sequence number P9071 that title message of M00 explanation
                             is defined.
      M98 Pyyyy          <-- Display the explanation of M00 (see D.2.2.2)
      #97008 = 8600      <-- Set program number of this macro program.
      M98 P8959
      #97007 = #97006
      #97009 = 0
      #97010 = 0         <-- Make sure program these lines.
      #97011 = 0
      #97012 = 0
      #97000 = 1
N1    M98P 8977
      M99P 1
```

D.2.2.2 Method of Displaying M code Explanation

Making macro program for displaying M code explanation.
Decide the program number of the macro program by MTB.

(1) Example for displaying the explanation of M00

```
Oyyyy;
      G240 P 6 L0C0
      G243 X0 Y1 P0972
      G243 X0 Y2 P0973
      G243 X0 Y3 P0974
      G243 X0 Y4 P0975
      M99
```

In order to display the explanation of M00, specify "G243X_Y_Pmmmm". Specify Pmmmm as the sequence number Nmmmm which the explanation of M00 is defined. Then, specify the display position "X_Y_" for displaying the explanation.

D.2.2.3 Method of Defining Message for M Code Explanation

Defining the message for the explanation of M-code by a macro program.
Decide the program number of the macro program.

(1) Example for defining Japanese message for the explanation of M00

```
Ozzzz;
      N0971 (' M00 ﾌﾟﾛｸﾞﾗﾑｽﾄｯﾌﾟ  ')
      N0972 ('M00 はﾊﾟｰﾄﾌﾟﾛｸﾞﾗﾑの実行･･')
      N0973 ('運転を続けるためには、･･  ')
      N0974 ('ことが必要になります。    ')
      N0975 ('                          ')
      N9999 M99
```

To the inside of "( )" follows the sequence number N0971, specify the title for the explanation of M00.

To the inside of "( )" follows the sequence number N0972 ～N0975, specify the explanation of M00.

* About the example of message that Manual Guide 0i displays in each language, refer to the section "D.2.4 Ample of Messages in Each Language"

D.2.3 Building Customized Conversational Macro Programs

Macro programs which are created by MTB are built in CNC as the divided module.
In the details, refer to the chapter “C LINKING MACRO CREATED BY MACHINE TOOL BUILDER WITH MANUAL GUIDE 0***i*** MACRO”.

D.2.4 Sample of Messages in Each Language

The following is sample of messages displayed in each language on the Manual Guide 0i screen.

D.2.4.1 Sample of Messages in Japanese

N0971 (' M00 ﾌﾟﾛｸﾞﾗﾑｽﾄｯﾌﾟ ')
N0972 ('M00 はﾊﾟｰﾄﾌﾟﾛｸﾞﾗﾑの実行を停止させます。 ')
N0973 ('運転を続けるためには、ｻｲｸﾙｽﾀｰﾄﾎﾞﾀﾝを押す')
N0974 ('ことが必要になります。 ')
N0975 (' ')

N0976 (' M01 ｵﾌﾟｼｮﾅﾙｽﾄｯﾌﾟ ')
N0977 ('ｵﾌﾟｼｮﾅﾙｽﾄｯﾌﾟ機能が有効な時のみ、M01 はﾊﾟｰ')
N0978 ('ﾄﾌﾟﾛｸﾞﾗﾑの実行を停止させます。通常これは')
N0979 ('操作盤のﾎﾞﾀﾝを押します。 ')
N0980 ('ｵﾌﾟｼｮﾅﾙｽﾄｯﾌﾟが有効な時、運転を実行する為')
N0981 ('にはｻｲｸﾙｽﾀｰﾄﾎﾞﾀﾝを押すことが必要になりま')
N0982 ('す。 ')
N0983 (' ')

N0984 (' M02 ﾌﾟﾛｸﾞﾗﾑｴﾝﾄﾞ ')
N0985 ('M02 はﾊﾟｰﾄﾌﾟﾛｸﾞﾗﾑの実行を終了します。 ')
N0986 ('M02 実行の後、運転を再開するにはﾘｾｯﾄｷｰを ')
N0987 ('押すことが必要になります。 ')
N0988 (' ')

N0989 (' M03 主軸正転 ')
N0990 ('M03 は主軸を時計回り方向に回転させます。 ')
N0991 ('主軸はﾌﾟﾛｸﾞﾗﾑされた主軸速度で回転します ')
N0992 ('主軸速度の変更は S ｺｰﾄﾞでされます。 ')
N0993 (' ')

N0994 (' M04 主軸逆転 ')
N0995 ('M04 は主軸を反時計回り方向に回転させます ')
N0996 ('主軸はﾌﾟﾛｸﾞﾗﾑされた主軸速度で回転します ')
N0997 ('主軸速度の変更は S ｺｰﾄﾞでされます。 ')
N0998 (' ')
N0999 (' ')

N1000 (' M05 主軸停止 ')
N1001 ('M05 主軸を停止させます。 ')

N1002 (' M06 工具交換 ')
N1003 ('M06 は機械上の工具交換の要求をします。 ')
N1004 ('工具交換ﾏｸﾛは主軸とｸｰﾗﾝﾄの停止をし、工具')
N1005 ('交換をする為の必要な初期化をします。 ')

```
N1006 ('                                                    ')
N1007 ('                                                    ')
N1008 ('                                                    ')
N1009 ('                                                    ')

N1010 (' M07  液ｸｰﾗﾝﾄｵﾝ                                     ')
N1011 ('M07 は機械の液ｸｰﾗﾝﾄをｵﾝする為の要求です。')
N1012 ('これは切削中の工具が過熱しないようにする')
N1013 ('ため強く冷却するものです。                          ')
N1014 ('                                                    ')

N1015 (' M08  ﾐｽﾄｸｰﾗﾝﾄｵﾝ          ')
N1016 ('M08 は機械のﾐｽﾄｸｰﾗﾝﾄをｵﾝする為の要求です')
N1017 ('これは切削負荷が小さいか、切削中に発生す')
N1018 ('る多くの熱が容易に逃げやすい材質を使用し')
N1019 ('ている場合、軽く冷却するものです。          ')
N1020 ('                                                    ')
N1021 ('                                                    ')

N1022 (' M09  ｸｰﾗﾝﾄｵﾌ          ')
N1023 ('M09 は機械のﾐｽﾄ、液ｸｰﾗﾝﾄの両方をｵﾌするも ')
N1024 ('のです。これは工具がﾜｰｸから離れる時等に ')
N1025 ('使います。                                          ')
N1026 ('                                                    ')
N1027 ('                                                    ')
N1028 ('                                                    ')

N1029 (' M19  主軸ｵﾘｴﾝﾃｰｼｮﾝ要求          ')
N1030 ('M19 はｽﾋﾟﾝﾄﾞﾙｼｬﾌﾄを決められた位置でﾛｯｸす ')
N1031 ('る為のｽﾋﾟﾝﾄﾞﾙｵﾘｴﾝﾃｰｼｮﾝを行う指令です。  ')
N1032 ('この機能は工具交換ｻｲｸﾙ等で使用されます。')
N1033 ('                                                    ')

N1034 (' M30  ﾌﾟﾛｸﾞﾗﾑｽﾄｯﾌﾟとﾘﾜｲﾝﾄﾞ        ')
N1035 ('M30 はﾌﾟﾛｸﾞﾗﾑの実行を停止し、CNC をﾘｾｯﾄ状')
N1036 ('態にし、ﾌﾟﾛｸﾞﾗﾑの頭出しを行うものです。  ')
N1037 ('                                                    ')
N1038 ('                                                    ')
N1039 ('                                                    ')
N1040 ('                                                    ')
N1041 ('                                                    ')
N1042 ('                                                    ')

N1043 (' M48  主軸速度ｵｰﾊﾞｰﾗｲﾄﾞ有効          ')
N1044 ('M48 は操作盤上の主軸ｵｰﾊﾞｰﾗｲﾄﾞを有効にする')
N1045 ('ものです。                                          ')
N1046 ('このｵｰﾊﾞｰﾗｲﾄﾞｽｲｯﾁで、ｵﾍﾟﾚｰﾀはﾌﾟﾛｸﾞﾗﾑを変')
N1047 ('更することなく主軸速度を変えることが出来')
N1048 ('ます。                                              ')
N1049 ('                                                    ')

N1050 (' M49  主軸速度ｵｰﾊﾞｰﾗｲﾄﾞ無効          ')
N1051 ('M49 は操作盤上の主軸ｵｰﾊﾞｰﾗｲﾄﾞを無効にする')
N1052 ('ものです。                                          ')
N1053 ('このﾓｰﾄﾞ中は、ｵﾍﾟﾚｰﾀはﾊﾟｰﾄﾌﾟﾛｸﾞﾗﾑを変更 ')
```

N1054 ('することなく主軸速度を変えることは出来ま')
N1055 ('せん。 ')
N1056 (' ')

N1057 (' M67 - M69 ﾓｰﾀｰ起動指令 ')
N1058 ('M67～M69 で機械に用意されたﾓｰﾀを起動する ')
N1059 ('ものです。 ')
N1060 ('ﾓｰﾀ 1 は M67 で起動され、ﾓｰﾀ 2 は M68 で、ﾓｰﾀ 3 は')
N1061 ('M69 で起動されます。 ')
N1062 (' ')
N1063 (' ')

N1064 (' M77 - M79 ﾓｰﾀｰ停止指令 ')
N1065 ('M77～M79 は M67～M69 を使って起動したﾓｰﾀを ')
N1066 ('停止させるための M ｺｰﾄﾞです。 ')
N1067 ('ﾓｰﾀ 1 は M77 で、ﾓｰﾀ 2 は M78 で、ﾓｰﾀ 3 は M79 で停 ')
N1068 ('止させます。 ')
N1069 (' ')
N1070 (' ')

N1071 (' M98 ｻﾌﾞﾌﾟﾛｸﾞﾗﾑ呼び出し')
N1072 ('M98 P9876 はｻﾌﾞﾌﾟﾛｸﾞﾗﾑ番号 9876 のﾌﾟﾛｸﾞﾗﾑを')
N1073 ('呼び出すものです。ｻﾌﾞﾌﾟﾛｸﾞﾗﾑに引数を使い')
N1074 ('ﾃﾞｰﾀを渡すことは出来ません。ｻﾌﾞﾌﾟﾛｸﾞﾗﾑは')
N1075 ('M99 で終了します。 ')
N1076 ('注意: ｻﾌﾞﾌﾟﾛｸﾞﾗﾑにもし M02、M30 が使われた')
N1077 ('場合、ﾌﾟﾛｸﾞﾗﾑは呼び出されたﾌﾟﾛｸﾞﾗﾑに戻ら')
N1078 ('ず終了してしまいます。 ')

N1079 (' M99 ｻﾌﾞﾌﾟﾛｸﾞﾗﾑﾘﾀｰﾝ ')
N1080 ('M99 がﾌﾟﾛｸﾞﾗﾑ実行中に読み込まれますと、実')
N1081 ('行は呼ばれたﾌﾟﾛｸﾞﾗﾑに戻ります。もし、ﾌﾟﾛ')
N1082 ('ｸﾞﾗﾑがどこからも呼ばれずに M99 を読み込み ')
N1083 ('ますと、ﾌﾟﾛｸﾞﾗﾑは終了することなく頭から ')
N1084 ('繰り返し実行されてしまいます。 ')
N1085 (' ')

N1168 (' M00 ﾌﾟﾛｸﾞﾗﾑｽﾄｯﾌﾟ')
N1169 (' M01 ｵﾌﾟｼｮﾅﾙｽﾄｯﾌﾟ')
N1170 (' M02 ﾌﾟﾛｸﾞﾗﾑｴﾝﾄﾞ')
N1171 (' M03 主軸正転')
N1172 (' M04 主軸逆転')
N1173 (' M05 主軸停止')
N1174 (' M06 工具交換')
N1175 (' M07 液ｸｰﾗﾝﾄ')
N1176 (' M08 ﾐｽﾄｸｰﾗﾝﾄ')

N1177 (' M09 ｸｰﾗﾝﾄｵﾌ')
N1178 (' M19 主軸ｵﾘｴﾝﾃｰｼｮﾝ')
N1179 (' M30 ﾌﾟﾛｸﾞﾗﾑｽﾄｯﾌﾟ とﾘﾜｲﾝﾄﾞ')
N1180 (' M48 主軸速度ｵｰﾊﾞｰﾗｲﾄﾞ 有効')
N1181 (' M49 主軸速度ｵｰﾊﾞｰﾗｲﾄﾞ 100%')
N1182 (' M67 ﾓｰﾀｰ 1 起動')

```
N1183 (' M68 ﾓｰﾀｰ 2 起動')
N1184 (' M69 ﾓｰﾀｰ 3 起動')
N1185 (' M77 ﾓｰﾀｰ 1 停止')
N1186 (' M78 ﾓｰﾀｰ 2 停止')
N1187 (' M79 ﾓｰﾀｰ 3 停止')
N1188 (' M98 ｻﾌﾞﾌﾟﾛｸﾞﾗﾑ呼び出し')
N1189 (' M99 ｻﾌﾞﾌﾟﾛｸﾞﾗﾑ戻り')
```

D.2.4.2 Sample of Messages in English

```
N0971 (' M00 PROGRAM STOP ')
N0972 ('M00 will cause the part program to stop ')
N0973 ('execution.  In order to continue, the  ')
N0974 ('operator will be required to press the  ')
N0975 ('cycle start key.                        ')
N0976 (' M01 OPTIONAL STOP ')
N0977 ('M01 will cause the part program to stop ')
N0978 ('execution only if the OPTIONAL STOP     ')
N0979 ('function is active.  This is usually a  ')
N0980 ('push button on the operator panel.      ')
N0981 ('If the optional stop becomes active, the')
N0982 ('operator will be required to press the  ')
N0983 ('cycle start button to continue.         ')
N0984 (' M02 PROGRAM END ')
N0985 ('M02 will cause the part program to      ')
N0986 ('terminate execution.  After execution of')
N0987 ('an M02, the operator must press RESET to')
N0988 ('re-start execution.                     ')
N0989 (' M03 SPINDLE CW ')
N0990 ('M03 requests starting the spindle in the')
N0991 ('clockwise direction.  The spindle will  ')
N0992 ('run at the programmed spindle speed.    ')
N0993 ('Changing speed is done with an S-code.  ')
N0994 (' M04 SPINDLE CCW ')
N0995 ('M04 requests starting the spindle in the')
N0996 ('counter-clockwise direction.  The       ')
N0997 ('spindle will run at the programmed      ')
N0998 ('spindle speed.  Changing the speed is   ')
N0999 ('done with an S-code.                    ')
N1000 (' M05 SPINDLE STOP ')
N1001 ('M05 requests stopping the spindle.      ')
N1002 (' M06 TOOL CHANGE ')
N1003 ('M06 requests a tool change on the       ')
N1004 ('machine.  The tool change macro should  ')
N1005 ('stop the spindle and coolant, and then  ')
N1006 ('initiate the necessary actions to       ')
N1007 ('perform the tool change.                ')
N1008 ('At the end of the tool change, the      ')
N1009 ('spindle remains turned off.             ')
N1010 (' M07 FLOOD COOLANT ON ')
N1011 ('M07 requests turning on the flood       ')
```

```
N1012 ('coolant on the machine.    This is a heavy')
N1013 ('flow of coolant intended to keep the       ')
N1014 ('cutting tool from overheating.             ')

N1015 (' M08 MIST COOLANT ON ')
N1016 ('M08 requests turning on the mist           ')
N1017 ('coolant on the machine.    This is a light')
N1018 ('flow of coolant for use when cutting       ')
N1019 ('forces are low, or the material easily     ')
N1020 ('dissipates much of the heat generated      ')
N1021 ('during cutting cycles.                     ')

N1022 (' M09 COOLANT OFF ')
N1023 ('M09 turns off both flood and mist          ')
N1024 ('coolant on the machine.    This function   ')
N1025 ('is useful when the tool is away from       ')
N1026 ('the part and requires inspection.          ')
N1027 ('M05 and M06 should also turn off the       ')
N1028 ('coolant, but that cannot be guaranteed. ')

N1029 (' M19 SPINDLE ORIENT REQUEST ')
N1030 ('M19 requests orienting the spindle         ')
N1031 ('in order to lock the spindle shaft in a ')
N1032 ('known position.    This feature is useful ')
N1033 ('during tool change cycles.                 ')

N1034 (' M30 PROGRAM STOP AND REWIND ')
N1035 ('M30 stops execution of the current         ')
N1036 ('part program and issues a reset and        ')
N1037 ('rewind request to the CNC.    This request')
N1038 ('is also passed to the machine control      ')
N1039 ('software to reset modal M-codes.           ')
N1040 ('Execution of the part program from the     ')
N1041 ('beginning is possible simply by pressing')
N1042 ('cycle start.                                ')

N1043 (' M48 SPDL SPEED OVERRIDE ALLOWED ')
N1044 ('M48 enables the functionality of the       ')
N1045 ('spindle override selector on the           ')
N1046 ('operator panel.    With this selector,     ')
N1047 ('the operator can modify the spindle        ')
N1048 ('speed without modifying the part           ')
N1049 ('program.                                    ')

N1050 (' M49 SPDL SPEED OVERRIDE NOT ALLOWED ')
N1051 ('M49 inhibits the functionality of the      ')
N1052 ('spindle override selector on the           ')
N1053 ('operator panel.    In this mode, the       ')
N1054 ('operator is not permitted to modify        ')
N1055 ('the spindle speed without changing         ')
N1056 ('the part program.                           ')

N1057 (' M67 - M69 Motor Start Commands ')
N1058 ('M67 through M69 are M-codes provided to ')
N1059 ('turn on a device on the machine.    These ')
N1060 ('devices may commonly be referred to as     ')
N1061 ('motors.    Motor 1 is started by           ')
N1062 ('programming M67, motor 2 with M68, and     ')
N1063 ('motor 3 with M69.                           ')
```

```
N1064 (' M77 - M79 Motor Stop Commands ')
N1065 ('M77 through M79 are M-codes provided to ')
N1066 ('turn off a device on the machine.    These')
N1067 ('devices may commonly be referred to as    ')
N1068 ('motors.    Motor 1 is stopped by                ')
N1069 ('programming M77, motor 2 with M78, and    ')
N1070 ('motor 3 with M79.                                  ')

N1071 (' M98 SUBPROGRAM CALL ')
N1072 ('M98 P9876 WILL CALL SUBPROGRAM NUMBER      ')
N1073 ('9876. ARGUMENTS CANNOT BE PASSED TO THE ')
N1074 ('SUBPROGRAM. THE SUBPROGRAM CALLED SHOULD')
N1075 ('TERMINATE WITH AN M99.                          ')
N1076 ('CAUTION: IF M02 OR M30 ARE USED IN THE    ')
N1077 ('SUBPROGRAM, THE PROGRAM WILL NOT RETURN ')
N1078 ('TO THE CALLING PROGRAM AND TERMINATE.      ')

N1079 (' M99 RETURN FROM SUBPROGRAM ')
N1080 ('WHEN M99 IS ENCOUNTERED DURING PROGRAM    ')
N1081 ('EXECUTION, IT WILL RETURN TO THE CALLING')
N1082 ('PROGRAM. IF THE PROGRAM WAS EXECUTED       ')
N1083 ('WITHOUT A CALLING PROGRAM, THE PROGRAM    ')
N1084 ('WILL LOOP TO THE BEGINNING AND REPEAT      ')
N1085 ('INDEFINITELY.                                      ')

N1168 (' M00 PROGRAM STOP')
N1169 (' M01 OPTIONAL STOP')
N1170 (' M02 PROGRAM END')
N1171 (' M03 SPINDLE CW')
N1172 (' M04 SPINDLE CCW')
N1173 (' M05 SPINDLE STOP')
N1174 (' M06 TOOL CHANGE')
N1175 (' M07 FLOOD COOLANT')
N1176 (' M08 MIST COOLANT')

N1177 (' M09 COOLANT OFF')
N1178 (' M19 SPINDLE ORIENT')
N1179 (' M30 PROGRAM STOP AND REWIND')
N1180 (' M48 SPDL SPEED OVERRIDE ALLOWED')
N1181 (' M49 SPDL SPEED OVERRIDE 100 PERCENT')
N1182 (' M67 START MOTOR 1')
N1183 (' M68 START MOTOR 2')
N1184 (' M69 START MOTOR 3')
N1185 (' M77 STOP MOTOR 1')

N1186 (' M78 STOP MOTOR 2')
N1187 (' M79 STOP MOTOR 3')
N1188 (' M98 SUBPROGRAM CALL')
N1189 (' M99 RETURN FROM SUBPROGRAM')
```

D.2.4.3 Sample of Messages in Germany

```
N0971 (' M00 PROGRAMMHALT')
N0972 ('M00 veranlasst das Teileprogramm, die')
N0973 ('Ausfuhrung anzuhalten. Um die Ausfuhrung')
N0974 ('danach fortzusetzen, muss die Zyklus-')
N0975 ('start-Taste betatigt werden.                ')
```

```
N0976 (' M01 OPTIONALER HALT')
N0977 ('M01 veranlasst das Teileprogramm, die')
N0978 ('Ausfuhrung dann anzuhalten, wenn die')
N0979 ('Funktion "Optionaler Halt" aktiv ist.')
N0980 ('Normalerweise ist das ein Drucktaster am')
N0981 ('Bedienfeld. Nach einem Optionalen Halt)
N0982 ('muss die Zyklusstart-Taste gedruckt')
N0983 ('werden, um die Ausfuhrung fortzusetzen.')

