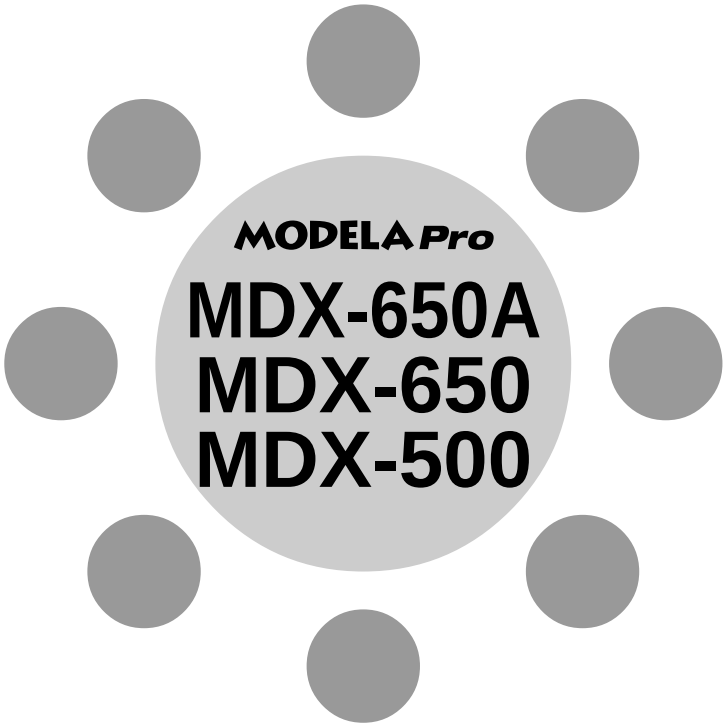
Roland



USER'S MANUAL 3

Cutting Using NC Code

Thank you very much for purchasing the product.

- To ensure correct and safe usage with a full understanding of this product's performance, please be sure to read through this manual completely and store it in a safe location.
- Unauthorized copying or transferral, in whole or in part, of this manual is prohibited.
- The contents of this operation manual and the specifications of this product are subject to change without notice.
- The operation manual and the product have been prepared and tested as much as possible. If you find any misprint or error, please inform us.
- Roland DG Corp. assumes no responsibility for any direct or indirect loss or damage which may occur through use of this product, regardless of any failure to perform on the part of this product.
- Roland DG Corp. assumes no responsibility for any direct or indirect loss or damage which may occur with respect to any article made using this product.

Roland DG Corporation

Contents

Introduction 2
Setting the Instruction Set to NC Code 2
Choosing the Spindle Type 2

Part 1 Basic Operation

1-1 Making Settings Using the Liquid-crystal Display 4
MDX-650A/650 4
MDX-500 5
1-2 Setting the Connection Parameters 6
1-3 Setting a Workpiece Coordinate System 7
Workpiece Coordinates Not Specified 7
Specified with G54 Through G59 8
Specified with G92 8
When Specifying EXOFS (External Workpiece-origin Offset) 9
1-4 Downloading Cutting Data 10
Postprocessing 10
Selecting the Character Code 11
Using Subprograms 12
Using a Communications Application for NC 14
Using the MS-DOS Prompt in Windows 14
Adjusting the Rotating Speed and Feed Speed 15
Terminating Cutting 16
1-5 Finishing 17

Part 2 User's Reference

2-1 Cutting Area 19
MDX-650A/650 19
MDX-500 21
2-2 Operating Each Function 23
Changing to Japanese Messages on the Liquid-crystal Display 23
Performing Repeat Cutting 23
Pausing Operation and Adjusting the Rotating Speed and Feed Speed 25
Terminating Cutting 25
2-3 Descriptions of the Display Menus 26
1 SPEED OVER RIDE 26
2 SPINDLE RPM 27
3 MECHA MOVING 27
4 TOOL DIAMETER 27
5 MOVING MODE 28
6 USE CODE 30
7 CONNECTION 30
8 SERIAL PARAMETER 30
9 COMPENSATE 31
10 SUB-PROGRAM 32
11 OTHERS 33
12 Rotary Control (MDX-650A/650 only) 35
13 ATC (MDX-650A only) 35
14 To Coordinate 36
2-4 What to Do If... 37
When the machine does not work... 37
When the spindle does not rotate 37
Data cannot be sent 38
The power does not come on... 38
2-5 Error Messages 39
2-6 Other Messages 41
2-7 Display Menus Flowchart 42
MDX-650A/650 42
MDX-500 47

Windows® and MS-DOS are registered trademarks or trademarks of Microsoft® Corporation in the United States and/or other countries.
Other company names and product names are trademarks or registered trademarks of their respective holders.

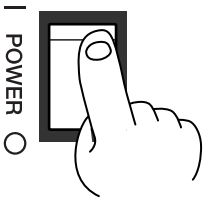
Copyright © 2001 Roland DG Corporation

Introduction

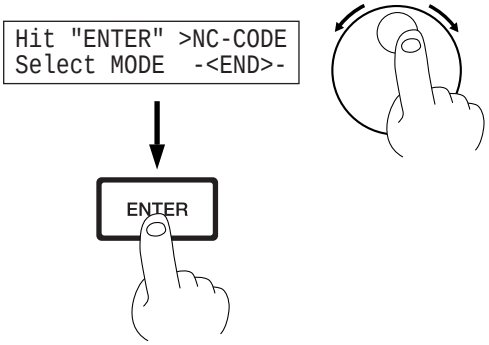
Setting the Instruction Set to NC Code

This chooses the instruction set immediately after switching on the power. When an instruction set has been chosen, it cannot be changed until the power is reset.

1 Switch on the power.
After the opening message, the screen for selecting the instruction set appears.



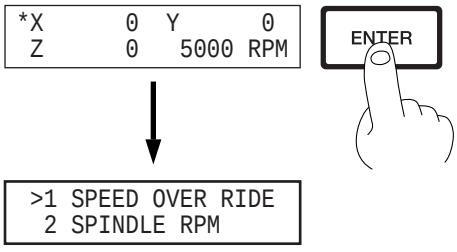
2 Turn the dial to move the arrow to [NC-CODE], then press the [ENTER] key.



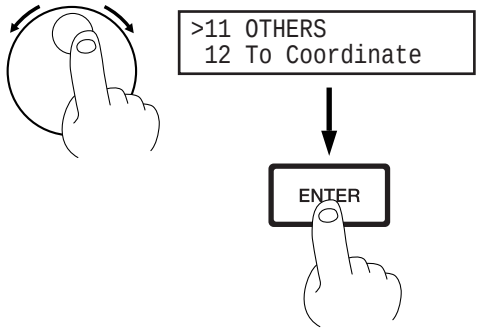
Choosing the Spindle Type

This sets the type of the installed spindle.
If a high-torque spindle is installed, choose [HIGH TORQUE]. If a high-speed spindle is installed, choose [HIGH SPEED]. An incorrect selection may result in insufficient power to the motor and make normal cutting impossible, or conversely may apply power beyond the rated capacity to the motor and cause an error to be displayed during cutting.