N0984 (' M02 PROGRAMMENDE ')
N0985 ('M02 veranlasst das Teileprogramm, die')
N0986 ('Ausfuhrung zu beenden. Um die Ausfuhrung')
N0987 ('danach neu zu starten, muss RESET)
N0988 ('gedruckt werden.                          ')

N0989 (' M03 SPINDEL IM US')
N0990 ('M03 veranlasst die Spindeldrehung im')
N0991 ('Uhrzeigersinn. Die Spindel dreht mit der')
N0992 ('programmierten Spindeldrehzahl.')
N0993 ('Ein S-Code andert die Spindeldrehzahl.')

N0994 (' M04 SPINDEL GEGEN US')
N0995 ('M04 veranlasst die Spindeldrehung')
N0996 ('gegen den Uhrzeigersinn. Die Spindel')
N0997 ('dreht mit der programmierten Spindel-')
N0998 ('drehzahl. Zum Andern der Spindeldreh-')
N0999 ('zahl verwenden Sie einen S-Code.')

N1000 (' M05 SPINDELHALT ')
N1001 ('M05 halt die Spindel an.    ')

N1002 (' M06 WERKZEUGWECHSEL ')
N1003 ('M06 veranlasst einen Werkzeugwechsel an')
N1004 ('der Maschine. Ein Werkzeugwechsel-Makro')
N1005 ('sollte Spindel und Kuhlmittel stoppen')
N1006 ('und alle weiteren Vorbereitungen fur den')
N1007 ('Werkzeugwechsel einleiten.')
N1008 ('Nach dem Werkzeugwechsel bleibt die')
N1009 ('Spindel ausgeschaltet.                    ')

N1010 (' M07 SPULKUHLMITTEL EIN')
N1011 ('M07 schaltet die Kuhlmittelspulung an')
N1012 ('der Maschine ein, einen starken)
N1013 ('Kuhlmittelstrom, der das Schneidwerkzeug')
N1014 ('vor Uberhitzung schutzen soll.          ')

N1015 (' M08 KUHMITTELNEBEL EIN')
N1016 ('M08 fordert Kuhlmittelnebel von der')
N1017 ('Maschine an, einen leichten Spruhstrom,')
N1018 ('wenn nur geringe Schneidkrafte wirken')
N1019 ('oder das Material die wahrend der')
N1020 ('Bearbeitung erzeugte Hitze ohne')
N1021 ('Probleme ableiten kann.')

N1022 (' M09 KUHLMITTEL AUS')
N1023 ('M09 schaltet sowohl Spul- als auch')
N1024 ('Spruhkuhlmittel aus. Wird normalerweise')
N1025 ('eingesetzt, wenn das Werkstuck zur')
N1026 ('Inspektion vom Werkstuck abgesetzt ist.')
N1027 ('Kuhlmittelabschaltung ist auch mit M05')
```

```
N1028 ('und M06 moglich, aber nicht so sicher.')

N1029 (' M19 SPINDELORIENTIERUNG')
N1030 ('M19 fordert eine Spindelorientierung an,')
N1031 ('bei der die Spindelwelle in einer')
N1032 ('bekannten Position arretiert.')
N1033 ('Hilfreich beim Werkzeugwechsel.')

N1034 (' M30 PROGRAMM ANHALTEN + ANFANG SUCHEN')
N1035 ('M30 stoppt die Ausfuhrung des aktuellen')
N1036 ('Teileprogramms und lost ein "Reset &')
N1037 ('Rucksetzen" aus. Die Anforderung wird')
N1038 ('auch an die Maschinensteuerung gegeben,')
N1039 ('damit sie modale M-Codes rucksetzt.')
N1040 ('Um danach die Programmausfuhrung')
N1041 ('von Anfang an zu starten, drucken Sie ')
N1042 ('nur die Zyklusstart-Taste.           ')

N1043 (' M48 SPINDELDREHZAHL-OVERRIDE ERLAUBT')
N1044 ('M48 schaltet den Ubersteuerungsschalter')
N1045 ('fur die Spindeldrehzahl am Bedienfeld')
N1046 ('frei. Mit diesem Wahlschalter kann die')
N1047 ('Spindeldrehzahl variiert werden ohne')
N1048 ('das Teileprogramm zu verandern.')
N1049 ('                                          ')

N1050 (' M49 SPINDELDREHZ-OVERR. NICHT ERLAUBT')
N1051 ('M49 sperrt den Ubersteuerungsschalter')
N1052 ('fur die Spindeldrehzahl am Bedienfeld.')
N1053 ('In diesem Modus kann die Spindeldrehzahl')
N1054 ('nur im Teileprogramm geandert werden.')
N1055 ('           ')
N1056 ('                                  ')

N1057 (' M67 - M69 Motorstartbefehle')
N1058 ('M67 mit M69 sind M-Codes, die zum')
N1059 ('Einschalten von Geraten an der Maschine')
N1060 ('dienen. Die Gerate sollen hier einfach')
N1061 ('"Motoren" heisen. ')
N1062 ('M67 im Programm startet den Motor 1,')
N1063 ('M68 den Motor 2, M69 den Motor 3.      ')

N1064 (' M77 - M79 Motorstoppbefehle ')
N1065 ('M77 mit M79 sind M-Codes, die zum')
N1066 ('Ausschalten von Geraten an der Maschine')
N1067 ('dienen. Die Gerate sollen hier einfach')
N1068 ('"Motoren" heisen.    ')
N1069 ('M77 im Programm stoppt den Motor 1,')
N1070 ('M78 den Motor 2, M79 den Motor 3.           ')

N1071 (' M98 UNTERPROGRAMMAUFRUF ')
N1072 ('M98 P9876 RUFT DAS UNTERPROGRAMM NUMMER')
N1073 ('9876 AUF. AN DAS U-PROGRAMM KONNEN KEINE')
N1074 ('ARGUMENTE UBERGEBEN WERDEN. BEENDEN SIE')
N1075 ('DAS AUFGERUFENE UNTERPROGRAMM MIT M99.')
N1076 ('ACHTUNG: WENN DAS UNTERPROGRAMM M02 ODER')
N1077 ('M30 ENTHALT, KEHRT ES NICHT ZUR BEENDI-')
N1078 ('GUNG ZUM AUFRUFENDEN PROGRAMM ZURUCK.')

N1079 (' M99 RUCKKEHR VOM UNTERPROGRAMM')
```

```
N1080 ('M99 BEWIRKT BEI DER PROGRAMMAUSFUHRUNG')
N1081 ('DIE RUCKKEHR ZUM AUFRUFENDEN PROGRAMM.')
N1082 ('WURDE DAS PROGRAMM VON KEINEM ANDEREN')
N1083 ('PROGRAMM AUFGERUFEN, ERFOLGT EINE RUCK-')
N1084 ('KEHR ZUM PROGRAMMANFANG, UND ES ENTSTEHT')
N1085 ('EINE ENDLOSSCHLEIFE.          ')

N1168 (' M00 PROGRAMMHALT')
N1169 (' M01 OPTIONALER HALT')
N1170 (' M02 PROGRAMMENDE')
N1171 (' M03 SPINDEL IM US')
N1172 (' M04 SPINDEL GEGEN US')
N1173 (' M05 SPINDELHALT')
N1174 (' M06 WERKZEUGWECHSEL')
N1175 (' M07 SPULKUHLMITTEL')
N1176 (' M08 KUHLMITTELNEBEL')

N1177 (' M09 KUHLMITTEL AUS')
N1178 (' M19 SPINDELORIENTIERUNG')
N1179 (' M30 PROGRAMM ANHALTEN + ANFANG SUCHEN')
N1180 (' M48 SPINDELDREHZAHL-OVERRIDE ERLAUBT')
N1181 (' M49 SPINDELDREHZAHL-OVERRIDE 100 %')
N1182 (' M67 MOTOR 1 STARTEN')
N1183 (' M68 MOTOR 2 STARTEN')
N1184 (' M69 MOTOR 3 STARTEN')
N1185 (' M77 MOTOR 1 STOPPEN')

N1186 (' M78 MOTOR 2 STOPPEN')
N1187 (' M79 MOTOR 3 STOPPEN')
N1188 (' M98 UNTERPROGRAMMAUFRUF')
N1189 (' M99 RUCKKEHR VOM UNTERPROGRAMM')
```

D.2.4.4 Sample of Messages in Italy

```
N0971 (' M00 ARRESTO PROGRAMMA')
N0972 ('M00 ferma l''esecuzione del programma')
N0973 ('pezzo.    Per continuare, l''ope-')
N0974 ('ratore deve premere il pulsante')
N0975 ('cycle start.                                  ')

N0976 (' M01 ARRESTO OPZIONALE ')
N0977 ('M01 ferma l''esecuzione del programma')
N0978 ('pezzo solo se e attiva la funzione ARRE-')
N0979 ('STO OPZIONALE.    (Normalmente,    e')
N0980 ('un pulsante del pannello operatore.')
N0981 ('Se la funzione e attiva, per continuare')
N0982 ('si deve premere il pulsante')
N0983 ('cycle start.                ')

N0984 (' M02 FINE PROGRAMMA')
N0985 ('M02 Mette fine all''esecuzione del ')
N0986 ('programma pezzo.    Dopo un M02,')
N0987 ('l''operatore deve premere RESET per')
N0988 ('riavviare l''esecuzione.')

N0989 (' M03 ROT.MANDRINO IN SENSO ORARIO')
N0990 ('M03 avvia la rotazione del mandrino in')
N0991 ('senso orario.    Il mandrino')
```

```
N0992 ('ruotera alla velocita programmata.       ')
N0993 ('Questa si specifica con un codice S.    ')
N0994 (' M04 ROT.MANDRINO IN SENSO ANTIORARIO')
N0995 ('M04 avvia la rotazione del mandrino in')
N0996 ('senso antiorario.    Il')
N0997 ('mandrino ruotera alla velocita pro-')
N0998 ('grammata.    Questa si specifica con    ')
N0999 ('un codice S.                           ')
N1000 (' M05 ARRESTO MANDRINO ')
N1001 ('M05 ferma la rotazione del mandrino.')
N1002 (' M06 CAMBIO UTENSILE')
N1003 ('M06 richiede un cambio utensile sulla ')
N1004 ('macchina.    La macro di cambio utensile')
N1005 ('deve fermare il mandrino e il refrige-')
N1006 ('rante ed avviare le azioni necessarie')
N1007 ('per eseguire il cambio utensile.')
N1008 ('Alla fine del cambio utensile, il man- ')
N1009 ('drino sara inattivo.                   ')
N1010 (' M07 REFRIGERANTE A GETTO')
N1011 ('M07 attiva il getto di refrigerante    ')
N1012 ('sulla macchina.    Si tratta di un getto')
N1013 ('di refrigerante inteso ad evitare il')
N1014 ('surriscaldamento dell''utensile.')
N1015 (' M08 REFRIGERANTE A NEBBIA')
N1016 ('M08 attiva il refrigerante a nebbia')
N1017 ('sulla macchina.    Si tratta di un getto')
N1018 ('meno intenso di refrigerante usato quan-')
N1019 ('do le forze di taglio sono basse o il')
N1020 ('materiale dissipa la maggior parte del')
N1021 ('calore generato dalla lavorazione. ')
N1022 (' M09 REFRIGERANTE DISATTIVATO')
N1023 ('M09 disattiva ambedue i tipi di refrige-')
N1024 ('rante sulla macchina.    Questa funzione')
N1025 ('e utile quando l''utensile e lontano')
N1026 ('dal pezzo per l''ispezione.             ')
N1027 ('M05 e M06 dovrebbero disattivare il re-')
N1028 ('frig., ma cio non puo essere garantito. ')
N1029 (' M19 ORIENTAMENTO MANDRINO ')
N1030 ('M19 richiede l''orientamento del man- ')
N1031 ('drino per bloccarlo in una posizione')
N1032 ('nota.    Questa funzione e utile')
N1033 ('durante i cicli di cambio utensile.')
N1034 (' M30 ARRESTO PROGR. E RIAVVOLGIMENTO')
N1035 ('M30 ferma l''esecuzione del programma')
N1036 ('pezzo, esegue un reset e riporta il pro-')
N1037 ('gramma all''inizio.    Il comando')
N1038 ('viene inviato al PMC per il reset dei')
N1039 ('codici M modali.              ')
N1040 ('Il programma puo essere rieseguito dal-')
N1041 ('l''inizio premendo semplicemente il')
N1042 ('pulsante cycle start. ')
```

```
N1043 (' M48 REG. VELOC. MANDRINO ABILITATA')
N1044 ('M48 abilita il selettore della regola-')
N1045 ('zione della velocita del mandrino sul')
N1046 ('pannello operatore.    Questo selettore')
N1047 ('permette di cambiare la velocita del ')
N1048 ('mandrino senza modificare il programma')
N1049 ('pezzo.                                 ')

N1050 (' M49 REG. VELOC. MANDRINO DISABIL.')
N1051 ('M49 inibisce il selettore della regola-')
N1052 ('zione della velocita del mandrino sul')
N1053 ('pannello operatore.    In questo modo')
N1054 ('non e possibile cambiare la velocita')
N1055 ('del mandrino senza modificare il pro-')
N1056 ('gramma pezzo.                          ')

N1057 (' M67 - M69 comandi di avvio motori')
N1058 ('M67-M69 sono forniti per attiva-')
N1059 ('re i dispositivi della macchina.    Questi')
N1060 ('dispositivi possono essere detti comune-')
N1061 ('mente motori.    Il motore 1 si avvia con')
N1062 ('M67, il motore 2 con M68, e il motore 3')
N1063 ('con M69.                               ')

N1064 (' M77 - M79 Comandi di arresto motori')
N1065 ('M77-M79 sono forniti per disattivare i')
N1066 ('dispositivi della macchina.    Questi')
N1067 ('dispositivi possono essere detti comune-')
N1068 ('mente motori.    Il motore 1 si ferma con')
N1069 ('M77, il motore 2 con M78, e il motore 3')
N1070 ('con M79.                               ')

N1071 (' M98 RICHIAMO SOTTOPROGRAMMA')
N1072 ('M98 P9876 RICHIAMA IL SOTTOPROGRAMMA N.')
N1073 ('9876. NON E" POSSIBILE PASSARE ARGO-')
N1074 ('MENTI AI SOTTOPROGRAMMI, CHE DEVONO')
N1075 ('FINIRE CON UN M99.                     ')
N1076 ('ATTENZIONE: M02 O M30 USATI IN UN')
N1077 ('SOTTOPROGRAMMA, IMPEDISCONO IL RITORNO')
N1078 ('AL PROGRAMMA RICHIAMANTE.      ')

N1079 (' M99 RITORNO DAL SOTTOPROGRAMMA')
N1080 ('QUANDO VIENE INCONTRATO M99,IL CONTROLLO')
N1081 ('RITORNA AL PROGRAMMA RICHIAMANTE.')
N1082 (' SE IL PROGR. CON M99 ERA STATO AVVIA-')
N1083 ('TO INDIPENDENTEMENTE DA UN ALTRO PRO-')
N1084 ('GRAMMA, ESSO TORNA ALL"INIZIO E VIENE')
N1085 ('RIESEGUITO INDEFINITAMENTE.')

N1168 (' M00 ARRESTO PROGRAMMA')
N1169 (' M01 ARRESTO OPZIONALE ')
N1170 (' M02 FINE PROGRAMMA')
N1171 (' M03 ROT.MANDRINO IN SENSO ORARIO')
N1172 (' M04 ROT.MANDRINO IN SENSO ANTIORARIO')
N1173 (' M05 ARRESTO MANDRINO ')
N1174 (' M06 CAMBIO UTENSILE')
N1175 (' M07 REFRIGERANTE A GETTO')
N1176 (' M08 REFRIGERANTE A NEBBIA')
```

```
N1177 (' M09 REFRIGERANTE DISATTIVATO')
N1178 (' M19 ORIENTAMENTO MANDRINO')
N1179 (' M30 ARRESTO PROGR. E RIAVVOLGIMENTO')
N1180 (' M48 REG. VELOC. MANDRINO ABILITATA')
N1181 (' M49 REG. VELOC. MANDRINO DISABILITATA')
N1182 (' M67 AVVIO MOTORE 1')
N1183 (' M68 AVVIO MOTORE 2')
N1184 (' M69 AVVIO MOTORE 3')
N1185 (' M77 ARRESTO MOTORE 1')
N1186 (' M78 ARRESTO MOTORE 2')
N1187 (' M79 ARRESTO MOTORE 3')
N1188 (' M98 RICHIAMO SOTTOPROGRAMMA')
N1189 (' M99 RITORNO DAL SOTTOPROGRAMMA')
```

D.2.4.5 Sample of Messages in French

```
N0971 (' M00 ARRET DE PROGRAMME ')
N0972 ('M00 provoquera l''arret de l''execution du')
N0973 ('programme. Pour continuer, l''operateur    ')
N0974 ('devra appuyer sur la touche    ')
N0975 ('depart cycle.                              ')
N0976 (' M01 ARRET OPTIONNEL ')
N0977 ('M01 provoquera l''arret de l''execution du')
N0978 ('programme si la fonction ARRET         ')
N0979 ('OPTIONNEL est active. Il s''agit souvent ')
N0980 ('d''un bouton sur le pupitre operat.          ')
N0981 ('Si l''arret optionnel est actif, l''oper.')
N0982 ('evra appuyer sur le bouton depart cycle ')
N0983 ('pour continuer.                ')
N0984 (' M02 FIN DE PROGRAMME ')
N0985 ('M02 provoquera l''arret de l''execution du')
N0986 ('programme. Apres l''execution d''un M02,')
N0987 ('l''operateur doit appuyer sur RESET pour')
N0988 ('redemarrer l''exec.                        ')
N0989 (' M03 BROCHE EN SENS HORAIRE ')
N0990 ('M03 demande le demarrage de la broche')
N0991 ('en sens horaire. La broche tournera a    ')
N0992 ('la vitesse programmee.       ')
N0993 ('Le chang. de vit. se fait avec code S.    ')
N0994 (' M04 BROCHE EN SENS ANTI-HORAIRE ')
N0995 ('M04 demande le demarrage de la broche')
N0996 ('en sens anti-horaire. La broche            ')
N0997 ('tournera a la vitesse programmee.          ')
N0998 ('Le changement de vitesse s''effectue      ')
N0999 ('par un code S.                             ')
N1000 (' M05 ARRET DE BROCHE ')
N1001 ('M05 demande l''arret de la broche.         ')
N1002 (' M06 CHANGEMENT D''OUTIL ')
N1003 ('M06 demande un changement d''outil             ')
N1004 ('a la machine. La macro de changement    ')
N1005 ('d''outil arrete la broche et l''arrosage, ')
N1006 ('puis initie les actions necessaires        ')
```

```
N1007 ('pour le chang. d"out.                    ')
N1008 ('A la fin du changement d"outil, la       ')
N1009 ('la broche reste arretee.                 ')

N1010 (' M07 ARROSAGE A FLOT ACTIF ')
N1011 ('M07 demande l"activation de l"arros.     ')
N1012 ('a flot sur la machine. Il s"agit d"un')
N1013 ('debit important d"arrosage pour eviter   ')
      N1014 ('la surchauffe de l"outil.          ')

N1015 (' M08 ARROSAGE BROUILLARD ACTIF ')
N1016 ('M08 demande l"activation de              ')
N1017 ('l"arrosage brouillard sur la machine.')
N1018 ('Arrosage leger utilise pour couper       ')
N1019 ('a moindre effort ou si matiere dissipe   ')
N1020 ('facilement la chaleur generee par les    ')
N1021 ('cycles d"usinage.                        ')

N1022 (' M09 ARRET ARROSAGE ')
N1023 ('M09 coupe les arrosages a flot et        ')
N1024 ('brouillard de la machine. Cette fonction')
N1025 ('est utile lorsque l"outil est eloigne    ')
N1026 ('de la piece et demande une inspection.   ')
N1027 ('M05 et M06 doivent aussi couper          ')
N1028 ('l"arrosage, mais ce n"est pas garanti. ')

N1029 (' M19 DEMANDE D"ORIENTATION DE BROCHE ')
N1030 ('M19 demande l"orientation de la          ')
N1031 ('broche afin de la verrouiller dans une ')
N1032 ('verrouiller dans une position connue. ')
N1033 ('Utile pour chang. d"outil.               ')

N1034 (' M30 ARRET PROGRAMME ET REMBOBINAGE ')
N1035 ('M30 Arrete l"execution du programme      ')
N1036 ('piece actuel, reinitialise et            ')
N1037 ('rembobine pour la CNC. Cette demande')
N1038 ('passe egalement au logiciel de contr.    ')
N1039 ('mach. pour reinit. codes M mod..         ')
N1040 ('Execution du programme piece possible    ')
N1041 ('a partir du debut en appuyant simplement')
N1042 ('depart cycle.                            ')

N1043 (' M48 MODUL. VITESSE DE BROCHE PERMISE ')
N1044 ('M48 valide la fonctionnalite du          ')
N1045 ('select. de modul. de broche du           ')
N1046 ('pupitre operateur. Avec ce ce sel.,      ')
N1047 ('l"operateur peut modifier la vitesse     ')
N1048 ('de la broche sans modifier le            ')
N1049 ('prgr. piece.                             ')

N1050 (' M49 MODUL. VITESSE BROCHE INTERDITE ')
N1051 ('M49 interdit la fonction du selecteur    ')
N1052 ('de modul. de broche sur le pupitre       ')
N1053 ('operateur. Dans ce mode, l"operateur     ')
N1054 ('n"est pas autorise a modifier la vit.    ')
N1055 ('de broche sans modifier le progr.        ')
N1056 ('piece.                                   ')

N1057 (' M67 - M69 Commandes demarrage moteur ')
N1058 ('M67 a M69 sont des codes M fournis pour ')
```

```
N1059 ('activer un dispositif de la machine. Ces')
N1060 ('dispositifs sont souvent les moteurs    ')
N1061 ('Le moteur 1 est demarre en                ')
N1062 ('programmant M67, le moteur 2 avec M68    ')
N1063 ('et le moteur 3 avec M69.                  ')

N1064 (' M77 - M79 Commandes d''arret moteur ')
N1065 ('M77 a M79 sont des codes M fournis pour ')
N1066 ('activer un dispositif de la machine. Ces')
N1067 ('dispositifs sont souvent les moteurs    ')
N1068 ('Le moteur 1 est arrete en programmant      ')
N1069 ('M77, le moteur 2 avec M78 et    ')
N1070 ('le moteur 3 avec M79.                     ')

N1071 (' M98 APPEL DE SOUS-PROGRAMME ')
N1072 ('M98 P9876 APPELLERA LE NUMERO DE SOUS-    ')
N1073 ('PROGRAMME 9876. LES ARGUMENTS NE PEUVENT')
N1074 ('PASSER AU SOUS-PROGRAMME. LE SOUS-PROGR.')
N1075 ('APPELE DOIT FINIR AVEC M99.                ')
N1076 ('ATTENTION: SI M02 OU M30 SONT UTILISES    ')
N1077 ('DANS LE SOUS-PROGR., LE PROGRAMME NE ')
N1078 ('REVIENDRA PAS AU PROG D''APP ET FINIRA.    ')

N1079 (' M99 REVIENT DU SOUS-PROGRAMME ')
N1080 ('LORSQUE M99 EST RENCONTRE PENDANT L''EXEC')
N1081 ('D''UN PROG, IL REVIENDRA AU PROG D''APPEL')
N1082 ('SI LE PROG. ETAIT EXECUTE SANS APPELER    ')
N1083 ('DE PROGRAMME, LE PROGRAMME SE    ')
N1084 ('REPETERA INDEFINIMENT.      ')
N1085 ('                                    ')

N1168 (' M00 ARRET DE PROGRAMME')
N1169 (' M01 ARRET OPTIONNEL')
N1170 (' M02 FIN DE PROGRAMME')
N1171 (' M03 BROCHE EN SENS HORAIRE')
N1172 (' M04 BROCHE EN SENS ANTI-HORAIRE')
N1173 (' M05 ARRET DE BROCHE')
N1174 (' M06 CHANGEMENT D''OUTIL')
N1175 (' M07 ARROSAGE A FLOT')
N1176 (' M08 ARROSAGE BROUILLARD')

N1177 (' M09 ARRET ARROSAGE')
N1178 (' M19 ORIENTATION DE BROCHE')
N1179 (' M30 ARRET DE PROGRAMME ET REMBOBINAGE')
N1180 (' M48 MODULATION VIT. DE BROCHE PERMISE')
N1181 (' M49 MODULATION VITESSE DE BROCHE 100%')
N1182 (' M67 DEMARRAGE MOTEUR 1')
N1183 (' M68 DEMARRAGE MOTEUR 2')
N1184 (' M69 DEMARRAGE MOTEUR 3')
N1185 (' M77 ARRET MOTEUR 1')

N1186 (' M78 ARRET MOTEUR 2')
N1187 (' M79 ARRET MOTEUR 3')
N1188 (' M98 APPEL DE SOUS-PROGRAMME')
N1189 (' M99 RETOUR D''UN SOUS-PROGRAMME')
```

D.2.4.6 Sample of Messages in Spanish

```
N0971 (' M00 PARADA PROGRAMADA ')
N0972 ('M00 hara que el prog. pieza pare ')
N0973 ('la ejec.    Para continuar, se pedira      ')
N0974 ('al oper. que pulse la tecla de inicio    ')
N0975 ('de ciclo.                                  ')

N0976 (' M01 PARADA OPCIONAL ')
N0977 ('M01 hara que el prog. de pieza deje de')
N0978 ('ejecutarse solo si esta acti. la funcion')
N0979 ('PARADA OPCIONAL.    Esta se activa con')
N0980 ('un pulsador del panel de operador.          ')
N0981 ('Si la parada opc. se activa, se pedira')
N0982 ('al operador que pulse el boton de ini. ')
N0983 ('de ciclo para continuar.                ')

N0984 (' M02 FIN PROGRAMA ')
N0985 ('M02 detiene la ejec. del programa          ')
N0986 ('pieza. Despues de ejecutar M02,')
N0987 ('el operador debe pulsar RESET para')
N0988 ('reanudar la ejecuc.                                ')

N0989 (' M03 CABEZAL HORARIO ')
N0990 ('M03 pide que se arranque el cabezal en')
N0991 ('sentido horario. El cabezal girara a la ')
N0992 ('vel. programada. Para cambiar la      ')
N0993 ('vel., utilice un codigo S.    ')

N0994 (' M04 CABEZAL ANTIHORARIO ')
N0995 ('M04 pide que se arranque el cabezal')
N0996 ('en sentido antihorario. Este girara a ')
N0997 ('la velocidad programada.            ')
N0998 ('Para cambiar la velocidad, utilizar      ')
N0999 ('un codigo S.                              ')

N1000 (' M05 PARADA CABEZAL ')
N1001 ('M05 pide la parada del cabezal. ')

N1002 (' M06 CAMBIAR HTA ')
N1003 ('M06 pide un cambio de hta en la            ')
N1004 ('maquina.    La macro de cambio de hta    ')
N1005 ('debe detener el cab. y el refrigerante    ')
N1006 ('y luego iniciar las acciones necesarias')
N1007 ('para cambiar la herramienta.          ')
N1008 ('Al final del cambio de herramienta, el ')
N1009 ('cabezal permanece desconectado      ')

N1010 ('M07 CONECTAR CHORRO REFRIG. ')
N1011 ('M07 pide conectar chorro refrigerante')
N1012 ('en la maquina.    Se trata de un potente')
N1013 ('chorro de refrigerante para impedir        ')
N1014 ('que se recaliente la herramienta        ')

N1015 (' M08 CONECTAR NEBLINA REFRI. ')
N1016 ('M08 pide conectar la neblina del             ')
N1017 ('refri. en la maq. Se trata de un flujo')
N1018 ('lig. de refri. para cuando los esf.         ')
N1019 ('de corte sean pequenos o el material    ')
```

```
N1020 ('disipe facilm. una gran parte del calor')
N1021 ('generado durante el cicl de corte ')

N1022 (' M09 DESC. REFRIGERANTE ')
N1023 ('M09 descon. la ducha y neblina de refri.')
N1024 ('en la maquina. Funcion practica ')
N1025 ('cuando la hta. este lejos de la pieza ')
N1026 ('y requiera inspeccion. ')
N1027 ('M05 y M06 tambien deben desconectar ')
 N1028 ('el refri., pero no puede garantizarse. ')

N1029 (' M19 PETI. ORIENT. CABE. ')
N1030 ('M19 pide orientar el cabezal para ')
N1031 ('bloquear el eje de este en una pos. ')
N1032 ('conocida. Funcion prac. durante los ')
N1033 ('ciclos de cambio de hta. ')

N1034 (' M30 PARA. Y REINI. PROGRAMA ')
N1035 ('M30 detiene la ejecucion del programa ')
N1036 ('pieza actual y envia petic. de reset ')
N1037 ('y reini. al CNC. Esta peti tamb. se pasa')
N1038 ('al soft. de control de la maqui. para ')
N1039 ('resetear los codigos M modales. ')
N1040 ('Es posible ejecutar el progr. de pieza ')
N1041 ('desde el comienzo pulsando Inicio ')
N1042 ('de ciclo. ')

N1043 (' M48 PERMITIDA CORR. VEL. CABE. ')
N1044 ('M48 habilita la funciona. del selec. ')
N1045 ('de corr. de cabe. en el panel de ')
N1046 ('operador. Con este selector, el ')
N1047 ('operador puede modificar la velo. ')
N1048 ('del cabe. sin modificar el prog. ')
N1049 ('de pieza. ')

N1050 (' M49 NO PERMITIDA CORR. VEL. CABE. ')
N1051 ('M49 inhibe la func. del selector de ')
N1052 ('corr. de cabezal en el panel de. ')
N1053 ('operador. En este modo, no se ')
N1054 ('permite al ope. modificar la vel. ')
N1055 ('del cabe. sin modificar el progr. ')
N1056 ('de pieza. ')

N1057 (' M67 - M69 Ordenes arranque motor ')
N1058 ('M67 hasta M69 son cod. M para conectar ')
N1059 ('un dispo. de la maquina. Estos dispo. ')
N1060 ('pueden denominarse en comun motores. ')
N1061 ('El motor 1 se arranca programando ')
N1062 ('M67, el motor 2 con M68 y el motor ')
N1063 ('3 con M69. ')

N1064 (' M77 - M79 Ordenes parada motor ')
N1065 ('M77 hasta M79 son cod. M para descon. ')
N1066 ('un dispo. de la maquina. Estos dispo.')
N1067 ('pueden denominase en comun motores. ')
N1068 ('El motor 1 se para programando ')
N1069 ('M77, el motor 2 con M78 y el motor ')
N1070 ('3 con M79. ')

N1071 (' M98 LLAMADA A SUBPROGRAMA ')
```

N1072 ('M98 P9876 LLAMARA AL NO. DE SUBPROGR. ')
N1073 ('9876. LOS ARGUM. NO PUEDEN PASARSE AL')
N1074 ('SUBPR. EL SUBPROGRMA LLAMADO DEBE')
N1075 ('TERMINAR CON M99. ')
N1076 ('PRECAU: SI SE UTILIZ. M02 O M30 EN EL ')
N1077 ('SUBPR., EL PROGRAMA NO VOLVERA ')
N1078 ('AL PROG. DE LLAMADA Y SE TERMIN. ')

N1079 (' M99 RETORNO DE SUBPROGRAMA ')
N1080 ('CUANDO SE ENCUENTRE M99 DURANTE EJE. ')
N1081 ('UN PROG, VOLVERA AL PROG DE LLAMADA.')
N1082 ('SI SE HA EJECUTADO EL PROG. SIN UN ')
N1083 ('PROG. DE LLAMADA, EL PROG. VOLVERA EN')
N1084 ('BUCLE AL COMIENZO Y SE REPETIRA ')
N1085 ('INDEFINIDAMENTE ')

N1168 (' M00 PARADA PROGRAMA')
N1169 (' M01 PARADA OPCIONAL')
N1170 (' M02 FIN PROGRAMA')
N1171 (' M03 CABEZAL HOR')
N1172 (' M04 CABEZAL ANTIH')
N1173 (' M05 PARADA CABEZAL')
N1174 (' M06 CAMBIAR HTA')
N1175 (' M07 CHOR REFRIG.')
N1176 (' M08 NEBLI. REFRIG.')

N1177 (' M09 DES. REFRIG.')
N1178 (' M19 ORIENT. CABEZAL')
N1179 (' M30 PARADA Y REINIC PROGRAMA')
N1180 (' M48 PERMITIDO CORRECCION VELO. CABE.')
N1181 (' M49 CORRECCION VEL. CABE. 100%')
N1182 (' M67 ARRANQUE MOTOR 1')
N1183 (' M68 ARRANQUE MOTOR 2')
N1184 (' M69 ARRANQUE MOTOR 3')
N1185 (' M77 PARADA MOTOR 1')

N1186 (' M78 PARADA MOTOR 2')
N1187 (' M79 PARADA MOTOR 3')
N1188 (' M98 LLAMADA A SUBPROGRAMA')
N1189 (' M99 RETORNO DE SUBPROGRAMA')

D.2.4.7 Sample of Messages in Chinese (Traditional character)

N0971 (' M00 程式停止')
N0972 ('M00 可使加工程式停止執行.要継続運転的話')
N0973 (',押 cycle start 鍵 ')
N0974 (' ')
N0975 (' ')

N0976 (' M01 選択性停止')
N0977 ('M01 可使加工程式停止執行,如果使用「選択 '」
N0978 ('停止)機能.通常操作方式為押操作面盤上的 ')
N0979 ('鍵.選択停止有効時,要執行運転押 CYCLE STA ')
N0980 ('RT 鍵 ')
N0981 (' ')
N0982 (' ')
N0983 (' ')

N0984 (' M02 程式終止')
N0985 ('M02 可終止程式執行.当 M02 執行後,操作者需 ')
N0986 ('押 RESET 才可再執行程式.')
N0987 (' ')
N0988 (' ')

N0989 (' M03 主軸正転')
N0990 ('M03 可使主軸順時針方向旋転,並依程式所指 ')
N0991 ('定主軸旋転速度旋転,下 S-C0DE 可変更主軸速 ')
N0992 ('度 ')
N0993 (' ')

N0994 (' M04 主軸反転')
N0995 ('M04 可使主軸逆時針方向旋転,並依程式所指 ')
N0996 ('定主軸旋転速度旋転,下 S-C0DE 可変更主軸速 ')
N0997 ('度 ')
N0998 (' ')
N0999 (' ')

N1000 (' M05 主軸停止')
N1001 ('M05 可使主軸停止')

N1002 ('M06 刀具交換')
N1003 ('M06 可使刀具交換.刀具交換 macro 応使切削液')
N1004 ('及主軸停止,然後執行需要的初期化以使刀具 ')
N1005 ('作交換. ')
N1006 (' ')
N1007 (' ')
N1008 (' ')
N1009 (' ')

N1010 (' M07 切削液開')
N1011 ('M07 可使切削液打開,強大的切削液水流可使')
N1012 ('刀具不会過熱 ')
N1013 (' ')
N1014 (' ')

N1015 ('M08 霧化式切削液開 ')
N1016 ('M08 可使霧化式切削液打開.小水流可用在軽')
N1017 ('切削場合 or 用在切削中容易散熱的材質 ')
N1018 (' ')
N1019 (' ')
N1020 (' ')
N1021 (' ')

N1022 (' M09 切削液関')
N1023 ('M09 可関閉切削液及霧化式切削液.此機能可 ')
N1024 ('在刀具離開工件時使用 ')
N1025 (' ')
N1026 (' ')
N1027 (' ')
N1028 (' ')

N1029 ('M19 主軸定位')
N1030 ('M19 為主軸定位,此機能用在刀具交換 ')
N1031 (' ')

N1032 ('　　')
N1033 ('　　')

N1034 ('M30　程式停止回到程式起点　　')
N1035 ('M30　可使程式停止並重置 CNC 再回到程式起点　')
N1036 ('　　')
N1037 ('　　')
N1038 ('　　')
N1039 ('　　')
N1040 ('　　')
N1041 ('　　')
N1042 ('　　')

N1043 (' M48　主軸転速倍率有効')
N1044 ('M48　可使操作面盤上的主軸転速倍率有効,操')
N1045 ('作者可直接控制主軸転速不用修改程式　　')
N1046 ('　　')
N1047 ('　　')
N1048 ('　　')
N1049 ('　　')

N1050 (' M49　主軸転速倍率無効　　')
N1051 ('M49　可使操作面盤上的主軸転速倍率無効,操作')
N1052 ('者不修改程式的話就無法改変主軸転速　　')
N1053 ('　　')
N1054 ('　　')
N1055 ('　　')
N1056 ('　　')

N1057 (' M67 - M69　馬達起動指令')
N1058 ('M67　到　M69　指令可起動馬達,M67 起動馬達 1,M')
N1059 ('68 起動馬達 2,M69 起動馬達 3　　')
N1060 ('　　')
N1061 ('　　')
N1062 ('　　')
N1063 ('　　')

N1064 (' M77 - M79　馬達停止指令')
N1065 ('M77 - M79　指令可使伺服馬達停止.M77 停止馬')
N1066 ('達 1,M78 停止馬達 2,M79 停止馬達 3　　')
N1067 ('　　')
N1068 ('　　')
N1069 ('　　')
N1070 ('　　')

N1071 (' M98　呼叫副程式')
N1072 ('M98 P9876　呼叫程式番号 9876 程式,但引数無')
N1073 ('法伝到副程式,用 M99 終止副程式　　')
N1074 ('　　')
N1075 ('　　')
N1076 ('要注意若是在副程式使用 M02、M30 場合.会無')
N1077 ('法回到原来程式,程式会自動終了.　　')
N1078 ('　　')

N1079 ('M99　由副程式退回')
N1080 ('程式執行中読到　M99　会返回原呼叫程式,若程')