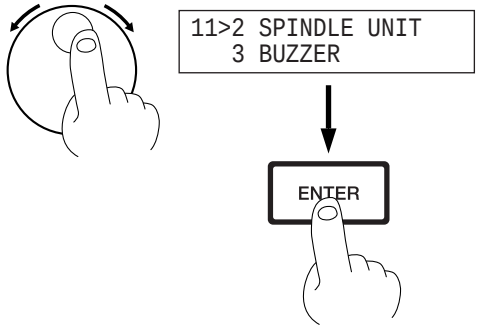
1 If the display shows coordinates, press the [ENTER] key to display the main menu.



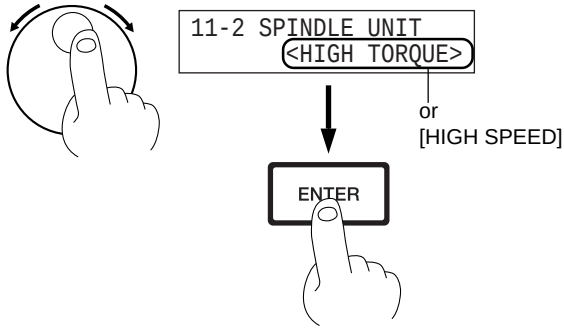
2 Turn the dial to move the arrow to [OTHERS], then press the [ENTER] key.



3 Turn the dial to move the arrow to [SPINDLE UNIT], then press the [ENTER] key.



4 Turn the dial to move the arrow to [HIGH TORQUE] or [HIGH SPEED], then press the [ENTER] key.
The selected mode is enclosed in angle brackets.

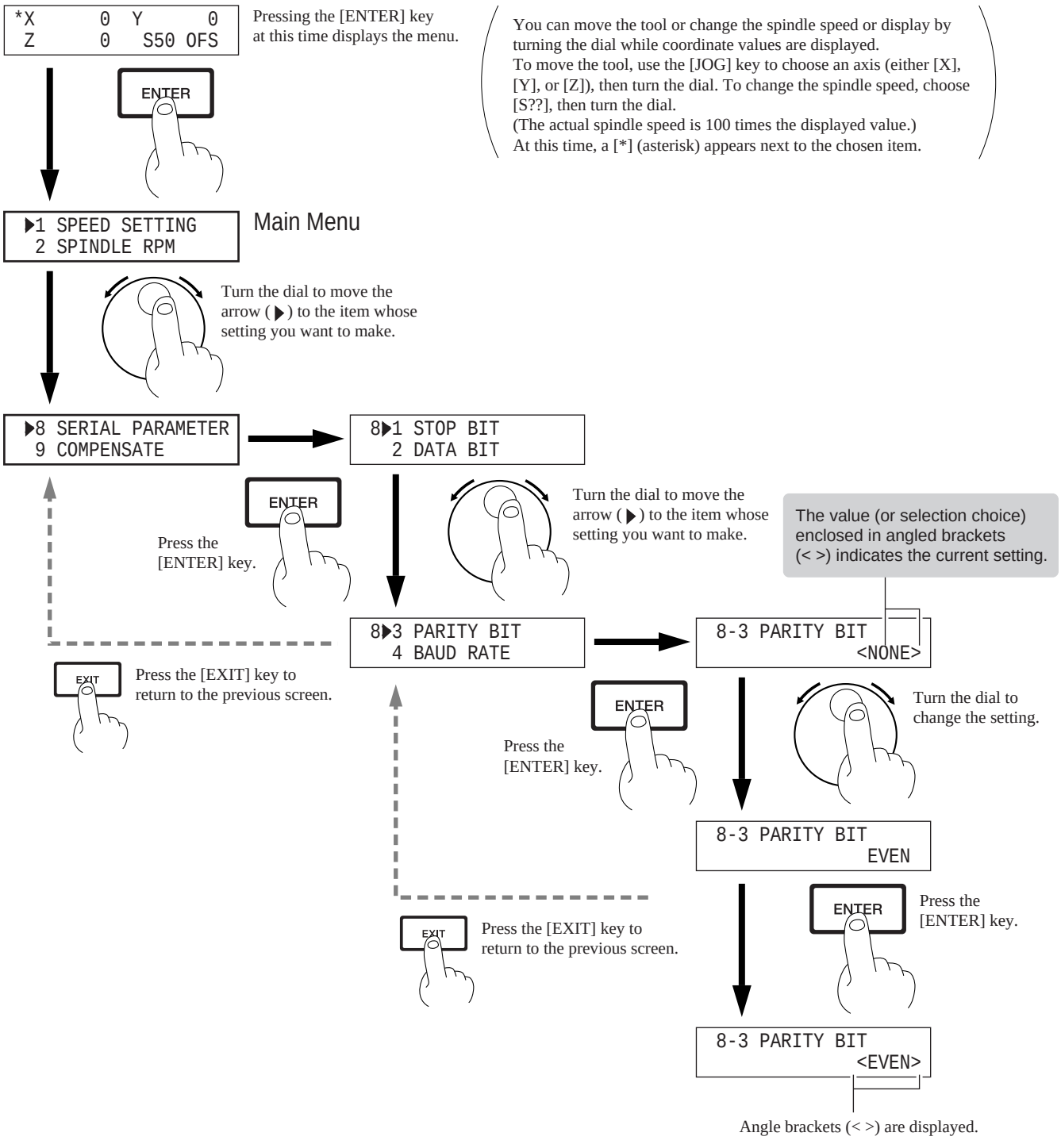


* To go back to the main menu, press the [EXIT] key several times.

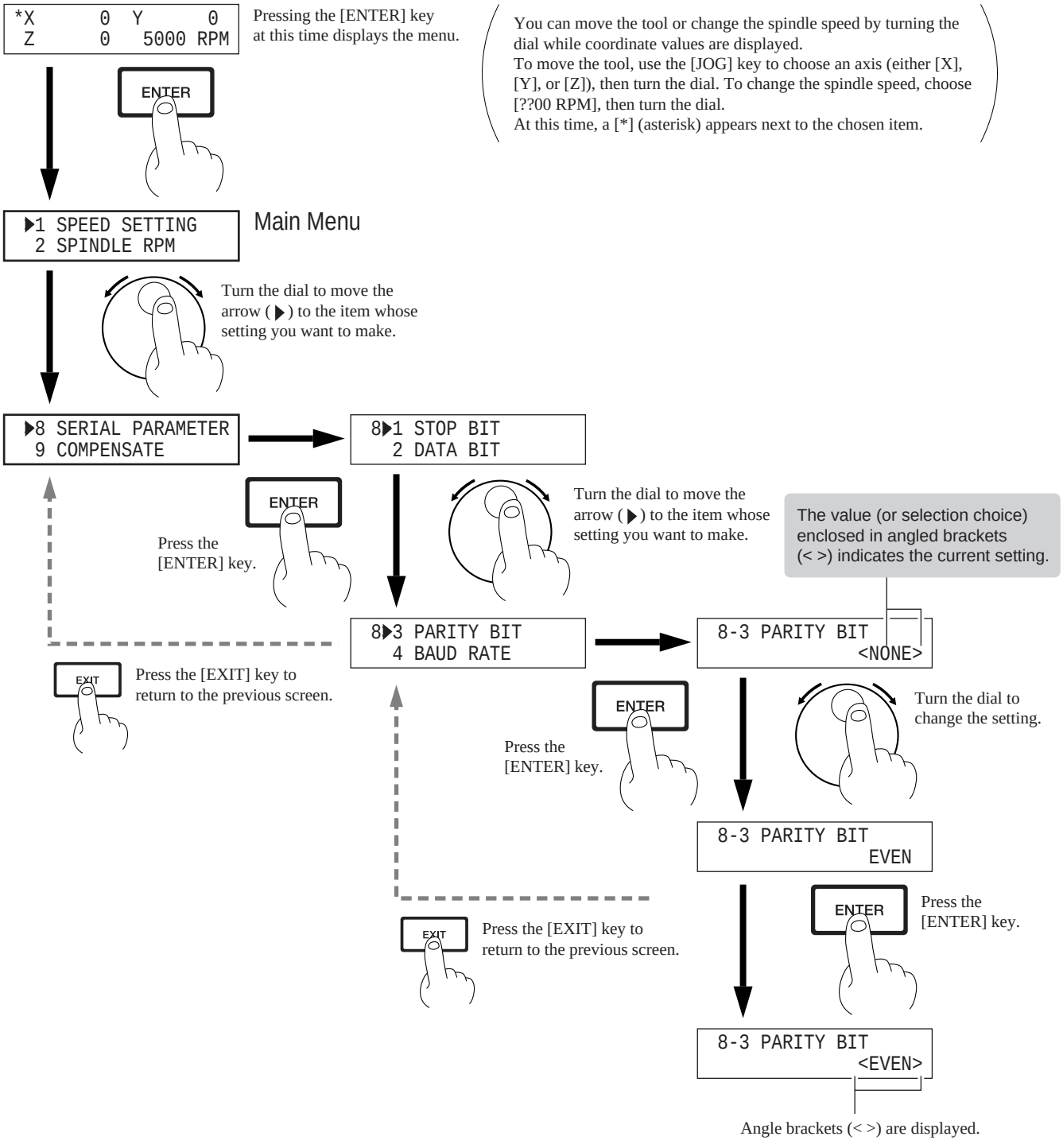
Part 1 Basic Operation

1-1 Making Settings Using the Liquid-crystal Display

MDX-650A/650



MDX-500


*X 0 Y 0
Z 0 5000 RPM
Pressing the [ENTER] key at this time displays the menu.
You can move the tool or change the spindle speed by turning the dial while coordinate values are displayed.
To move the tool, use the [JOG] key to choose an axis (either [X], [Y], or [Z]), then turn the dial. To change the spindle speed, choose [??00 RPM], then turn the dial.
At this time, a [*] (asterisk) appears next to the chosen item.
ENTER
▶1 SPEED SETTING
2 SPINDLE RPM
Main Menu
Turn the dial to move the arrow (▶) to the item whose setting you want to make.
▶8 SERIAL PARAMETER
9 COMPENSATE
8▶1 STOP BIT
2 DATA BIT
ENTER
Press the [ENTER] key.
Turn the dial to move the arrow (▶) to the item whose setting you want to make.
The value (or selection choice) enclosed in angled brackets (< >) indicates the current setting.
8▶3 PARITY BIT
4 BAUD RATE
8-3 PARITY BIT
<NONE>
EXIT
Press the [EXIT] key to return to the previous screen.
ENTER
Press the [ENTER] key.
Turn the dial to change the setting.
8-3 PARITY BIT
EVEN
ENTER
Press the [ENTER] key.
EXIT
Press the [EXIT] key to return to the previous screen.
8-3 PARITY BIT
<EVEN>
Angle brackets (< >) are displayed.

1-2 Setting the Connection Parameters

Connection with a parallel cable is called a “parallel connection,” and connection with a serial cable is called a “serial connection.” Make the appropriate settings on both the computer and the machine to configure the equipment for the type of connection that has been made. To make the settings on the computer, refer to the manual for the computer or the software in use.

1 Press the [EXIT] key to display the main menu.

2 Turn the dial to move the arrow to [CONNECTION], then press the [ENTER] key.

3 Turn the dial to choose [AUTO], then press the [ENTER] key.

4 Press the [EXIT] key once to return to the screen at right.

5 Turn the dial to move the arrow to [SERIAL PARAMETER], then press the [ENTER] key.

6 Turn the dial to move the arrow to the item you want, then press the [ENTER] key.

7 Turn the dial to choose a value (or selection), then press the [ENTER] key.

For serial connection only

1-3 Setting a Workpiece Coordinate System

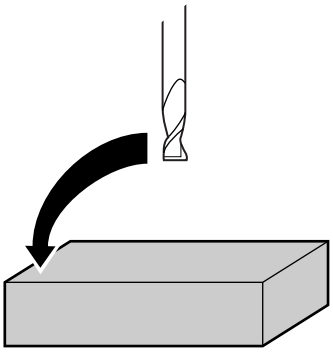
The workpiece coordinate system is for machining loaded cutting material. Set the origin point at where you want on the workpiece. The program assumes that the origin is set with the machine. Before making the setting, check the program. This section describes the following three cases.

- Workpiece coordinates not specified
- Specified with G54 through G59
- Specified with G92
- When specifying EXOFS (External Workpiece-origin Offset)

Workpiece Coordinates Not Specified

When workpiece coordinates are not specified in the program, the workpiece coordinates set with the machine are used. The substituted workpiece coordinate system is G54.
The G54 workpiece-coordinate origin point is set as follows. This explanation assumes that the origin point for the workpiece coordinate system is set at the front-left corner of the material

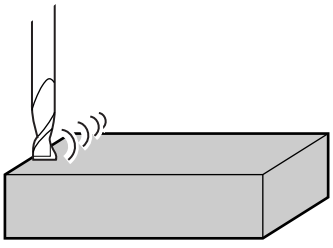
1 Press the arrow keys and the Tool up/down keys to move the cutting tool to a position close to the front left corner of the workpiece.



2 Use the [JOG] key to move the on-screen **[*]** to [X], [Y], or [Z].

```
*X      0  Y      0
 Z      0   .......
```

3 Rotate the dial to move the tool a little at a time.



4 Repeat steps 2 and 3 to align the tool at the front-left corner of the material to be machined.

5 Press the [XY/A] key (If you're using MDX-500, press [XY] key).
The screen at right appears.

```
>1 Set G54(XY)
 2 Set G55(XY)
```

6 Turn the dial to move the arrow to [SET G54(XY)], then press the [ENTER] key.
The origin point for the X and Y axes is set at the present tool position.

```
>1 Set G54(XY)
 2 Set G55(XY)
```

7 Press the [Z] key.
The screen at right appears.

```
>1 Set G54(Z)
 2 Set G55(Z)
```

8 Turn the dial to move the arrow to [SET G54(Z)], then press the [ENTER] key.

```
>1 Set G54(Z)
 2 Set G55(Z)
```

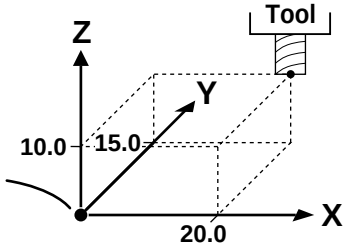
Specified with G54 Through G59

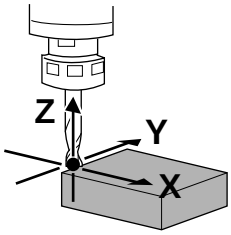
When the workpiece coordinates are specified with G54 through G59, all workpiece coordinates stated in the program are set with the machine.
For information about how to make the setting, see the previous section.
The previous section describes how to make the G54 setting, but you can also make the settings for other workpiece coordinates in the same way. The setting for EXOFS (external workpiece origin offset) is also made in the same way.
For a detailed description of G54 through G59, refer to the "NC Code Programmer's Manual."