```
N1081 ('式執行没有呼叫副程式的話,会一直重覆執行  ')
N1082 ('此程式.                                    ')
N1083 ('                                           ')
N1084 ('                                           ')
N1085 ('                                           ')

N1168 (' M00  程式停止')
N1169 (' M01  選択停止')
N1170 (' M02  程式終了')
N1171 (' M03  主軸正転')
N1172 (' M04  主軸反転')
N1173 (' M05  主軸停止')
N1174 (' M06  刀具交換')
N1175 (' M07  切削液開')
N1176 (' M08  霧化式切削液開')

N1177 (' M09  切削液関')
N1178 (' M19  主軸定位')
N1179 (' M30  程式終了回到起点')
N1180 (' M48  主軸速度倍率調整有効')
N1181 (' M49  主軸速度倍率調整 100%')
N1182 (' M67  起動馬達  1')
N1183 (' M68  起動馬達  2')
N1184 (' M69  起動馬達  3')
N1185 (' M77  停止馬達  1')

N1186 (' M78  停止馬達  2')
N1187 (' M79  停止馬達  3')
N1188 (' M98  呼叫副程式')
N1189 (' M99  従副程式返回')
```

D.2.4.8 Sample of Messages in Portuguese

```
N0971 (' M00 PARADA PROGRAMA ')
N0972 ('M00 fara com que o programa da peca ')
N0973 ('pare a execucao.    Para continuar, o      ')
N0974 ('operador tera de premir a tecla inicio ')
N0975 ('de ciclo.                                  ')
N0976 (' M01 PARADA OPCIONAL ')
N0977 ('M01 faz com que o programa PECA pare ')
N0978 ('exec. so se funcao PARADA OPCIONAL         ')
N0979 ('estiver ATIVA.    Trata-se normalmente    ')
N0980 ('de botao pressao no painel oper.           ')
N0981 ('Se a PARADA opcional for ativada, o')
N0982 ('operador tera de premir o botao inicio     ')
N0983 ('ciclo para continuar.                ')
N0984 (' M02 FIM DO PROGRAMA ')
N0985 ('M02 fara com que programa da PECA          ')
N0986 ('termine a execucao.    Apos a execucao de')
N0987 ('um M02, o operador tem premir RESET para')
N0988 ('reiniciar execucao.                        ')
N0989 (' M03 BUCHA S.HOR. ')
N0990 ('M03 solicita o INICIO da bucha no')
```

N0991 ('sentido horario. A bucha rodara ')
N0992 ('A veloc. programada da bucha. ')
N0993 ('A mudanca vel. e feita com codigo S. ')
N0994 (' M04 BUCHA S.ANTI-HOR. ')
N0995 ('M04 solicita o inicio da bucha no')
N0996 ('sentido anti-horario. A ')
N0997 ('bucha rodara a veloc. programada ')
N0998 ('da bucha. A mudanca de veloc. e ')
N0999 ('feita com codigo S. ')
N1000 (' M05 PARADA DA BUCHA ')
N1001 ('M05 solicita parada da bucha. ')
N1002 (' M06 TROCA DE FERR. ')
N1003 ('M06 solicita uma troca ferr. na ')
N1004 ('maquina. A macro troca de ferr. deve ')
N1005 ('parar bucha e fluido arref. e ')
N1006 ('iniciar acoes necessarias para ')
N1007 ('realizar a troca ferr. ')
N1008 ('No fim da troca de FERR., a ')
N1009 ('bucha permanece desligada. ')
N1010 (' M07 FLUIDO ARREF. LIQUIDO LIG. ')
N1011 ('M07 solicita ligacao fluido arref. ')
N1012 ('liq. na maquina. E um fluxo denso')
N1013 ('de fl. arref. para evitar que ')
N1014 ('a ferr. corte sobreaqueca. ')
N1015 (' M08 FLUIDO ARREF. NEBLINA LIG. ')
N1016 ('M08 solicita lig. fluido arref. ')
N1017 ('neblina na maquina. E um fluxo')
N1018 ('leve de fl. arr. p/usar c/baixas ')
N1019 ('forcas corte ou qdo material dissipa ')
N1020 ('facilmente grande parte calor gerado ')
N1021 ('dte ciclos de corte. ')
N1022 (' M09 FLUIDO ARREF. DESL. ')
N1023 ('M09 desliga o fl. arrefecimento ')
N1024 (' liq. e neblina na maq. Esta funcao ')
N1025 ('e util quando ferr. esta afastada da ')
N1026 ('peca e requer inspecao. ')
N1027 ('M05 E M06 tambem deviam desligar fl. ')
N1028 ('arref., mas tal nao pode ser garantido. ')
N1029 (' M19 PEDIDO DE ORIENT. DA BUCHA ')
N1030 ('M19 solicita a orientacao da bucha ')
N1031 ('de modo a bloquear o veio da bucha numa ')
N1032 ('posicao conhecida. Esta funcao E til ')
N1033 ('dte ciclos troca de ferr. ')
N1034 (' M30 PARADA PROGRAMA E REBOBINAMENTO ')
N1035 ('M30 para a execucao do prog.pca ')
N1036 ('atual e emite um pedido de reset e ')
N1037 ('rebobinamento para o CNC. Este pedido')
N1038 ('e tambem passado ao sw ctrl maq.p. ')
N1039 ('fazer reset cod. M modais. ')
N1040 ('E possivel execucao programa da peca ')
N1041 ('do inicio premindo simplesmente')
N1042 ('ini. ciclo. ')
N1043 (' M48 SUPLANT. VEL. BUCHA PERMITIDA ')
N1044 ('M48 ATIVA a funcao do ')
N1045 ('selector de supl. bucha no ')

```
N1046 ('painel operacao.    Com este selector,       ')
N1047 ('operador pode modificar vel. bucha         ')
N1048 ('sem mudar o programa               ')
N1049 ('da peca.                                    ')
N1050 (' M49 SUPLAN. VEL. BUCHA NAO PERMITIDA ')
N1051 ('M49 inibe a funcao do      ')
N1052 ('seletor de supl. bucha no             ')
N1053 ('painel de operacao.    Neste modo, o          ')
N1054 ('operador nao pode modificar         ')
N1055 ('a VELOC. da bucha sem mudar          ')
N1056 ('o programa peca.                             ')
N1057 (' M67 - M69 Comandos inicio do motor ')
N1058 ('M67 a M69 sao codigos M fornecidos para ')
N1059 ('ligar um aparelho na maquina.    Estes ')
N1060 ('aparelhos podem ser designados como    ')
N1061 ('motores.    Motor 1 E inic. pela                ')
N1062 ('programacao de M67, motor 2 com M68, e    ')
N1063 ('motor 3 com M69.                             ')
N1064 (' M77 - M79 Comandos PARADA do motor ')
N1065 ('M77 a M79 sao codigos M fornecidos para ')
N1066 ('desligar um aparelho na maquina.    Estes')
N1067 ('aparelhos podem ser designados como    ')
N1068 ('motores.    Motor 1 e parado por                ')
N1069 ('programacao de M77, motor 2 com M78, e    ')
N1070 ('motor 3 com M79.                              ')
N1071 (' M98 CHAMADA DE SUB-PROGRAMA ')
N1072 ('M98 P9876 CHAMA SUB-PROGRAMA NUM.      ')
N1073 ('9876. ARG.NAO PODEM SER PASSADOS PARA O ')
N1074 ('SUB-PROGRAMA. SUB-PROGRAMA CHAMADO DEVE')
N1075 ('TERMINAR COM UM M99.                          ')
N1076 ('ATENCAO: SE FOREM USADOS M02 OU M30 NO    ')
N1077 ('SUB-PROGRAMA, O PROGRAMA NAO REGRESSARA ')
N1078 ('AO PROGRAMA DE CHAMADA E TERMINA.      ')
N1079 (' REGRESSO M99 DO SUB-PROGRAMA ')
N1080 ('QDO M99 ENCONTRADO DTE EXEC. PROGRAMA    ')
N1081 ('REGRESSARA AO PROGRAMA')
N1082 ('DE CHAMADA. SE PRO. TIVER SIDO EXEC.       ')
N1083 ('SEM UM PROGRAMA DE CHAMADA, O PROGRAMA    ')
N1084 ('PERFAZ CICLO E REGR. INICIO, SENDO      ')
N1085 (' REP. INFINI.                                ')
N1168 (' M00 PARADA PROGRAMA')
N1169 (' M01 PARADA OPCIONAL')
N1170 (' M02 FIM DO PROGRAMA')
N1171 (' M03 BUCHA S.HOR.')
N1172 (' M04 BUCHA S.ANTI-HOR.')
N1173 (' M05 PARADA DA BUCHA')
N1174 (' M06 TROCA DE FERR.')
N1175 (' M07 FLUIDO ARREF. LIQUIDO')
N1176 (' M08 FLUIDO ARREF. NEBLINA')
N1177 (' M09 FLUIDO ARREF. DESL.')
N1178 (' M19 ORIENT. BUCHA')
N1179 (' M30 PARADA PROGRAMA E REBOBINAMENTO')
N1180 (' M48 SUPLANT. VEL. BUCHA PERMITIDA')
N1181 (' M49 SUPLANT. VEL. BUCHA 100 POR CENTO')
N1182 (' M67 INICIO MOTOR 1')
```

```
N1183 (' M68 INICIO MOTOR 2')
N1184 (' M69 INICIO MOTOR 3')
N1185 (' M77 PARAG. MOTOR 1')
N1186 (' M78 PARAG. MOTOR 2')
N1187 (' M79 PARAG. MOTOR 3')
N1188 (' M98 CHAMADA DE SUB-PROGRAMA')
N1189 (' M99 REGRESSO DO SUB-PROGRAMA')
```

D.2.4.9 Sample of Messages in Polish

```
N0971 (' M00 STOP PROGRAMU ')
N0972 ('Funkcja M00 powoduje zatrzymanie              ')
N0973 ('wykonywania programu. W celu wznowienia ')
N0974 ('wykonywania, operator musi wcisnac          ')
N0975 ('odpowiedni przycisk na panelu.              ')
N0976 (' M01 WARUNKOWY STOP PROGRAMU ')
N0977 ('Funkcja M01 powoduje zatrzymanie            ')
N0978 ('wykonywania programu jezeli wcisniety     ')
N0979 ('jest przycisk ZATRZYMANIA WARUNKOWEGO.    ')
N0980 ('Zwykle jest to przycisk na panelu           ')
N0981 ('operatora. W celu wznowienia                ')
N0982 ('wykonywania, operator musi wcisnac          ')
N0983 ('odpowiedni przycisk na panelu.              ')
N0984 (' M02 KONIEC PROGRAMU ')
N0985 ('Funkcja M02 powoduje zatrzymanie            ')
N0986 ('wykonywania programu. W celu wznowienia ')
N0987 ('wykonywania po zatrzymaniu funkcja M02, ')
N0988 ('operator musi wcisnac przycisk RESET.     ')
N0989 (' M03 OBROTY WRZECIONA ZRWZ ')
N0990 ('Funkcja M03 wlacza obroty wrzeciona         ')
N0991 ('zgodne z ruchem wskazowek zegara (ZRWZ).')
N0992 ('Wrzeciono pracuje z zaprogramowana za     ')
N0993 ('pomoca adresu S predkoscia.                 ')
N0994 (' M04 OBROTY WRZECIONA PRWZ ')
N0995 ('Funkcja M04 wlacza obroty wrzeciona         ')
N0996 ('przeciwne do ruchu wskazowek zegara         ')
N0997 ('(PRWZ) Wrzeciono pracuje z zaprogramo-    ')
N0998 ('wana za pomoca adresu S predkoscia.         ')
N0999 ('                                            ')
N1000 (' M05 ZATRZYMANIE WRZECIONA ')
N1001 ('Funkcja M05 zatrzymuje wrzeciono.         ')
N1002 (' M06 ZMIANA NARZEDZIA ')
N1003 ('Funkcja M06 sluzy do zmiany narzedzia     ')
N1004 ('obrabiarki. Makro do zmiany narzedzia     ')
N1005 ('powinno zatrzymac wrzeciono i chlodziwo,')
N1006 ('a nastepnie zainicjowac dzialania           ')
N1007 ('zwiazane ze zmiana narzedzia. Po            ')
N1008 ('zakonczeniu zmiany narzedzia, wrzeciono ')
N1009 ('pozostaje nie wlaczone.                     ')
N1010 (' M07 WLACZENIE CHLODZIWA INTENSYWNEGO ')
N1011 ('Funkcja M07 wlacza chlodziwo. Jest          ')
N1012 ('to chlodziwo o duzej intensywnosci,         ')
N1013 ('przeznaczone do ochrony narzedzia przed ')
N1014 ('przegrzaniem.                               ')
N1015 (' M08 WLACZENIE CHLODZIWA NORMALNEGO ')
```

```
N1016 ('Funkcja M08 wlacza chlodziwo. Jest          ')
N1017 ('to chlodziwo o malej intensywnosci,         ')
N1018 ('wykorzystywane przy malych silach           ')
N1019 ('skrawania lub gdy material latwo            ')
      N1020 ('rozprasza cieplo generowane w czasie        ')
N1021 ('obrobki.                                    ')
N1022 (' M09 WYLACZENIE CHLODZIWA ')
N1023 ('Funkcja M09 wylacza chlodziwo (zarowno    ')
N1024 ('normalne jak i o duzej intensywnosci).    ')
N1025 ('Funkcja ta jest uzyteczna, jezeli narze-')
N1026 ('dzie znajduje sie z dala od detalu i        ')
N1027 ('wymaga kontroli. Chlodziwo jest rowniez ')
N1028 ('wylacz. przez M05 i M06, ale nie zawsze.')
N1029 (' M19 ZORIENTOWANE ZATRZYM. WRZECIONA ')
N1030 ('Funkcja M19 zatrzymuje wrzeciono w          ')
N1031 ('okreslonej pozycji. Funkcja ta jest         ')
N1032 ('wykorzystywana przez cykle wymiany          ')
N1033 ('narzedzi.                                   ')
N1034 (' M30 KONIEC PROGRAMU I PRZEWINIECIE ')
N1035 ('Funkcja M30 zatrzymuje wykonywanie          ')
N1036 ('programu oraz wysyla polecenie zreseto- ')
N1037 ('wania i przewiniecia do CNC. Polecenie    ')
N1038 ('to jest rowniez przesylane do oprogramo-')
N1039 ('wania sterujacego obrabiarka w celu         ')
N1040 ('odwolania modalnych funkcji M. W celu     ')
N1041 ('uruchomienia programu od poczatku,          ')
N1042 ('wcisnac odpowiedni przycisk na panelu.    ')
N1043 (' M48 ZEZWOLENIE NA KOREKTE PRED. WRZ. ')
N1044 ('Funkcja M48 pozwala na korzystanie z        ')
N1045 ('przelacznika do korekty predkosci           ')
N1046 ('wrzeciona. Za pomoca tego przelacznika    ')
N1047 ('operator moze zmienic predkosc              ')
N1048 ('wrzeciona, bez wprowadzania zmian           ')
N1049 ('w samym programie.                          ')
N1050 (' M49 BEZ KOREKTY PREDKOSCI WRZECIONA    ')
N1051 ('Funkcja M49 nie pozwala na korzystanie z')
N1052 ('przelacznika do korekty predkosci           ')
N1053 ('wrzeciona. Za pomoca tego przelacznika    ')
N1054 ('operator nie moze zmienic predkosci         ')
N1055 ('wrzeciona, bez wprowadzania zmian           ')
N1056 ('w samym programie.                          ')
N1057 (' M67-M69 Funkcje uruchamiania silnikow')
N1058 ('Funkcje M67-M69 sluza do uruchamiania     ')
N1059 ('urzadzen obrabiarki. Urzadzenia te sa     ')
N1060 ('powszechnie nazywane silnikami. Silnik 1')
N1061 ('jest uruchamiany funkcja M67, silnik 2    ')
N1062 ('funkcja M68 a silnik 3 za pomoca            ')
N1063 ('funkcji M69.                                ')
N1064 (' M77-M79 Funkcje zatrzymyw. silnikow ')
N1065 ('Funkcje M77-M79 sluza do zatrzymywania    ')
N1066 ('urzadzen obrabiarki. Urzadzenia te sa     ')
N1067 ('powszechnie nazywane silnikami. Silnik 1')
N1068 ('jest zatrzymywany funkcja M77, silnik 2 ')
N1069 ('funkcja M78 a silnik 3 za pomoca            ')
N1070 ('funkcji M79.                                ')
```