Specified with G92

G92 sets the current tool location to a desired location on a workpiece coordinate system.
Normally, this is done by moving the tool to the location to become the workpiece-coordinate origin point, then executing G92X0Y0Z0.

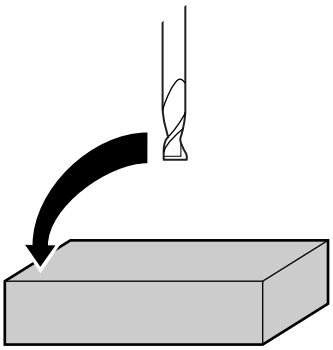
Executing **G92X20.0Y15.0Z10.0** sets the coordinate system taking this as the origin point.





Move the tool to the starting point assumed by the program before sending the data. This example assumes that the front-left corner of the workpiece is to be specified as the origin point for the workpiece coordinate system.

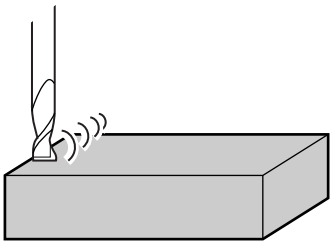
1 Press the arrow keys and the Tool up/down keys to move the cutting tool to a position close to the front left corner of the workpiece.



2 Use the [JOG] key to move the on-screen [*] to [X], [Y], or [Z].

```
*X      0  Y      0
 Z      0   .......
```

3 Rotate the dial to move the tool a little at a time.



4 Repeat steps 2 and 3 to align the tool at the front-left corner of the material to be machined.

When Specifying EXOFS (External Workpiece-origin Offset)

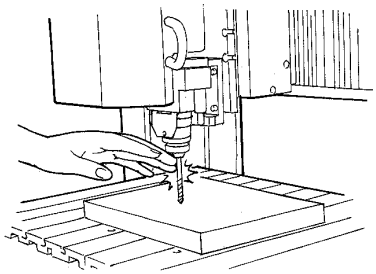
This is similar to the setting method for G54.
Turn the dial to move the arrow to [SET EXOFS], then press the [ENTER] key.

If you're using the MDX-650A/650, you can display the set coordinate values of the coordinate systems.
At the coordinate-view screen, press the [JOG] key and move the on-screen [*] to the [OFS, G54-59] at the lower right.
Turn the dial to choose the coordinate system to display, then press the [ENTER] key to enable the setting.

1-4 Downloading Cutting Data

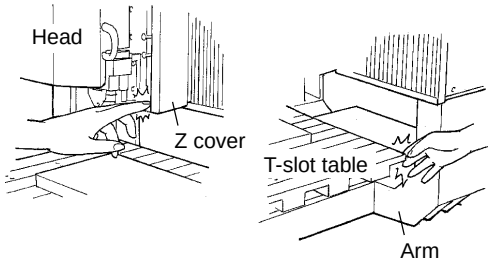
⚠CAUTION

Do not insert the fingers between the XY table and base or between the head and Z cover.
Doing so may result in injury.



Do not operate beyond capacity or subject the tool to undue force.
The tool may break or fly off in a random direction. If cutting beyond capacity is mistakenly started, immediately turn off the switch.

Do not insert the fingers between the T-slot table and arms or between the head and Z cover.
The fingers may be pinched, resulting in injury.



Before Sending the Data

Postprocessing

When a program has been created with general-use CAM software, it may need to be rewritten using NC codes that the cutting machine supports. This is called postprocessing.
Postprocessing eliminates code that the cutting machine does not support, and converts the program to one that the target machine can execute.
For more information about postprocessing, see the "Code Conversion Table" and "Program Statements" below.
For information on how to make settings, see the documentation for the CAM software you're using.

Code Conversion Table

Before conversion	After conversion	
Program Number (O)	O	
Address P for Subprogram Call (P)	P	
Subprogram Call (M98)	M98	
End of Subprogram (M99)	M99	
End of Program (M30)	M30	
Address D for Cutter Compensation (D)	D	
Address X (X)	X	
Address Y (Y)	Y	
Address Z (Z)	Z	
Address A (A)	A	*Available only when the optional rotary axis unit is installed

Program Statements (for Reference)

Enter the following codes at the beginning and end of the program (or subprogram).

```
%                                   Data start
G90G17G00Z?                         Set escape location which does not touch workpiece at [Z?]
X0Y0                                Move tool to XY origin
M03                                 Rotate spindle motor
•••••
  a program for cutting
•••••
G00Z?                               Set escape location which does not touch workpiece at [Z?]
X0Y0                                Return tool to XY origin
M05                                 Spindle rotation halt
M02                                 End of program
%                                   Data end
```

Caution

1 Use only codes listed in the "NC Code Programmer's Manual." Codes which are not listed there are ignored.
2 Codes for coolant control are not supported. However, on/off control of the EXT2 connector interlocked with spindle-rotation signals is possible. (For more information, refer to "Descriptions of the Display Menus.")
3 The setting for interpretation of the decimal point is made on the machine. Make the setting for the same interpretation assumed by the program. You can choose [NORMAL] or [CALCULATOR].

Selecting the Character Code

The machine supports ASCII, ISO, and EIA character codes.
Because the auto-recognition feature for character codes is set to "on" when shipped from the factory, so there is no need to select the character code.
However, if "Parity Error" is displayed while receiving cutting data, refer to "Descriptions of the Display Menus" and make the setting for the character code.
For more information about parity errors, see "Error Messages."

Using Subprograms

When programming using subprograms, you must register the subprograms with the machine before you send the main program. Register all subprograms called by the main program. If not registered, and error is displayed during cutting.
Subprograms are stored in a buffer (temporary memory), so they disappear when you switch off the power.
The method used to registering subprograms is as follows.

1 Turn the dial to move the arrow to [SUB-PROGRAM], then press the [ENTER] key.

```
>10 SUB-PROGRAM
 11 OTHERS
```

2 Turn the dial to move the arrow to [ENTRY SUB-PROG.], then press the [ENTER] key.

```
10>1 ENTRY SUB-PROG.
   2 SUB-PROG. SIZE
```

If a subprogram has already been registered the screen at right appears.
To register a new program, you must delete the program already registered. To delete it, press the [ENTER] key.

```
Sub-Program Clear!
Are You Sure?[ENTER]
```

3 The screen at right appears.

```
10-1 ENTRY SUB-PROG.
  Send Sub-Program
```

4 Operate the computer and send the subprograms. For more information on how to send data, take a look at the next section.
During program registration, the screen at right is displayed.

```
10-1 ENTRY SUB-PROG.
*Now Entry Sub-Prog.
```

(The [*] flashes while data is being received.)

5 When sending of one subprogram finishes, the screen changes to the one at right. To continue with sending the next subprogram, continue sending with the same setup, without pressing the [EXIT] key.
You cannot register an addition subprogram after the [EXIT] key has been pressed. Adding will require re-registering all subprograms, including the one to be added.

After finishing sending all the subprograms needed to execute the program, press the [EXIT] key.

```
10-1 ENTRY SUB-PROG.
 Finish! Push [EXIT]
```

The data space for saving programs, including the main program and any subprograms, is 2 MB (2,048 kB).
The subprogram data area can be varied within a range of 0 to 1,536 kB, and the amount of space remaining after subtracting the subprogram area from the total space is the amount of space for the main program. (For example, if the subprogram space is set at 1,536 kB, the main program space is 512 kB.)

To perform engraving using subprograms, you must register all subprograms. If there is not enough data space to register all subprograms, refer to the next page to free up more data space.
However, even if the main program exceeds the remaining data space, you can still perform engraving for the sent data. Note that in this case, however, you cannot repeat engraving by pressing the [COPY] key.

■ If There Is Not Enough Data Space to Register a Subprogram

If the error at right appears while send data, it means that there is not enough data space, and a subprogram could not be stored. Stop sending data, and press the [EXIT] key. Data sent when the space was exceeded is all lost without being saved.
To increase the amount of data space for storing subprograms, follow the steps below.

```
10-1 ENTRY SUB-PROG.
 Entry Area Nothing
```

1 Turn the dial to move the arrow to [SUB-PROGRAM], then press the [ENTER] key.

```
>10 SUB-PROGRAM
 11 OTHERS
```

2 Turn the dial to move the arrow to [SUB-PROG. SIZE], then press the [ENTER] key.

```
10>2 SUB-PROG. SIZE
   3 To Main MENU
```

3 The screen at right appears. Press the [ENTER] key.

```
All Buffer Clear!
Are You Sure?[ENTER]
```

4 Turn the dial to change the data space, then press the [ENTER] key.

```
10-2 SUB-PROG. SIZE
        < 512 KByte>
```

Sending Cutting Data

Using a Communications Application for NC

When using a communications application for NC programs to send data to the machine, make the settings as shown below. Communication programs that support Protocol A cannot be used. Use a program that supports Protocol B and also permits communication with RS and CD signals. (DC codes are not supported.)

Character code	Set this to match the setting on the machine.
Communication parameters	Set these to match the settings on the machine. The factory-default settings are bit rate of 9,600, no parity bit, data length of 8 bits, and 1 stop bit.
DC code	Not supported (including XON/XOFF). Hardware control with RS and CS signals is performed.
TV check	Not supported.
Send buffer size	Any setting is acceptable.
Delimiter code	[LF] for ISO and ASCII and [CR] for EIA

Using the MS-DOS Prompt in Windows

Use CAM software, a text editor, or the like to create the program, then save it with "test.nc" as the file name. In this case, you can send the program to the cutting machine by entering the following at the MS-DOS command line. Because the format differs according to the computer, refer to the computer's documentation for more information.

Serial connection	C:\> `copy test.nc aux`
Parallel connection	C:\> `copy test.nc prn`

* When using a serial connection, set the same communication parameters for the computer and the machine. Make the settings for the communication parameters as shown below.

1 Start [MS-DOS Prompt] or [Command Prompt] in windows.
2 If the communication parameters for the COM1 port are as follows...

Bit rate:	9,600
Parity bit:	None
Data bit length:	8
Stop bit:	1

...then type in the following.

C:\WINDOWS> `mode com1 baud=9600 parity=n data=8 stop=1`

Waiting for Data During Serial Connection

During a serial connection, when sending data that contains many fine line segments, operation may stop while cutting is in progress. At such times, the machine is waiting for the next portion of cutting data to be sent.
You can avoid such interruptions by changing to a parallel connection to increase the data-transmission speed, or by increasing the amount of data stored in memory through such methods as sending cutting data for finishing while performing rough cutting.

What You Can Do During Cutting

Adjusting the Rotating Speed and Feed Speed

The programmed rotating speed and feed speed can be adjusted while cutting is in progress. Below is a list of the items that can be adjusted.

Positioning override	This sets the G00 operating speed, with the maximum speed taken to be 100%.
Cutting override	This specifies a percentage of the feed rate set by programming (F code). According to this setting, the feed rate is set to a percentage of all feed rates specified by F codes.
Cutting speed	This specifies the feed rate for cutting in “mm/min.” units. If an F code is specified after restarting cutting, the speed specified by the F code is used.
Spindle speed	This sets the speed of the spindle. If an S code is specified after restarting cutting, the speed specified by the S code is used.

1 While operation is in progress, press the [PAUSE] key. Movement of the tool and table stops. Note that because this is not an emergency stop, movement may continue for two or three seconds before stopping.
The screen at right appears.

```
PAUSE>CONTINUE
      STOP
```

2 Turn the dial to move the arrow to the desired item, then press the [ENTER] key.

```
PAUSE>CUT OVER RIDE
      CUT SPEED
```

3 Turn the dial to change the value, then press the [ENTER] key.
To change another item, press the [EXIT] key, then repeat steps 2 and 3.

```
PAUSE:CUT OVER RIDE
             <100 %>
```

4 Press the [EXIT] key to return to the screen at right.

```
PAUSE>CONTINUE
      STOP
```

5 Turn the dial to move the arrow to [CONTINUE], then press the [ENTER] key.
The paused state is canceled and cutting resumes.

Terminating Cutting

If you wish to correct the program and restart cutting from the beginning, or if the cutting data was different from what was desired, carry out the procedure below.

1 While operation is in progress, press the [PAUSE] key. Movement of the tool and table stops. Note that because this is not an emergency stop, movement may continue for two or three seconds before stopping.
The screen at right appears.

2 Stop sending data from the computer

3 Turn the dial to move the arrow to [STOP], then press the [ENTER] key.
Stop execution of the program.

1-5 Finishing

⚠CAUTION

Do not touch the tip of the blade with your fingers.
Doing so may result in injury.

Please use a vacuum cleaner to remove cutting dust.
Do not use any blower like airbrush.
Otherwise, dust spread in the air may harm your health.

Do not touch the tool immediately after cutting operating stops.
The tool may have become hot due to friction heat and may cause burns if touched.

Use a commercially available brush to remove metal cuttings.
Attempting to use a vacuum cleaner to take up metal cuttings may cause fire in the vacuum cleaner.

After cutting has been finished, detach the tool, remove the material, and clean away chips.