```
N1071 (' M98 WYWOLANIE PODPROGRAMU ')
N1072 ('M98 P9876 WYWOLUJE PODPROGRAM NR 9876.    ')
N1073 ('NIE MOZNA PRZESYLAC PARAMETROW DO POD-    ')
N1074 ('PROGRAMU. PODPROGRAM POWINIEN BYC         ')
N1075 ('ZAKONCZONY M99. UWAGA: JEZELI WYWOLYWANY')
N1076 ('PODPROGRAM ZAWIERA FUNKCJE M02 LUB M30, ')
N1077 ('NIE NASTAPI ZAKONCZENIE I POWROT DO       ')
N1078 ('PROGRAMU WYWOLUJACEGO.                    ')
N1079 (' M99 POWROT Z PODPROGRAMU ')
N1080 ('PO NAPOTKANIU FUNKCJI M99 NASTEPUJE       ')
N1081 ('POWROT Z PODPROGRAMU DO PROGRAMU          ')
N1082 ('WYWOLUJACEGO. JEZELI PROGRAM BYL          ')
N1083 ('WYKONYWANY BEZ PROGRAMU WYWOLUJACEGO,     ')
N1084 ('NASTAPI PRZEJSCIE DO POCZATKU I           ')
N1085 ('POWTORZENIE WYKONYWANIA.                  ')
N1168 (' M00 STOP PROGRAMU')
N1169 (' M01 WARUNKOWY STOP PROGRAMU')
N1170 (' M02 KONIEC PROGRAMU')
N1171 (' M03 OBROTY WRZECIONA ZRWZ')
N1172 (' M04 OBROTY WRZECIONA PRWZ')
N1173 (' M05 ZATRZYMANIE WRZECIONA')
N1174 (' M06 ZMIANA NARZEDZIA')
N1175 (' M07 WLACZENIE CHLODZIWA INTENSYWNEGO')
N1176 (' M08 WLACZENIE CHLODZIWA NORMALNEGO')
N1177 (' M09 WYLACZENIE CHLODZIWA')
N1178 (' M19 ZORIENTOWANE ZATRZYM. WRZECIONA')
N1179 (' M30 KONIEC PROGRAMU I PRZEWINIECIE')
N1180 (' M48 ZEZWOLENIE NA KOREKTE PRED. WRZ.')
N1181 (' M49 BEZ KOREKTY PREDKOSCI WRZECIONA')
N1182 (' M67 URUCHOMIENIE SILNIKA 1')
N1183 (' M68 URUCHOMIENIE SILNIKA 2')
N1184 (' M69 URUCHOMIENIE SILNIKA 3')
N1185 (' M77 ZATRZYMANIE SILNIKA 1')
N1186 (' M78 ZATRZYMANIE SILNIKA 2')
N1187 (' M79 ZATRZYMANIE SILNIKA 3')
N1188 (' M98 WYWOLANIE PODPROGRAMU')
N1189 (' M99 POWROT Z PODPROGRAMU')
```

D.2.4.10 Sample of Messages in Swedish

```
N0971 (' M00 PROGRAMSTOPP ')
N0972 ('M00 far delprogrammet att stoppa sitt     ')
N0973 ('utforandet. For att fortsatta,            ')
N0974 ('ombeds operatoren att trycka ner          ')
N0975 ('cykelstarttangenten.                      ')
N0976 (' M01 MANUELLT STOPP ')
N0977 ('M01 far delprogrammet att stoppa sitt     ')
N0978 ('utforande END.OM Manuellt stopp-          ')
N0979 ('funktionen ar aktiv. Den utgors           ')
N0980 ('ofta av en tryckknapp pa panelen.         ')
N0981 ('Om det manuella stoppet ar aktivt         ')
N0982 ('ombeds operatoren att trycka ner          ')
N0983 ('cykelstartknappen for att fortsatta.      ')
N0984 (' M02 PROGRAMSLUT')
N0985 ('M02 far delprogrammet att avsluta         ')
```

```
N0986 ('utforandet. Efter utforandet av ett M02,')
N0987 ('maste operatoren trycka ner RESET ')
N0988 ('for att aterstarta utforandet. ')
N0989 (' M03 SPINDEL-CW ')
N0990 ('M03 requests starting spindeln in the ')
N0991 ('clockwise direction. The spindeln will ')
N0992 ('run at the programmed spindelspeed. ')
N0993 ('Changing speed ar done with an S-code. ')
N0994 (' M04 SPINDEL CCW ')
N0995 ('M04 begar start av spindeln i ')
N0996 ('motsols riktning. Spindeln kors ')
N0997 ('vid programmerad spindelhastighet ')
N0998 ('Andring av hastigheten gors m.en S-kod. ')
N0999 ('gjort med en S-kod. ')
N1000 (' M05 SPINDELSTOPP ')
N1001 ('M05 begar stoppning av spindeln. ')
N1002 (' M06 VERKTYGSBYTE ')
N1003 ('M06 begar verktygsbyte pa ')
N1004 ('maskinen. Verktygsbytesmakrot bor ')
N1005 ('stoppa spindeln & kylmedlet & sedan ')
N1006 ('starta nodvandiga atgarder for att ')
N1007 ('genomfora ett verktygsbyte. ')
N1008 ('Efter verktygsbytet, ')
N1009 ('forblir spindeln avstangd. ')
N1010 (' M07 SPOLKYLMEDEL PA ')
N1011 ('M07 begar paslag av spol- ')
N1012 ('kylmedlet i maskinen. Detta ar en rejal ')
N1013 ('strale kylmedel, som forhindrar att ')
N1014 ('skarverktyget overhettas. ')
N1015 (' M08 DIMKYLMEDEL PA ')
N1016 ('M08 begar paslag av dim- ')
N1017 ('kylmedlet i maskinen. Detta ar en smal ')
N1018 ('strale kylmedel, som anvands nar skar- ')
N1019 ('krafterna ar sma/nar materialet latt ')
N1020 ('avger det mesta av varmen, som uppstar ')
N1021 ('under skarning. ')
N1022 (' M09 KYLMEDEL AV ')
N1023 ('M09 slar av bade spol- & dim- ')
N1024 ('kylmedlet i maskinen. Denna funktion ')
N1025 ('ar nyttig nar verktyget ar borta fran ')
N1026 ('arbetsstycket & behover inspekteras. ')
N1027 ('M05 & M06 bor aven sla av ')
N1028 ('kylmedlet, men detta kan ej garanteras. ')
N1029 (' M19 BEGARAN OM SPINDELORIENTERING')
N1030 ('M19 begar orientering av av spindeln ')
N1031 ('for att lasa spindelaxeln i en ')
N1032 ('kand position. Denna funktion ar nyttig ')
N1033 ('under verktygsbyte-cykler . ')
N1034 (' M30 PROGRAMSTOPP & ATERSPOLN. ')
N1035 ('M30 stoppar utforande av aktuellt ')
N1036 ('delprog. & sander begaran om nollstalln.')
N1037 ('och aterspoln. till CNC:N. Denna begaran')
N1038 ('lamnas ocksa till maskinstyrningen ')
N1039 ('for att nollstalla modala M-koder. ')
N1040 ('Utforande av delprogrammet fran ')
```

```
N1041 ('borjan kan enkelt goras genom att trycka')
N1042 ('Cykel start.                                    ')
N1043 (' M48 SPDL-HASTIGHETSINGREPP TILLATEN ')
N1044 ('M48 aktiverar funktionen hos                    ')
N1045 ('spindelingreppsvaljaren pa                      ')
N1046 ('manoverpanelen. Med denna valjare,              ')
N1047 ('kan operatoren andra spindel-                   ')
N1048 ('hastighet utan att andra del-                   ')
N1049 ('programmet.                                     ')
N1050 (' M49 SPINDELHASTIGHETEN LAST ')
N1051 ('M49 stoppar funktionen hos                      ')
N1052 ('spindelingreppsvaljaren pa                      ')
N1053 ('manoverpanelen. I denna mod far                 ')
N1054 ('operatoren inte andra                           ')
N1055 ('spindelhastigheten utan andring av              ')
N1056 ('delprogrammet.                                  ')
N1057 (' M67-M69 Motorstartkommandon ')
N1058 ('M67-M69 ar M-koder for att                      ')
N1059 ('sla pa en anordning pa maskinen. Dessa          ')
N1060 ('anordningar kan i allmanhet kallas              ')
N1061 ('motorer. Motor 1 startas med                    ')
N1062 ('en programmering av M67, motor 2 med            ')
N1063 ('M68, & motor 3 med M69.                         ')
N1064 (' M77-M79 Motor-stoppkommandon ')
N1065 ('M77-M79 ar M-koder for att                      ')
N1066 ('sla av en anordning pa maskinen. Dessa          ')
N1067 ('anordningar kan i allmanhet kallas              ')
N1068 ('motorer. Motor 1 stoppas med                    ')
N1069 ('en programmering av M77, motor 2 med            ')
N1070 ('M78 & motor 3 med M79.                          ')
N1071 (' M98 UNDERPROGRAMANROP ')
N1072 ('M98 P9876 ANROPA UNDERPROGRAMNUMMER             ')
N1073 ('9876. ARGUMENT KAN EJ BE PASSERADE TILL ')
N1074 ('UNDERPROGRAM. ANROPAT. BOR                      ')
N1075 ('SLUTA MED EN M99.                               ')
N1076 ('VARNING: OM M02 ELLER M30 ANVANDS I             ')
N1077 ('UNDERPROGRAM. DET ATERGAR EJ                    ')
N1078 ('TILL ANROPSPROGRAM & AVSLUTAS.                  ')
N1079 (' M99 RETUR FRAN UNDERPROGRAM ')
N1080 ('NAR M99 PATRAFFAS UNDER PROGRAM                 ')
N1081 ('UTFORANDE, ATERGAR DEN TILL ANROPS-             ')
N1082 ('PROGRAMMET. OM PROGRAMMET UTFORDES              ')
N1083 ('UTAN ANROPSPROGRAM, LOOPAR PROGRAMMET           ')
N1084 ('TILL BORJAN & REPETERAR                         ')
N1085 ('I OANDLIGHET.                                   ')
N1168 (' M00 PROGRAMSTOPP')
N1169 (' M01 MANUELLT STOPPP')
N1170 (' M02 PROGRAMSLUT')
N1171 (' M03 SPINDEL-CW')
N1172 (' M04 SPINDEL-CCW')
N1173 (' M05 SPINDELSTOPP')
N1174 (' M06 VERKT.BYTE')
N1175 (' M07 SPOLKYLMEDEL')
N1176 (' M08 DIMKYLMEDEL')
N1177 (' M09 KYLMEDEL AV')
```

```
N1178 (' M19 SPINDELORIENT')
N1179 (' M30 PROGRAMSTOPP & REWIND')
N1180 (' M48 SPDL-HAST.INGREPP TILLATET')
N1181 (' M49 SPDL-HAST.INGREPP 100 %')
N1182 (' M67 START MOTOR 1')
N1183 (' M68 START MOTOR 2')
N1184 (' M69 START MOTOR 3')
N1185 (' M77 STOPP MOTOR 1')
N1186 (' M78 STOPP MOTOR 2')
N1187 (' M79 STOPP MOTOR 3')
N1188 (' M98 U-PROGRAMANROP')
N1189 (' M99 RETUR FRAN U-PROGRAM')
```

D.2.4.11 Sample of Messages in Czech

```
N0971 (' M00 STOP PROGRAMU ')
N0972 ('M00 zastavi zpracovani programu ')
N0973 ('soucasti.  Ma-li program pokracovat,  ')
N0974 ('musi operator stisknout tlacitko ')
N0975 ('startu.                              ')
N0976 (' M01 VOLITELNY STOP ')
N0977 ('M01 zastavi zpracovani programu ')
N0978 ('soucasti, pouze kdyz je aktivni funkce ')
N0979 ('VOLITELNY STOP.  Je to zpravidla  ')
N0980 ('tlacitko na strojnim panelu.         ')
N0981 ('Kdyz je funkce volitelneho stopu')
N0982 ('aktivni, musi operator stisknout tlacit.')
N0983 ('start cyklu, aby program pokracoval.  ')
N0984 (' M02 KONEC PROGRAMU ')
N0985 ('M02 ukonci zpracovani          ')
N0986 ('programu soucasti. Po vykonani povelu')
N0987 ('M02 musi operator tlacitkem RESET ')
N0988 ('restartovat zpracovani. ')
N0989 (' M03 VRETENO CW')
N0990 ('M03 zapne otaceni vretena po sm. hod.')
N0991 ('rucicek.  Vreteno se bude  ')
N0992 ('tocit naprogramovanymi otackami.     ')
N0993 ('Zmenu otacek lze provest S kodem.  ')
N0994 (' M04 VRETENO CCW')
N0995 ('M04 zapne otaceni vretena proti sm. hod.')
N0996 ('rucicek.  Vreteno ')
N0997 ('se bude tocit naprogramovanymi       ')
N0998 ('otackami.  Zmenu otacek lze    ')
N0999 ('provest S kodem.                       ')
N1000 (' M05 STOP VRETENA ')
N1001 ('M05 zastavi otaceni vretena.         ')
N1002 (' M06 ZMENA NASTROJE ')
N1003 ('M06 zadava vymenu nastroje v obrabecim')
N1004 ('stroji. Makro pro vymenu nastroje musi')
N1005 ('zastavit vreteno, privod chladiva a')
N1006 ('potom zahajit akce, potrebne pro     ')
N1007 ('vymenu nastroje.                     ')
N1008 ('Po ukonceni vymeny nastroje          ')
N1009 ('zustane vreteno zastavene.                ')
N1010 (' M07 PROUD CHLADIVA ZAP. ')
```

N1011 ('M07 zapina chlazeni nastroje ')
N1012 ('proudem chladiva. Jedna se o')
N1013 ('silny proud chladiva, zamezujici ')
N1014 ('prehrati rezneho nastroje. ')
N1015 (' M08 CHLAZENI MLHOU ZAP. ')
N1016 ('M08 zapina chlazeni nastroje ')
N1017 ('mlhou chladiva. Jedna se o')
N1018 ('chlazeni malym mnozstvim chladiva, ')
N1019 ('kdyz je zatizeni nastroje nizke a ')
N1020 ('material snadno pohlti nadmerne teplo ')
N1021 ('vznikajici pri obrabeni. ')
N1022 (' M09 CHLAZENI VYP. ')
N1023 ('M09 Vypina na stroji proud a ')
N1024 ('mlhu chladiva. Tuto funkci je ')
N1025 ('vhodne pouzit, kdyz je nastroj odtazen')
N1026 ('od soucasti a vyzaduje kontrolu. ')
N1027 ('Povely M05 a M06 by take mely vypnout ')
N1028 ('chlazeni, avsak nelze to zarucit. ')
N1029 (' M19 POZADAVEK ORIENTACE VRETENA ')
N1030 ('M19 provadi orientovani vretena, ')
N1031 ('aby byla hridel vretena blokovana ve ')
N1032 ('zname poloze. Tuto funkci je vhodne ')
N1033 ('pouzit "behem zmeny cyklu. ')
N1034 (' M30 STOP A PREVINUTI PROGRAMU ')
N1035 ('M30 zastavi zpracovani aktualniho ')
N1036 ('programu soucasti a vysle do CNC systemu')
N1037 ('pozadavek resetu a previnuti. Pozadavek')
N1038 ('je take predan ridicimu softwaru stroje ')
N1039 ('pro reset modalnich M kodu. ')
N1040 ('Zpracovani programu soucasti od ')
N1041 ('zacatku je mozne pouhym stisknutim')
N1042 ('tlacitka pro start cyklu. ')
N1043 (' M48 POVOLIT DODAT. ZMENU OTACEK VRET.')
N1044 ('M48 Aktivuje funkci volice dodatecne ')
N1045 ('zmeny otacek vretena na ')
N1046 ('strojnim panelu. Timto volicem muze ')
N1047 ('operator zmenit otacky vretena ')
N1048 ('bez upravy programu soucasti. ')
N1049 (' ')
N1050 (' M49 ZAKAZAT DODAT.ZMENU OTACEK VRET.')
N1051 ('M49 Deaktivuje funkci volice dodatecne')
N1052 ('zmeny otacek vretena na ')
N1053 ('strojnim panelu. V tomto rezimu ')
N1054 ('operator nesmi menit otacky vretena ')
N1055 ('bez upravy programu soucasti. ')
N1056 (' ')
N1057 (' M67 - M69 Povely pro spusteni motoru ')
N1058 ('M67 az M69 jsou M kody, ktere slouzi ')
N1059 ('k zapnuti ruznych zarizeni obrab.stroje.')
N1060 ('Temito zarizenimi jsou vetsinou motory.')
N1061 (' Motor 1 se spousti ')
N1062 ('povelem M67, motor 2 povelem M68 a ')
N1063 ('motor 3 povelem M69. ')
N1064 (' M77 - M79 Povely pro vypnuti motoru ')
N1065 ('M77 az M79 jsou M kody, ktere slouzi ')