1 Press the [EXIT] key to display the main menu.

```
>1 SPEED OVER RIDE
 2 SPINDLE RPM
```

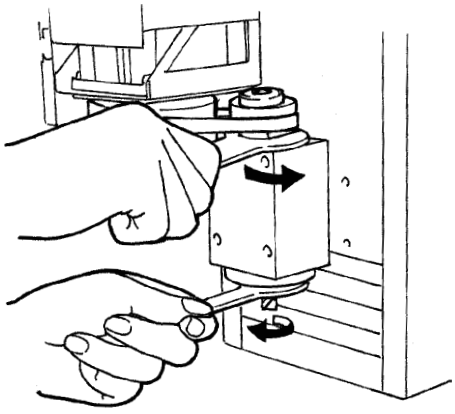
2 Turn the dial to move the arrow to [MECHA MOVING], then press the [ENTER] key.

```
>3 MECHA MOVING
 4 TOOL DIAMETER
```

3 Turn the dial to move the arrow to [Go LIMIT Pos.], then press the [ENTER] key.

```
3>11 Go LIMIT Pos.
  12 To Main MENU
```

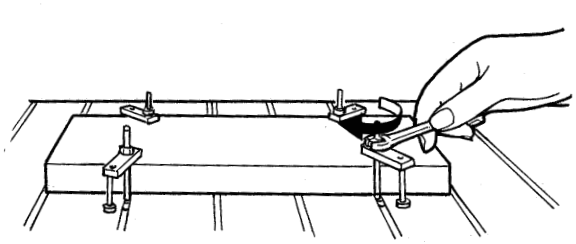
4 Detach the tool.



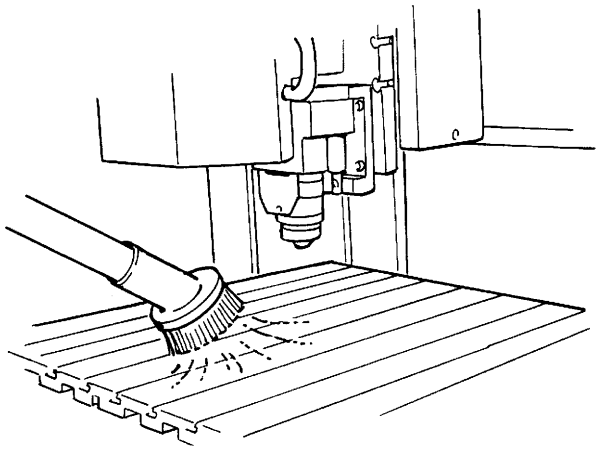
5 Turn the dial to move the arrow to [Go VIEW Pos.], then press the [ENTER] key.

```
3 >3 Go VIEW Pos.
   4 Go G54(XY)
```

6 Remove the material.



7 Use a commercially available vacuum cleaner to remove chips.



Part 2 User's Reference

2-1 Cutting Area

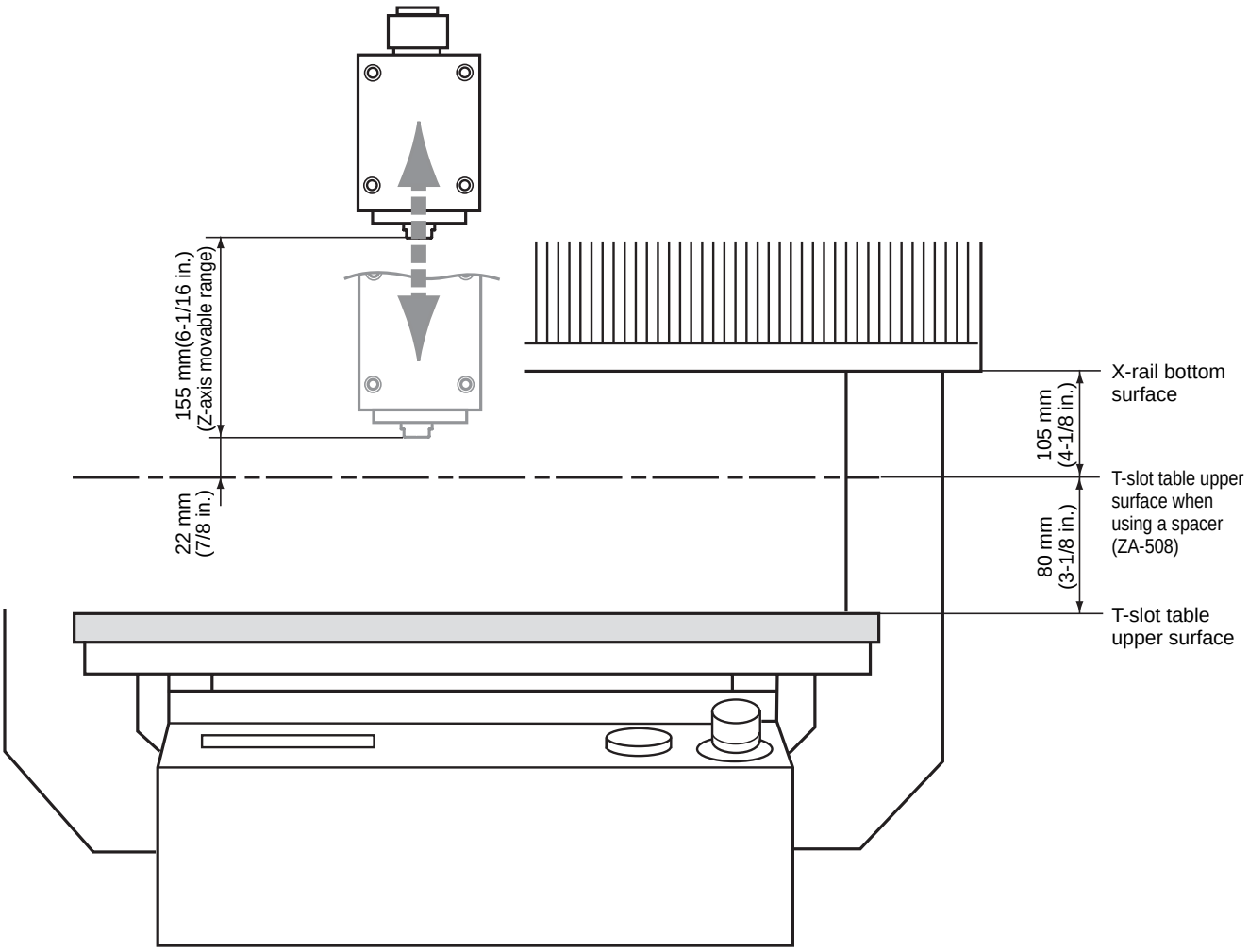
MDX-650A/650

The maximum cutting area of the MDX-650A/650 is 650 mm x 450 mm x 155 mm (25-9/16 in. x 17-11/16 in. x 6-1/16 in.).
The actual cutting area differs according to the type of spindle installed.

When Use the High-torque Spindle (ZS-650T)

When a high-torque spindle (ZS-650T) is installed, the range that you can actually cut (in the height direction) is subject to the following restrictions and is smaller than the maximum cutting range described earlier.