```
N1066 ('k vypnuti ruznych zarizeni obrab.stroje.')
N1067 ('Temito zarizenimi jsou vetsinou motory.')
N1068 ('    Motor 1 se vypne                ')
N1069 ('povelem M77, motor 2 povelem M78 a    ')
N1070 ('motor 3 povelem M79.                  ')
N1071 (' M98 VOLANI PODPROGRAMU ')
N1072 ('M98 P9876 ZAVOLA PODPROGRAM CISLO     ')
N1073 ('9876. PODPROGRAMU NELZE PREDAVAT ')
N1074 ('ARGUMENTY. VOLANY PODPROGRAM MUSI BYT')
N1075 ('UKONCEN POVELEM M99.                  ')
N1076 ('POZOR: KDYZ JE V PODPROGRAMU POUZIT   ')
N1077 ('KOD M02 NEBO M30, PODPROGRAM SE NEVRATI ')
N1078 ('DO VOLAJICIHO PROGRAMU A SKONCI.      ')
N1079 (' M99 NAVRAT Z PODPROGRAMU ')
N1080 ('KDYZ JE BEHEM ZPRAC. PROGRAMU ZJISTEN')
N1081 ('KOD M99, PROVEDE SE NAVRAT DO VOLAJICIHO')
N1082 ('PROGRAMU. POKUD BYL PROG. ZPRACOVAN BEZ')
N1083 ('VOLANI Z JINEHO PROGRAMU, ZACNE SE    ')
N1084 ('ZPRACOVAVAT OD ZACATKU V NEURCITE     ')
N1085 ('SMYCCE.                               ')
N1168 (' M00 STOP PROGRAMU')
N1169 (' M01 VOLITELNE ZASTAVENI')
N1170 (' M02 KONEC PROGRAMU')
N1171 (' M03 VRETENO CW')
N1172 (' M04 VRETENO CCW')
N1173 (' M05 STOP VRETENA')
N1174 (' M06 VYMENA NASTROJE')
N1175 (' M07 PLNE CHLAZENI')
N1176 (' M08 CHLAZENI MLHOU')
N1177 (' M09 VYP. CHLAZENI')
N1178 (' M19 ORIENTOVAT VRETENO')
N1179 (' M30 STOP A PREVINUTI PROGRAMU')
N1180 (' M48 POVOLIT DODAT. ZMENU OTACEK VRET')
N1181 (' M49 100 PROC. DODAT. ZM. OTACEK VRET.')
N1182 (' M67 SPUSTIT MOTOR 1')
N1183 (' M68 SPUSTIT MOTOR 2')
N1184 (' M69 SPUSTIT MOTOR 3')
N1185 (' M77 VYPNOUT MOTOR 1')
N1186 (' M78 VYPNOUT MOTOR 2')
N1187 (' M79 VYPNOUT MOTOR 3')
N1188 (' M98 VOLANI PODPROGRAMU')
N1189 (' M99 NAVRAT Z PODPROGRAMU')
```

ADDITIONAL INFORMATION

Addition to FANUC MANUAL GUIDE 0*i*
Common to Lathe System / Machining Center System
OPERATOR'S MANUAL

1. Type of applied technical documents

Name	FANUC MANUAL GUIDE 0*i* OPERATOR'S MANUAL
Spec. No.／Ed.	B-64434EN/02

2. Summary of Change

Group	Name／Outline	New, Add, Correct, Delete	Applicable Date
Basic Function	The new functions are added to Chapter VI.	Add	Immediately
Optional Function			
Unit			
Maintenance Parts			
Notice			
Correction			
Another			

VI. SUPPLEMENTAL INSTRUCTIONS

1 C-axis drilling cycle

Using this function enables a hole machining by positioning in C-axis.

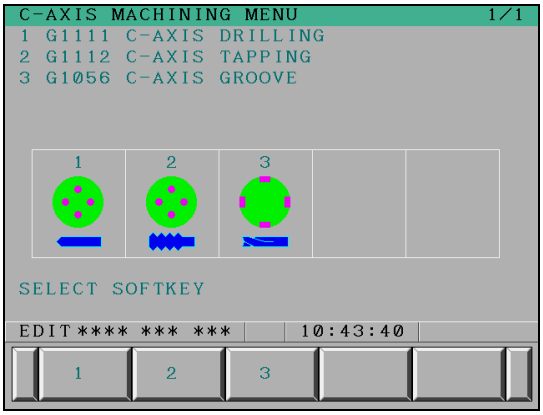
NOTE
1 8.4”color LCD is necessary to use this function.
2 This function is enabled when the parameter No.9302#0 is set to 1.

1.1 OPERATING METHOD

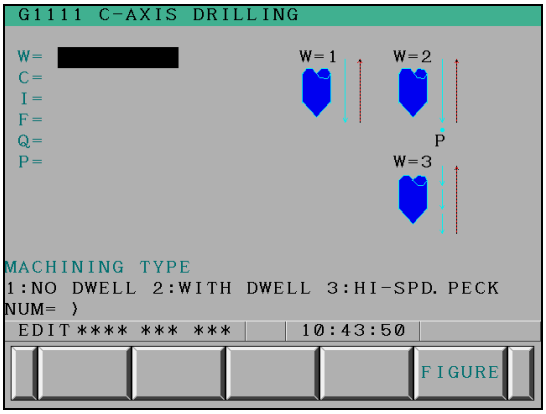
Press the soft key [C-AXIS] on the main menu screen.

(Example of operating procedure)

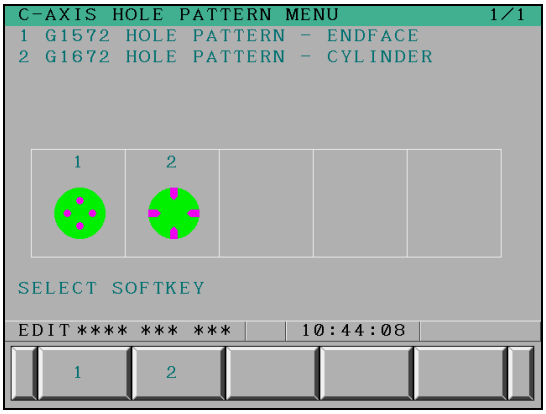
(1) When C-axis machining is selected, the machining menu screen shown below is displayed.



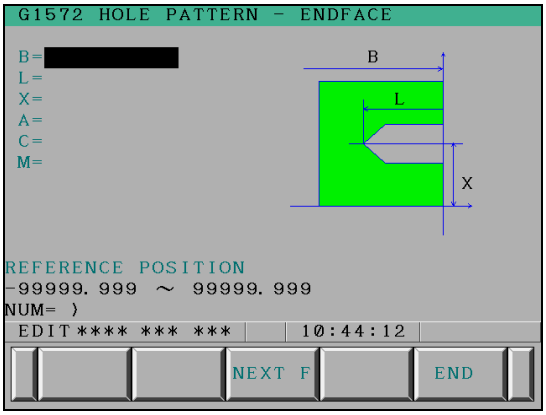
(2) Select the desired machining type with the corresponding soft key. The data input screen of the selected machining type block (when C-axis drilling is selected) as shown below appears. For details on the input items of each machining type block, see the next section. To cancel insertion, press the leftmost soft key.



(3) When the necessary data is entered and soft key [FIGURE] is pressed, a figure menu screen as shown below appears. Only the menu of figures that can be used in the machining type block entered immediately before is displayed. The machining type block entered here is added to the machining program.



(4) When a figure is selected with the corresponding soft key, a data input screen as shown below appears (HOLE PATTERN - ENDFACE is selected in the following figure). For details on the input item of each figure block, see the next section.



(5) When the necessary data has been entered, press soft key [NEXT F] or [END]. When [END] is pressed, the entered figure block is added to the machining program and the main menu is displayed.

1.2 INPUT ITEM

1.2.1 Machining Type Block

C-axis drilling: G1111

Data item		Comment
W	MACHINING TYPE	1 : Drilling without dwell 2 : Drilling with dwell 3 : High speed peck drilling or Peck drilling
C	CLEARANCE	Distance from the surface of workpiece to the start point of actual cutting motion (radius)
I	REFERENCE POSITION RETURN	1 : Return to initial point at moving to the next hole position 2 : Return to R point (start point of actual cutting motion) at moving to the next hole position
F	FEED RATE	Cutting federate
Q	CUTTING DEPTH	Cutting in depth of one cutting motion (radius)
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec

NOTE
1 High speed peck drilling and peck drilling of MACHINING TYPE can be switched by the parameter No.5101#2.
2 The system refers to the parameter No.5114 for the return amount. Therefore, please set parameter No.5114 to the suitable value before machining.

C-axis tapping: G1112

Data item		Comment
W	MACHINING TYPE	1 : Normal tapping 2 : Reverse tapping 3 : Rigid tapping 4 : Reverse rigid tapping
C	CLEARANCE	Distance from the surface of workpiece to the start point of actual cutting motion (radius)
I	REFERENCE POSITION RETURN	1 : Return to initial point at moving to the next hole position 2 : Return to R point (start point of actual cutting motion) at moving to the next hole position
P	DWELL TIME	Dwell time at the bottom of hole, in unit of msec
L	THREADING LEAD	Lead of a tapping thread
S	SPINDLE SPEED	Spindle rotating speed (min-1)

NOTE
1 If MACHING TYPE is normal tapping or reverse tapping, it is not necessary to enter the SPINDLE SPEED. If the SPINDLE SPEED data is input, it will be neglected.
2 When you use rigid tapping mode M code command (bit 0 (G84) of parameter No.5200 = 0), the system refers to No.5210 for the value of M code. Therefore, please set No.5210 to suitable value before machining.

1.2.2 Figure Block

Endface – Hole pattern: G1572

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	BOTTOM –Z	Hole depth (usually negative value)
X	X AXIS POSITION (RADIUS)	X coordinate of a hole position
A	START ANGLE	Center angle of the 1st hole point
C	PITCH ANGLE	Center angle between 2 holes point
M	NUMBER OF HOLE	Number of holes

Cylinder – Hole pattern: G1572

Data item		Comment
B	REFERENCE POSITION (RADIUS)	X-coordinate of the workpiece surface (radius)
L	BOTTOM –X (RADIUS)	Hole depth (radius, usually negative value)
Z	Z AXIS POSITION	Z coordinate of a hole position
A	START ANGLE	Center angle of the 1st hole point
C	PITCH ANGLE	Center angle between 2 holes point
M	NUMBER OF HOLE	Number of holes

1.3 PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
5101						RTR		

RTR = 0 : G83 and G87 specify a high-speed peck drilling cycle.
= 1 : G83 and G87 Specify a peck drilling cycle.

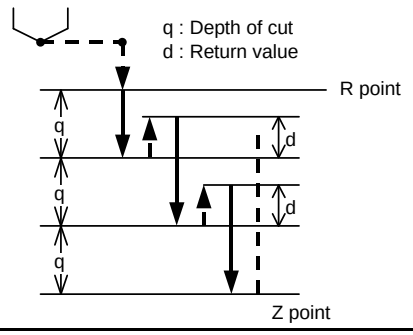
5110	CLAMP M

CLAMP M This parameter sets an M code for C-axis clamping in a drilling canned cycle.
[Valid data range] 0 to 99999998
[Unit of data] 1

5114	

This parameter sets the return value in high-speed peck drilling cycle.
[Valid data range] -999999.999 to +99999999 / -99999.9999 to +99999.9999
[Unit of data] mm / inch

G83 (T series, when the parameter RTR (No.5101#2) is set to 0)

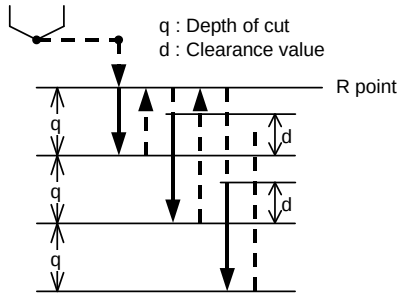


5115	

This parameter sets a clearance value in a peck drilling cycle.

[Valid data range] -999999.999 to +99999999 / -99999.9999 to +99999.9999
[Unit of data] mm / inch

G83 (T series, when the parameter RTR (No.5101#2) is set to 1)



	#7	#6	#5	#4	#3	#2	#1	#0
5200								G84

G84 = 0 : Method for specifying rigid tapping: An M code specifying the rigid tapping mode is specified prior to the issue of the G84 (or G74) command. (See parameter No.5210).
= 1 : Method for specifying rigid tapping: An M code specifying the rigid tapping mode is not used. (G84 cannot be used as a G code for the tapping cycle; G74 cannot be used for the reverse tapping cycle.)

5210	TAP MODE M-CODE

TAP MODE M-CODE This parameter sets an M code that specifies the rigid tapping mode.
The M code is judged to be 29 (M29) when 0 is set.

[Valid data range] 0 to 65535
[Unit of data] 1

	#7	#6	#5	#4	#3	#2	#1	#0
9302								CAX

CAX = 0 : Disables the C-axis drilling.
= 1 : Enables the C-axis drilling.

27005	CLMPM

CLMPM M code for C-axis clamping.
When this parameter is set to 0, nothing is output.
[Valid data range] 0 to 32767
[Unit of data] 1

NOTE
When M code for C-axis clamping is output, it is necessary to set this parameter to the following values.
No.27005 is set to value of No.5110.

27006	UCLMPM

UCLMPM M code for C-axis unclamping.
When this parameter is set to 0, nothing is output.
[Valid data range] 0 to 32767
[Unit of data] 1

NOTE
When M code for C-axis unclamping is output, it is necessary to set this parameter to the following values.
No.27005 is set to value of No.5110 + 1.

27208	MILTMN

MILTMN M code to be specified before tapping by the rotating tool.
Nothing is output when 0 is set.
[Valid data range] 0 to 999999999
[Unit of data] 1

27209	MILTMR

MILTMN M code to be specified before reverse tapping by the rotating tool.
Nothing is output when 0 is set.
[Valid data range] 0 to 999999999
[Unit of data] 1

27210	MLNMLM

MLNMLM M code for turning the rotating tool in the normal direction.
M3 is output when 0 is set.
[Valid data range] 0 to 999999999
[Unit of data] 1

1.4 ALARMS

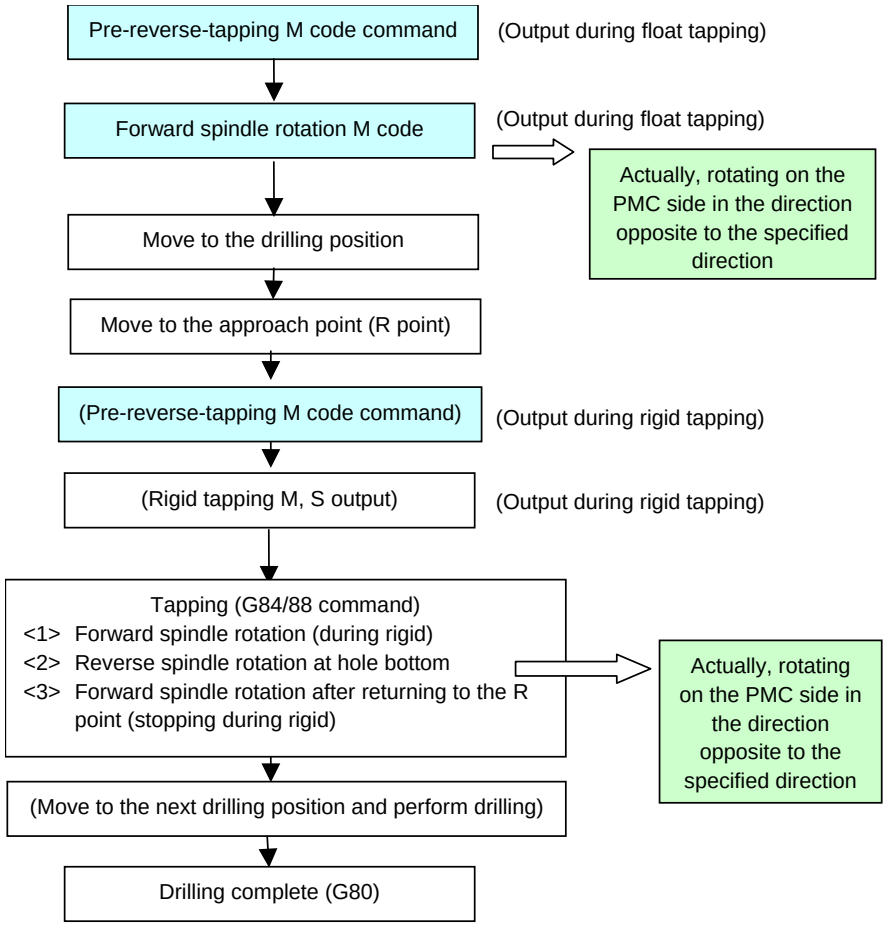
The following alarms might occur when this cycle is executed.

Alarm No.	Description	
3050	Message	NO MACHINING CYCLE BLOCK
	Cause	No machining cycle is found. No machining type block is found; only a figure block is specified.
	Action	Modify the machining program by, for example, adding the necessary machining type block.
3051	Message	WRONG G-CODE
	Cause	No machining cycle is found. An unavailable machining cycle is specified.
	Action	Necessary options may not be added. Contact the machine tool builder.
3054	Message	NO NECESSARY ADDRESS
	Cause	With a cycle machining command or other 4-digit G commands, necessary arguments are not entered.
	Action	Modify the machining program by, for example, adding necessary arguments.
3055	Message	WRONG PARAMETER SETTING
	Cause	The parameter setting is invalid.
	Action	Check the parameter setting.
3056	Message	WRONG MACHINING TYPE
	Cause	The machining type specification is invalid.
	Action	Modify the machining program to specify an appropriate machining type.
3057	Message	WRONG REF. POSITION RETURN
	Cause	The reference position return is invalid.
	Action	Modify the machining program to specify an appropriate reference position return.
3058	Message	WRONG DWELL TIME
	Cause	The dwell time specification is invalid.
	Action	For example, a negative value may be entered as the dwell time. Modify the machining program to specify an appropriate dwell time.
3059	Message	WRONG THREADING LEAD
	Cause	The thread lead specification is invalid.
	Action	For example, a negative value may be entered as the thread lead. Modify the machining program to specify an appropriate thread lead.
3060	Message	WRONG CLEARANCE
	Cause	The clearance is invalid.
	Action	A value not specifiable as a clearance is entered, such as a negative value. Modify the machining program to specify an appropriate clearance.
3061	Message	WRONG FEED RATE
	Cause	The feedrate is invalid.
	Action	A value not specifiable as a feedrate is entered, such as zero. Modify the machining program to specify an appropriate feedrate.

Alarm No.	Description	
3062	Message	WRONG CUTTING DEP. SETTEING
	Cause	The cutting depth is invalid.
	Action	A value not specifiable as turning and grooving or other cutting depths is entered. Modify the machining program to specify an appropriate value.
3063	Message	WRONG SPINDLE SPEED
	Cause	The spindle speed is invalid.
	Action	A value not specifiable as the spindle speed, for example 0, is entered. Modify the machining program to specify an appropriate spindle speed.
3064	Message	MODAL IS OUT OF G0,1,2,3
	Cause	Modal can't be sheltered and restored.
	Action	Command G00, G01, G02 or G03 before the cycle is executed.
3065	Message	MODAL IS OUT OF G80.
	Cause	Modal can't be sheltered and restored.
	Action	Command G80 before the cycle is executed.
3066	Message	WRONG STANDARD POSITION
	Cause	The reference position is invalid.
	Action	A value not specifiable as the reference position of a figure block is entered. Modify the machining program to specify an appropriate value.
3067	Message	WRONG HEIGHT/DEPTH
	Cause	The height/depth is invalid
	Action	A value not specifiable as the height/depth of a figure block is entered. Modify the machining program to specify an appropriate value.
3068	Message	WRONG FIGURE ANGLE
	Cause	The figure angle specification is invalid.
	Action	A value not specifiable as the angle of a figure block is entered. Modify the machining program to specify an appropriate value.
3069	Message	WRONG HOLE/GROOVE NUMBER
	Cause	The number of holes/grooves specification is invalid.
	Action	A value not specifiable as the number of holes or grooves in a figure block is entered. Modify the machining program to specify an appropriate value.
3070	Message	WRONG COORDINATE POSITION
	Cause	The coordinate specification is invalid.
	Action	A value not specifiable as a coordinate of a figure block is entered. Modify the machining program to specify an appropriate value.
3071	Message	MODAL IS OUT OF G40.
	Cause	Modal can't be sheltered and restored.
	Action	Command G40 before the cycle is executed.
3072	Message	NO M-CODE BEFORE TAP CYCL
	Cause	In reverse tapping cycle, parameters for M code to be specified before reverse tapping are not set.
	Action	Set the parameters No.27209 for M code to be specified before reverse tapping.

1.5 LADDER PROGRAM CREATION

To perform reverse tap cycles, a ladder program needs to be created.
In a reverse tap cycle, the M code for reporting reverse rotation is output as shown below. Reverse rotation of the spindle is performed by use of the M code under control of the PMC.

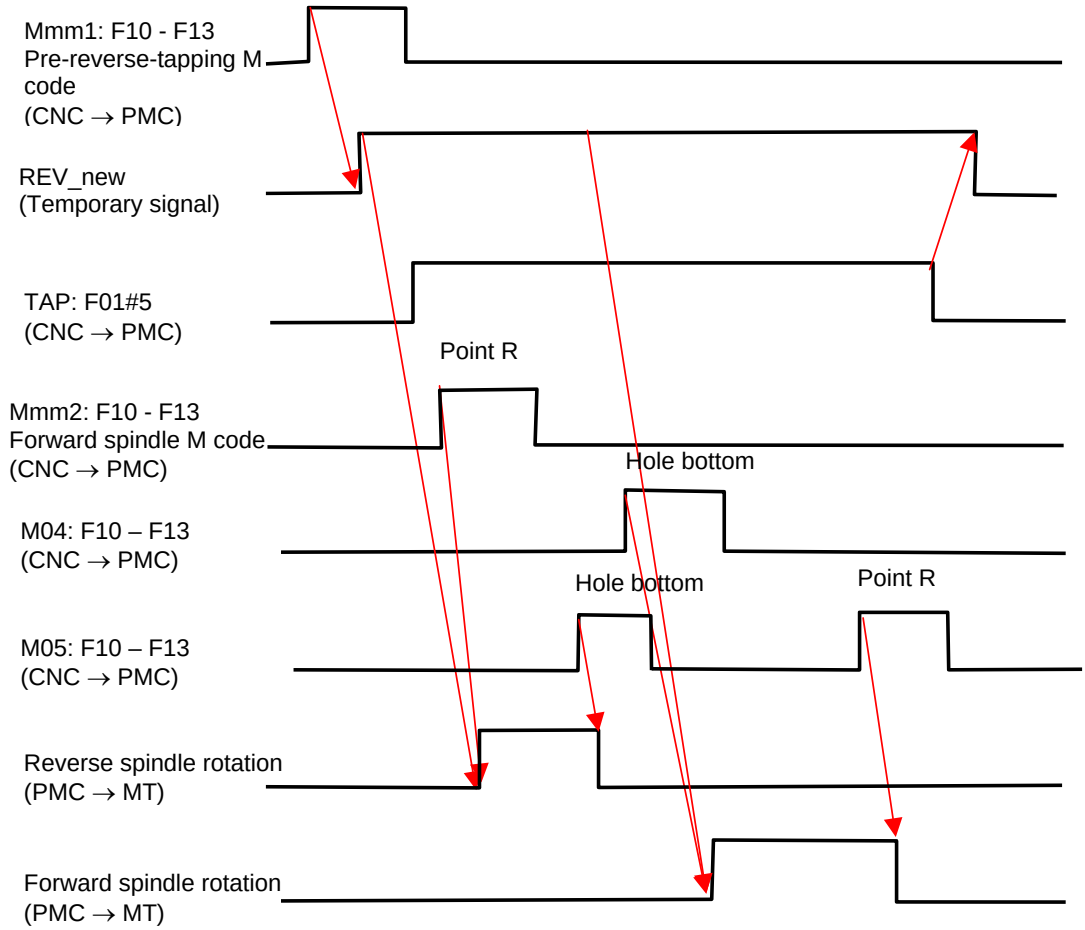


NOTE
The values of the pre-reverse-tapping M code and forward spindle rotation M code are the values of parameters No. 27208 to 27210.

In lathe tap cycle G84, the forward spindle command at the hole bottom always outputs M04 in both forward tapping and reverse tapping. Similarly, when the tool returns to the R point, M03 is output. (In rigid tapping, M03 is output after moving to the R point, M05 is output at the hole bottom, and M05 is output after returning to the R point).
As shown above, the same M code is output from the NC in both forward tapping and reverse tapping, so reverse the actual spindle rotation command on the PMC side in the case of reverse tapping.

On the PMC side, when the above M code receives, it is necessary to provide a signal for reversing the subsequent spindle rotation commands and reverse the spindle rotation commands by detecting the signal. The new signal needs to be canceled by processing, for example, at the trailing edge of the TAP signal (TAP <F0002#5>), to be added to the PMC ladder. The ladder sequence is shown below.
(REV_new is a temporary signal for reversing the spindle rotation.)

Timing chart



2 C-axis grooving cycle

Using this function enables a groove machining by positioning in C-axis.

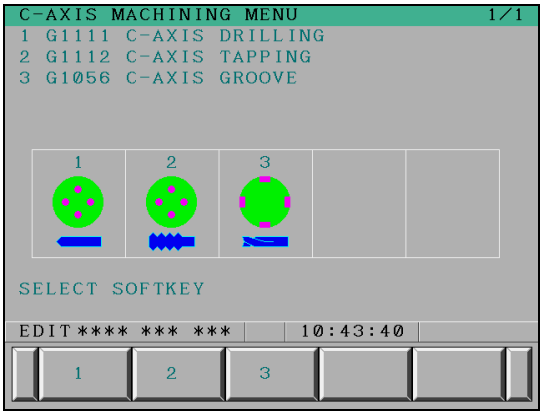
NOTE
1 8.4”color LCD is necessary to use this function.
2 This function is enabled when the parameter No.9302#0 is set to 1.

2.1 OPERATING METHOD

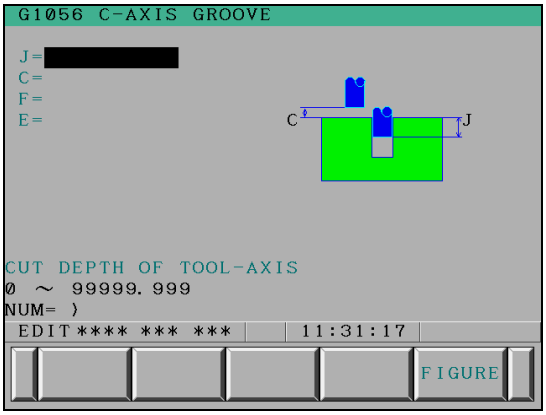
Press the soft key [C-AXIS] on the main menu screen.

(Example of operating procedure)

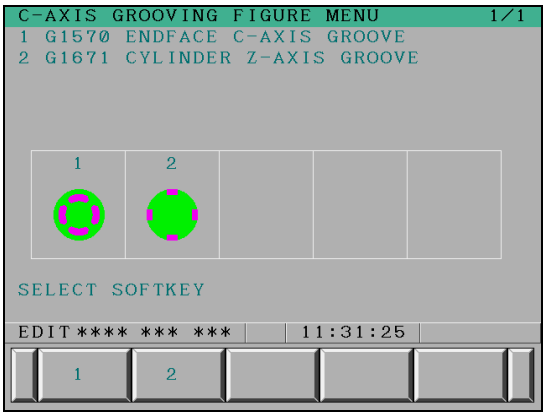
(1) When C-axis machining is selected, the machining menu screen shown below is displayed.



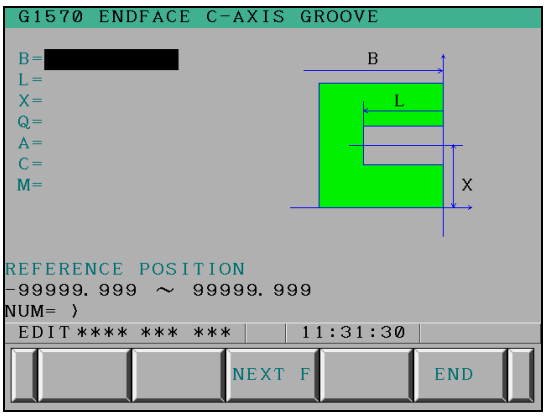
(2) Select the desired machining type with the corresponding soft key. The data input screen of the selected machining type block (when C-axis groove is selected) as shown below appears. For details on the input items of each machining type block, see the next section. To cancel insertion, press the leftmost soft key.



(3) When the necessary data is entered and soft key [FIGURE] is pressed, a figure menu screen as shown below appears. Only the menu of figures that can be used in the machining type block entered immediately before is displayed. The machining type block entered here is added to the machining program.



(4) When a figure is selected with the corresponding soft key, an data input screen as shown below appears (ENDFACE C-AXIS GROOVE is selected in the following figure). For details on the input item of each figure block, see the next section.



(5) When the necessary data has been entered, press soft key [NEXT F] or [END]. When [END] is pressed, the entered figure block is added to the machining program and the main menu is displayed.

2.2 INPUT ITEM

2.2.1 Machining Type Block

C-axis grooving: G1056

Data item		Comment
J	CUT DEPTH OF TOOL-AXIS	Depth of cut in the tool axis direction per cutting operation. If omitted or input 0, one cutting operation is used. (radius)
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point (radius)
F	FEED RATE OF TOOL-RADIUS	Feedrate applicable when cutting is performed in the side face direction of the end mill
E	FEED RATE OF TOOL-AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined

2.2.2 Figure Block

Endface c-axis groove: G1570

Data item		Comment
B	REFERENCE POSITION	Z-coordinate of the workpiece surface
L	GROOVE DEPTH	Distance from workpiece surface to the bottom of pocket. (usually negative value)
X	X AXIS POSITION (RADIUS)	X-coordinate of the center line of a groove (radius)
Q	GROOVE ANGLE (SIZE)	Center angle formed by the start point (tool center) and end point (tool center) of a groove
A	START ANGLE	C coordinate of the start point (tool center) of the first groove
C	PITCH ANGLE	Center angle formed by the start points (tool centers) of two adjacent grooves
M	NUMBER OF GROOVE	Number of grooves to be cut

Cylinder z-axis groove: G1671

Data item		Comment
B	REFERENCE POSITION (RADIUS)	X-coordinate of the workpiece surface (radius)
L	GROOVE DEPTH	Distance from workpiece surface to the bottom of pocket. (radius, usually negative value)
Z	Z AXIS POSITION	Z-coordinate of the start point of a groove
P	GROOVE LENGTH	Distance between the start point (tool center) and end point (tool center) of a groove
A	START ANGLE	C coordinate of the start point (tool center) of the first groove
C	PITCH ANGLE	Center angle formed by the start points (tool centers) of two adjacent grooves
M	NUMBER OF GROOVE	Center angle formed by the start points (tool centers) of two adjacent grooves

2.3 PARAMETERS

	#7	#6	#5	#4	#3	#2	#1	#0
9302								CAX

CAX = 0 : Disables the C-axis drilling.
= 1 : Enables the C-axis drilling.

27005	CLMPM

CLMPM M code for C-axis clamping.
When this parameter is set to 0, nothing is output.
[Valid data range] 0 to 32767
[Unit of data] 1

27006	UCLMPM

UCLMPM M code for C-axis unclamping.
When this parameter is set to 0, nothing is output.
[Valid data range] 0 to 32767
[Unit of data] 1

2.4 ALARMS

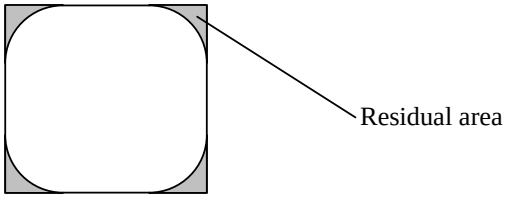
The following alarms might occur when this cycle is executed.

Alarm No.	Description	
3050	Message	NO MACHINING CYCLE BLOCK
	Cause	No machining cycle is found. No machining type block is found; only a figure block is specified.
	Action	Modify the machining program by, for example, adding the necessary machining type block.
3051	Message	WRONG G-CODE
	Cause	No machining cycle is found. An unavailable machining cycle is specified.
	Action	Necessary options may not be added. Contact the machine tool builder.
3054	Message	NO NECESSARY ADDRESS
	Cause	With a cycle machining command or other 4-digit G commands, necessary arguments are not entered.
	Action	Modify the machining program by, for example, adding necessary arguments.
3055	Message	WRONG PARAMETER SETTING
	Cause	The parameter setting is invalid.
	Action	Check the parameter setting.
3060	Message	WRONG CLEARANCE
	Cause	The clearance is invalid.
	Action	A value not specifiable as a clearance is entered, such as a negative value. Modify the machining program to specify an appropriate clearance.
3061	Message	WRONG FEED RATE
	Cause	The feedrate is invalid.
	Action	A value not specifiable as a feedrate is entered, such as zero. Modify the machining program to specify an appropriate feedrate.
3064	Message	MODAL IS OUT OF G0,1,2,3
	Cause	Modal can't be sheltered and restored.
	Action	Command G00, G01, G02 or G03 before the cycle is executed.
3065	Message	MODAL IS OUT OF G80.
	Cause	Modal can't be sheltered and restored.
	Action	Command G80 before the cycle is executed.
3066	Message	WRONG STANDARD POSITION
	Cause	The reference position is invalid.
	Action	A value not specifiable as the reference position of a figure block is entered. Modify the machining program to specify an appropriate value.
3067	Message	WRONG HEIGHT/DEPTH
	Cause	The height/depth is invalid
	Action	A value not specifiable as the height/depth of a figure block is entered. Modify the machining program to specify an appropriate value.

Alarm No.	Description	
3068	Message	WRONG FIGURE ANGLE
	Cause	The figure angle specification is invalid.
	Action	A value not specifiable as the angle of a figure block is entered. Modify the machining program to specify an appropriate value.
3069	Message	WRONG HOLE/GROOVE NUMBER
	Cause	The number of holes/grooves specification is invalid.
	Action	A value not specifiable as the number of holes or grooves in a figure block is entered. Modify the machining program to specify an appropriate value.
3070	Message	WRONG COORDINATE POSITION
	Cause	The coordinate specification is invalid.
	Action	A value not specifiable as a coordinate of a figure block is entered. Modify the machining program to specify an appropriate value.
3071	Message	MODAL IS OUT OF G40.
	Cause	Modal can't be sheltered and restored.
	Action	Command G40 before the cycle is executed.
3074	Message	WRONG CUTTING AREA
	Cause	The machining area is invalid.
	Action	Modify the machining program to specify a correct machining area, such as a part figure and a blank figure in a machining profile.
3075	Message	WRONG CUTTING CONDITION
	Cause	The cutting conditions are invalid.
	Action	Modify the machining program to specify normal cutting conditions such as the feedrate.
3076	Message	WRONG CUTTING DEPTH
	Cause	The depth of cut is invalid.
	Action	A value not specifiable as a depth of cut is entered, such as a negative value. Modify the machining program to specify an appropriate depth of cut.
3077	Message	WRONG FIGURE LENGTH
	Cause	The figure length is invalid.
	Action	A value not specifiable as the length of a figure block is entered. Modify the machining program to specify an appropriate value.
3078	Message	WRONG PITCH SETING
	Cause	The pitch specification is invalid.
	Action	A value not specifiable as the pitch angle of a figure block is entered. Modify the machining program to specify an appropriate value.

3 Residual Machining cycle

This cycle can machine residual area in a corner after pocket machining and so on.



3.1 Format of G code

(Format of G code)
G1029 Hh Vv Pp Rr Ii Jj Bb Dd Ww Kk Ll Qq Xx Cc Ff Ee;

(Argument of G code)

Address	Meaning	Valid range
H	REFERENCE POSITION OF CORNER X	Metric: -99999.999 - 99999.999 Inch : -9999.9999 - 9999.9999
V	REFERENCE POSITION OF CORNER Y	
P	CORNER DIRECTION	1 : Lower left, 2 : Upper left, 3 : Lower right, 4 : Upper right
R	CORNER R	Metric: 0 - 99999.999 Inch : 0 - 9999.9999
I	CUTTING WIDTH X-AXIS	
J	CUTTING WIDTH Y-AXIS	
B	HEIGHT OF CUTTING SURFACE Z	Metric: -99999.999 - 99999.999 Inch : -9999.9999 - 9999.9999
D	TOOL DIAMETER	Metric: 0 - 99999.999 Inch : 0 - 9999.9999
W	UP CUT/DOWN CUT	1 : Down cutting, 2 : Up cutting
K	SIDE FINISH AMOUNT	Metric: 0 - 99999.999 Inch : 0 - 9999.9999
L	BOTTOM FINISH AMOUNT	
Q	APPROACH DISTANCE	
X	ESCAPE DISTANCE	
C	CLEARANCE OF TOOL-AXIS	
F	FEEDARATE OF TOOL-RADIUS	Metric: 1.00 - 240000 Inch : 1.00 - 9600
E	FEEDARATE OF TOOL-AXIS	

3.2 The operation to create Residual Machining

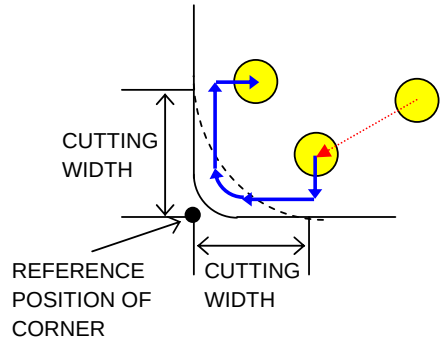
The residual machining cycle is created by the following operation.

(1) Select POCKET machining on the main menu screen and the machining cycle menu screen is shown.
(2) Select residual machining cycle menu on the second page and the data input screen for residual machining cycle is displayed.
(3) Input necessary data and press soft key [NEXT F] or [END].
When [END] is pressed, G code for residual machining cycle is inserted to a program.

Input items		Note
H	REFERENCE POSITION OF CORNER X	Reference position of corner (X axis)
V	REFERENCE POSITION OF CORNER Y	Reference position of corner (Y axis)
P	CORNER DIRECTION	A direction there is a corner
R	CORNER R	Radius of corner
I	CUTTING WIDTH X-AXIS	Cutting width in a corner (X axis)
J	CUTTING WIDTH Y-AXIS	Cutting width in a corner (Y axis)
B	HEIGHT OF CUTTING SURFACE Z	The height of the cutting surface (Z axis)
D	TOOL DIAMETER	Tool diameter
W	UP CUT/DOWN CUT	1 : Down cutting 2 : Up cutting
K	SIDE FINISH AMOUNT	Finishing amount of the side wall part
L	BOTTOM FINISH AMOUNT	Finishing amount of the bottom part
Q	APPROACH DISTANCE	Approach distance in the tool radius direction
X	ESCAPE DISTANCE	Escape distance in the tool radius direction
C	CLEARANCE OF TOOL-AXIS	Distance from the surface of workpiece to the approaching point in the tool axis direction (Z axis)
F	FEEDARATE OF TOOL-RADIUS	Cutting feedrate in the tool radius direction
E	FEEDARATE OF TOOL-AXIS	Cutting feedrate in the tool axis direction (Z axis)

3.3 Tool path of Residual Machining cycle

The following tool path is created by execution of residual machining cycle.



(1) The tool approaches to the approach point.
(2) The tool moves to the height of the cutting surface.
(3) The residual area in the corner is machined.
(4) The tool escapes by escape distance.
(5) The tool returns in Z axis direction to the position before the start of the machining cycle.

3.4 Condition to be used

It can be used for only 8.4” display.

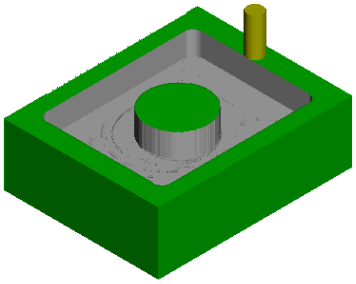
3.5 Alarms

The following alarms might occur when this cycle is executed.

Alarm No.	Description	
3001	Message	INPUT VALUE IS ERRONEOUS.
	Cause	Necessary data is not entered. Or entered data is invalid.
	Action	Display input data screen of the block data, at which the alarm occurred, and enter the correct data after confirming it.
3028	Message	MODAL IS OUT OF G17.
	Cause	Modal can't be sheltered and restored.
	Action	Command G17 before the cycle is executed.
3029	Message	MODAL IS OUT OF G90.
	Cause	Modal can't be sheltered and restored.
	Action	Command G90 before the cycle is executed.
3059	Message	WRONG TOOL DIAMETER
	Cause	The tool diameter is invalid.
	Action	A value not specifiable as a tool diameter is entered, such as a negative value. Modify the machining program to specify an appropriate tool diameter.
3062	Message	WRONG CLEARANCE
	Cause	The clearance is invalid.
	Action	A value not specifiable as a clearance is entered, such as a negative value. Modify the machining program to specify an appropriate clearance.
3063	Message	WRONG FINISHING ALLOWANCE
	Cause	The finishing allowance is invalid.
	Action	A value not specifiable as a finishing allowance is entered, such as a negative value. Modify the machining program to specify an appropriate finishing allowance.
3064	Message	WRONG FEEDRATE
	Cause	The feedrate is invalid.
	Action	A value not specifiable as a feedrate is entered, such as zero. Modify the machining program to specify an appropriate feedrate.
3070	Message	WRONG CORNER SETTING
	Cause	The corner rounding is invalid.
	Action	A value not specifiable as the corner rounding is entered. Modify the machining program to specify an appropriate value.
3071	Message	MODAL IS OUT OF G40.
	Cause	Modal can't be sheltered and restored.
	Action	Command G40 before the cycle is executed.

4 Pocket machining cycle with islands

Pocket machining cycle with islands is available by this function.



4.1 Format of program

4.1.1 Format of the program of Pocket machining cycle with islands

Pocket machining cycle with islands is made from the machining condition block and figure blocks. And, figure blocks is made from pocket figure block and island figure block.

(Format of the program of pocket machining cycle with islands)

G1049 Dd Tt Ll Jj Kk Hh Ff Ee Ww Bb Cc ;	Machining condition block
G1990 ;	Figure block start
G1200 T3 Hh Vv Bb Ll Pp Qq Uu Ww Bb Ll Rr ;	Pocket figure block
～	
G1206 ;	
G1200 T2 Hh Vv Bb Ll Pp Qq Uu Ww Bb Ll Rr ;	Island figure block
～	
G1206 ;	
G1991 ;	Figure block end

Figure block

* 16 islands can be defined .