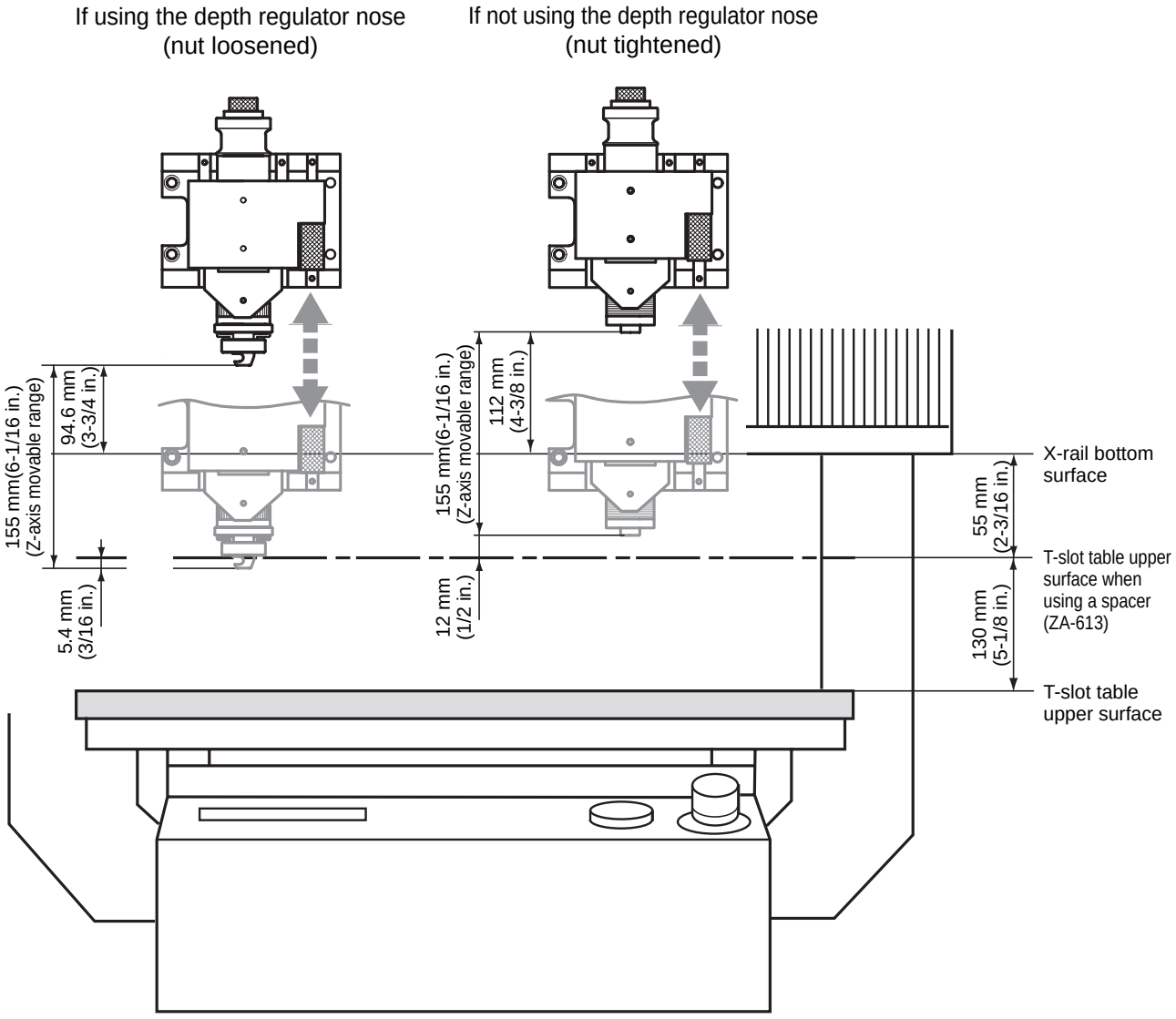
- Length of the installed tool
- Position of the XY table where the workpiece to cut is loaded
- If using a spacer for the T-slot table (ZA-600/500 series), the height of the spacer



When Use the High-speed Spindle

When a high-speed spindle is installed, the range that you can actually cut (in the height direction) is subject to the following restrictions and is smaller than the maximum cutting range.

- Length of the installed tool
- Position of the XY table where the workpiece to cut is loaded
- If using a spacer for the T-slot table (ZA-600/500 series), the height of the spacer
- If using a depth regulator nose, the stroke of the spindle due to the nut (approx. 5 mm)