* Pocket figure block and island figure block are made from G1200~G1206.
Please refer to the next paragraph for details.

4.1.2 Machining condition block

G1049 : Command of machining condition

Data item		Description
D	TOOL DIAMETER	Tool diameter.
T	BOTTOM THICKNESS	Cutting allowance of the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)
L	CUT DEPTH OF RADIUS	Depth of cut on the side face (tool radius direction) per cutting operation (radius value, positive value)
J	CUT DEPTH OF AXIS	Depth of cut in the tool axis direction per cutting operation (radius value, positive value)
K	SIDE FINISH AMOUNT	Finishing allowance on the side face. The blank is regarded as 0. (radius value, positive value)
H	BOTTOM FINISHI AMT	Finishing allowance at the bottom in side face machining. The blank is regarded as 0. (radius value, positive value)
F	FEED RATE- SING.CUT	Feedrate applicable when an end mill is used for cutting.
E	FEED RATE- AXIS	Feedrate applicable when cutting is performed in the tool axis direction toward the bottom of a side face being machined
W	UP CUT/DOWN CUT	Performs machining in up-cut/down-cut mode, assuming that the tool is rotating clockwise. 1:up-cut, 2:down-cut
B	CLEARANCE OF RADIUS	Distance between the wall of a pocket and a tool retract position in the tool radius direction (radius value, positive value)
C	CLEARANCE OF AXIS	Distance between the surface of a blank being machined and a cutting start point (point R) in the tool axis direction (radius value, positive value)

4.1.3 Figure block

4.1.3.1 Pocket figure block

G1200 : Command of pocket figure start

Data item		Description
P	CENTER POINT (X-AXIS)	Center point X-coordinate of the pocket
Q	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the pocket
U	LENGTH ALONG TO THE X-AXIS	Length of square side along X-axis
W	W LENGTH ALONG TO THE Y-AXIS	Length of square side along Y-axis
B	BASE POSITION	Z coordinate of the top surface of a workpiece
L	HEIGHT/DEPTH	Height / depth of pocket
R	R CORNER RADIUS	Radius of each corner of the square. The blank is regarded as 0.
T	FIGURE TYPE	Type of figure (T3 = pocket)
H	START POINT X	X coordinate of the start point of the pocket figure
V	START POINT Y	Y coordinate of the start point of the pocket figure

G1201 : Command of pocket figure straight line

Data item		Description
H	END POINT (X-AXIS)	X coordinate of the end point of the line
V	END POINT (Y-AXIS)	Y coordinate of the end point of the line

G1202,G1203 : Command of pocket figure arc

Data item		Description
H	END POINT (X-AXIS)	X coordinate of the end point of the arc
V	END POINT (Y-AXIS)	Y coordinate of the end point of the arc
R	ARC RADIUS	Arc radius of the pocket figure
I	CENTER POINT (X-AXIS)	Center point X-coordinate of the arc of the pocket
J	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the arc of the pocket

G1206 : Command of pocket figure end

Note
Please do not delete or change the block and data in the program editor of NC.
The operation by this cycle cannot be supported when the block is deleted or changed in the program editor of NC.

4.1.3.2 Island figure block

G1200 : Command of island figure start

Data item		Description
P	CENTER POINT (X-AXIS)	Center point X-coordinate of the island
Q	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the island
U	LENGTH ALONG TO THE X-AXIS	Length of square side along X-axis
W	W LENGTH ALONG TO THE Y-AXIS	Length of square side along Y-axis
B	BASE POSITION	Z coordinate of the top surface of a workpiece
L	HEIGHT/DEPTH	Height / depth of island
R	R CORNER RADIUS	Radius of each corner of the square. The blank is regarded as 0.
T	FIGURE TYPE	Type of figure (T2 = island)
H	START POINT X	X coordinate of the start point of the island figure
V	START POINT Y	Y coordinate of the start point of the island figure

G1201 : Command of island figure straight line

Data item		Description
H	END POINT (X-AXIS)	X coordinate of the end point of the line
V	END POINT (Y-AXIS)	Y coordinate of the end point of the line

G1202,G1203 : Command of island figure arc

Data item		Description
H	END POINT (X-AXIS)	X coordinate of the end point of the arc
V	END POINT (Y-AXIS)	Y coordinate of the end point of the arc
R	ARC RADIUS	Arc radius of the island figure
I	CENTER POINT (X-AXIS)	Center point X-coordinate of the arc of the island
J	CENTER POINT (Y-AXIS)	Center point Y-coordinate of the arc of the island

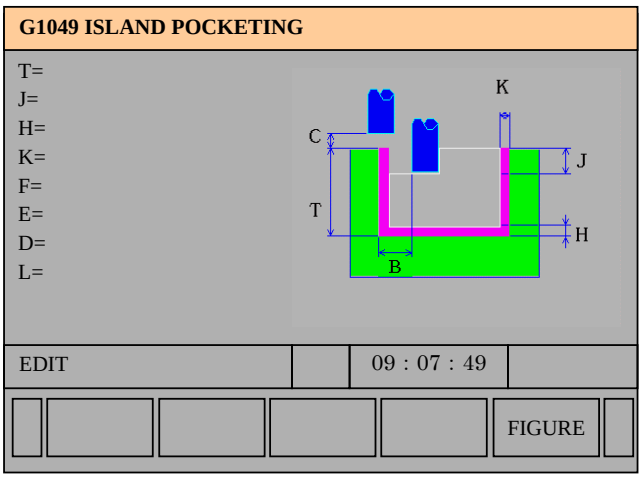
G1206 : Command of island figure end

Note
Please do not delete or change the block and data in the program editor of NC.
The operation by this cycle cannot be supported when the block is deleted or changed in the program editor of NC.

4.2 Method for creation of program

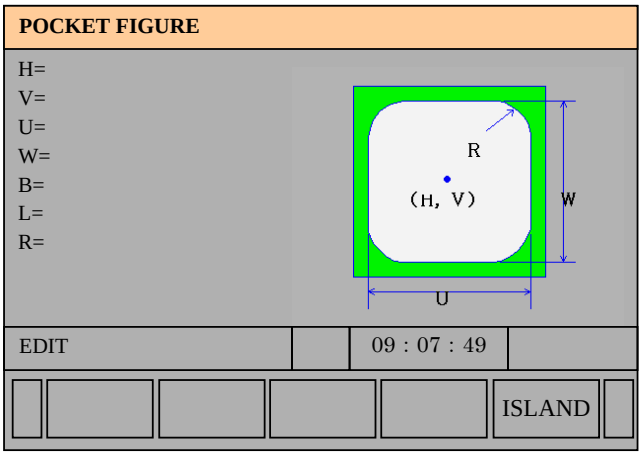
4.2.1 New creation of program

<1> On the main menu screen, press the soft key [POCKET].
<2> On the cycle menu screen, change a page by mdi key [↓ PAGE]/[↑ PAGE], then, press the [6] soft key. (Select the “6 G1049 Island Pocketing” on the screen.)
<3> Input the items on the following data input screen of the machining condition block.



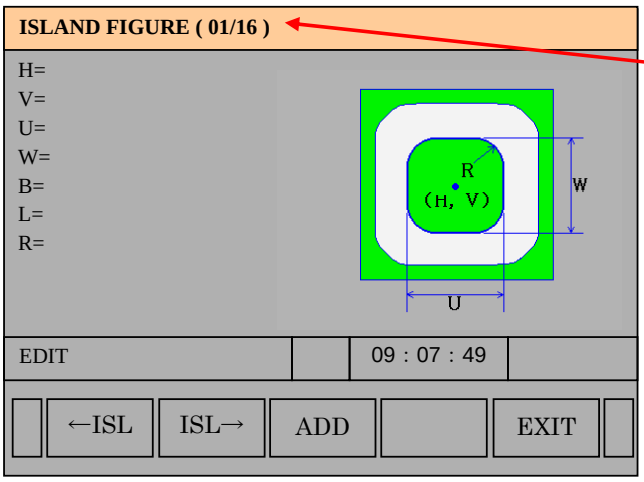
[FIGURE] soft key : Machining condition block is inserted to the program, and data input screen of the pocket figure block is displayed.
Left [ ] soft key : Inserting machining condition block is canceled.

<4> Input the items on the following data input screen of pocket figure block.



[ISLAND] soft key : Machining condition block is recorded to the temporary, and data input screen of the island figure block is displayed.
Left [ ] soft key : Inserting pocket figure block is canceled.

<5> Input the items on the following data input screen of island figure block.



The number of island

(Editing number / Total numer)

[ADD] soft key : The data input screen for input newly island is displayed. At this time, input data of island figure block under the edit, is recorded. Then TOTAL number is increased by one.

[←ISL] soft key : The data input screen for the island of the previous number is displayed.

[ISL→] soft key : The data input screen for the island of the next number is displayed.

[EXIT] soft key : All input pocket figure blocks and island figure block is inserted or updated to the program. Then, the main menu screen is displayed.

Left [ ] soft key : When the number of island is 1, inserting island figure block is canceled.

Note

All input pocket figure block and island figure block is not inserted or updated until [EXIT] soft key is pressed.

4.2.2 Modification of program

(1) For modification of machining condition figure block
<1> In program editor of NC, by pressing the [C.A.P.] soft key when the cursor is on a G1049 block, the data input screen for the machining condition block is displayed.
<2> Press the [OK] soft key after editing the input items. Then, G1049 block is rewritten by the value input to the input item, and program editor of NC is displayed.

(1) For modification of pocket figure block and island figure block
<1> In program editor of NC, by pressing the [C.A.P.] soft key when the cursor is on a G1990 block, the data input screen for the pocket figure block is displayed.
<2> Press the [ISLAND] soft key after editing the input item. Then, the data input screen for the island figure block is displayed.
<3> Select the island to be edit by [←ISL] / [ISL→] soft key, and edit the input items.
<4> When all of editing is done, press the [EXIT] soft key. Then, the data of the pocket figure block and island figure block is rewritten, and program editor of NC is displayed.

Note
The figure block in this cycle can not be inserted / edited, by [FIGURE] soft key on the main menu screen.

4.3 Alarm message

The following alarm messages might be occurred, when the Pocket machining cycle with islands is executed.

Alarm No.	Description	
3050	Message	NO CUTTING AREA
	Cause	There is no area that can be machined. The tool is too large for the specified machining area.
	Action	Modify the machining program to use a smaller tool.
3053	Message	NO MACHINING CYCLE BLOCK
	Cause	No machining cycle is found. No machining type block is found; only a figure block is specified.
	Action	Modify the machining program by, for example, adding the necessary machining type block.
3054	Message	WRONG G-CODE
	Cause	No machining cycle is found. An unavailable machining cycle is specified.
	Action	Necessary options may not be added. Contact the machine tool builder.
3056	Message	NO NECESSARY ADDRESS
	Cause	With a cycle machining command or other 4-digit G commands, necessary arguments are not entered.
	Action	Modify the machining program by, for example, adding necessary arguments.
3057	Message	WRONG CUTTING DIRECTION
	Cause	The cutting direction specification is invalid.
	Action	A value not specifiable as turning or other cutting directions is entered. Modify the machining program to specify an appropriate cutting direction.
3058	Message	WRONG THICKNESS
	Cause	The surplus thickness specification is invalid.
	Action	A value not specifiable as pocketing or other surplus thicknesses is entered, such as a negative value. Modify the machining program to specify an appropriate surplus thickness.
3059	Message	WRONG TOOL DIAMETER
	Cause	The cutter radius is invalid.
	Action	A value not specifiable as a cutter radius is entered, such as a negative value. Modify the machining program to specify an appropriate cutter radius.
3060	Message	WRONG CUTTING DEPTH
	Cause	The depth of cut is invalid.
	Action	A value not specifiable as a depth of cut is entered, such as a negative value. Modify the machining program to specify an appropriate depth of cut.
3062	Message	WRONG CLEARANCE
	Cause	The clearance is invalid.
	Action	A value not specifiable as a clearance is entered, such as a negative value. Modify the machining program to specify an appropriate clearance.

Alarm No.	Description	
3063	Message	WRONG FINISHING ALLOWANCE
	Cause	The finishing allowance is invalid.
	Action	A value not specifiable as a finishing allowance is entered, such as a negative value. Modify the machining program to specify an appropriate finishing allowance.
3064	Message	WRONG FEED RATE
	Cause	The feedrate is invalid.
	Action	A value not specifiable as a feedrate is entered, such as zero. Modify the machining program to specify an appropriate feedrate.
3065	Message	MODAL IS OUT OF G0,1,2,3
	Cause	Modal can't be save and restored.
	Action	Command G00, G01, G02 or G03 before the cycle is executed.
3066	Message	MODAL IS OUT OF G80.
	Cause	Modal can't be save and restored.
	Action	Command G80 before the cycle is executed.
3067	Message	WRONG FIGURE TYPE
	Cause	The figure type is invalid.
	Action	A type not specifiable as the figure type of a figure block is selected. Specify correct type.
3068	Message	WRONG STANDARD POSITION
	Cause	The reference position is invalid.
	Action	A value not specifiable as the reference position of a figure block is entered. Specify correct value.
3069	Message	WRONG HEIGHT/DEPTH
	Cause	The height/depth is invalid.
	Action	A value not specifiable as the height/depth of a figure block is entered. Specify correct value.
3071	Message	MODAL IS OUT OF G40.
	Cause	Modal can't be save and restored.
	Action	Command G40 before the cycle is executed.
3072	Message	GROUP SET MISMATCH
	Cause	The arbitrary figure group specification is not correct.
	Action	In the group specification used in, for example, pocketing with islands, either the start or end block is not entered. Enter an appropriate block.
3073	Message	NUMBER OF ISLANDS EXCEED MAX.
	Cause	The number of islands defined in one pocket has exceeded the limitation.
	Action	Decrease the number of islands or divide the machining cycle.
3098	Message	NO MEMORY
	Cause	There is no memory by an internal problem..
	Action	—
3099	Message	SYSTEM ERROR
	Cause	system error by an internal problem occurred.
	Action	—

Addition to FANUC MANUAL GUIDE 0*i*
OPERATOR'S MANUAL

1. Type of applied technical documents

Name	FANUC MANUAL GUIDE 0*i* OPERATOR'S MANUAL
Spec. No./Ed.	B-64434EN/02

2. Summary of Change

Group	Name/Outline	New, Add, Correct, Delete	Applicable Date
Basic Function	Series 0*i*-F is available.	Add	Immediately
Optional Function			
Unit			
Maintenance Parts			
Notice	This manual is a common manual with Series 0*i*-D.		
Correction			
Another			

ABOUT USE WITH Series 0*i*-F

The usage of MANUAL GUIDE 0*i* in Series 0*i*-F is basically the same as that in Series 0*i*-D.
Please read the explanation of Series 0*i*-D as the explanation of Series 0*i*-F.
The points noted with the Series 0*i*-F is follows.

Replace the following description to Section 1,"MAIN MENU SCREEN" in Part II, "BASIC SCREEN AND OPERATIONS".

NOTE

1 MANUAL GUIDE 0*i* requires a work program that stores a CNC program to be created temporarily. Set parameter No. 9358 to the number of this work program.
(If this parameter is not set, the screen of MANUAL GUIDE 0*i* does not appear.)

2 The program to be displayed by soft key [C.A.P.] depends on the cursor position on the NC program edit screen.
- When the cursor positions in the block containing a 4-digit code for machining cycle, the machining cycle data input screen is displayed.
- When the cursor positions in a block not containing a 4-digit code for machining cycle, the main menu screen is displayed.

3 When soft key [C.A.P.] is pressed if there is no CNC program, a warning saying "INITIALIZATION OF MG0I CANNOT BE COMPLETED." is displayed. Be sure to create one or more CNC programs before pressing soft key [C.A.P.].

4 In Series 0*i*-F, as a program is managed by the program folder, the program outputted by MANUAL GUIDE 0*i* is created on the current program folder.

5 In Series 0*i*-F, a program with arbitrary program name can be used. But, in case of selecting the program as a main program, MANUAL GUIDE 0*i* cannot be started up. In case of starting it, the warning is displayed.

INDEX

REVISION RECORD

Edition	Date	Contents
02	Dec., 2010	・ Addition of the explanation on the condition for using MANUAL GUIDE 0*i* ・ Division of the explanation on the contour figure input and turning figure input ・ Addition of the alarm number and the explanation of the added alarm ・ Deletion of the explanation on the customization of G-code support and Process support ・ Correction of errors, etc
01	Jul., 2008	

B-64434EN/02

B-64434EN/02.02