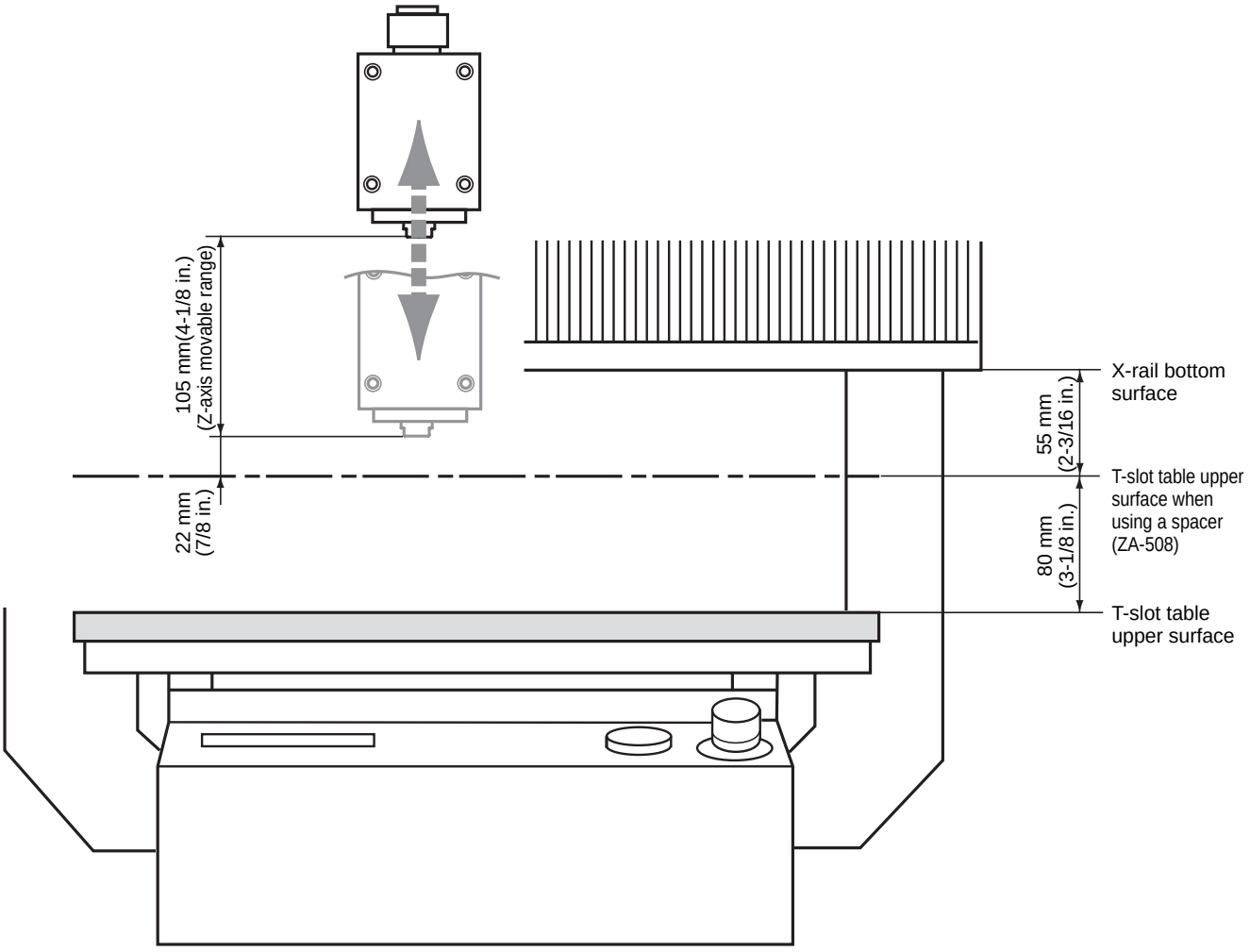
MDX-500

The maximum cutting area of the MDX-500 is 500 mm x 330 mm x 105 mm (19-5/8 in. x 12-15/16 in. x 4-1/8 in.).
The actual cutting area differs according to the type of spindle installed.

When Use the High-torque Spindle (ZS-500T)

When a high-torque spindle (ZS-500T) is installed, the range that you can actually cut (in the height direction) is subject to the following restrictions and is smaller than the maximum cutting range described earlier.

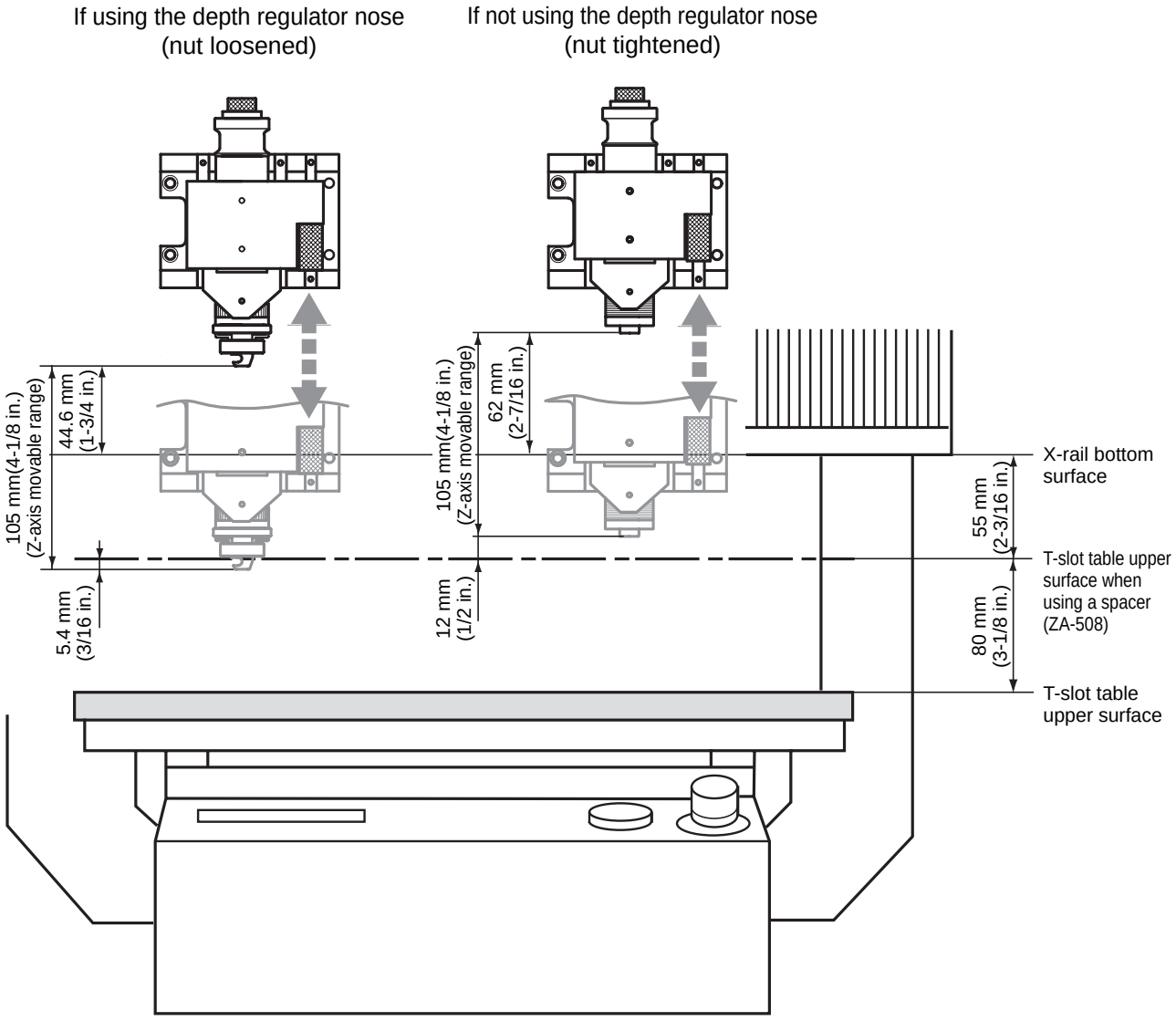
- Length of the installed tool
- Position of the XY table where the workpiece to cut is loaded
- If using a spacer for the T-slot table (ZA-600/500 series), the height of the spacer



When Use the High-speed Spindle

When a high-speed spindle is installed, the range that you can actually cut (in the height direction) is subject to the following restrictions and is smaller than the maximum cutting range.

- Length of the installed tool
- Position of the XY table where the workpiece to cut is loaded
- If using a spacer for the T-slot table (ZA-600/500 series), the height of the spacer
- If using a depth regulator nose, the stroke of the spindle due to the nut (approx. 5 mm)

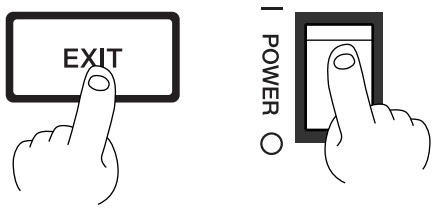


2-2 Operating Each Function

Changing to Japanese Messages on the Liquid-crystal Display

You can choose either English or Japanese for the display language.

1 Switch on the power while holding down the [EXIT] key.



2 Turn the dial to move the arrow to [JAPANESE], then press the [ENTER] key.

```
>2 JAPANESE
                -<END>-
```

3 Messages on the display now appear in Japanese.

```
ENTERｷｰ ﾖ    >NC-CODE
   ｵｼﾃｸﾀﾞｻｲ  -<ｵﾜﾘ>-
```

* To return the display to English-language messages, carry out Step 1 again. When the language-selection menu appears (similar to the one in Step 1, but in Japanese), move the arrow to “English” and press the [ENTER] key.

Performing Repeat Cutting

The data buffer is the place where data received from the computer is stored temporarily. (The data in the data buffer can be erased by switching off the power or clearing the data.
Pressing the [COPY] key calls up the all cutting data stored in the data buffer and executes the replotting procedure. When you perform replotting, clear the data from the data buffer before sending the cutting for replotting from the computer.

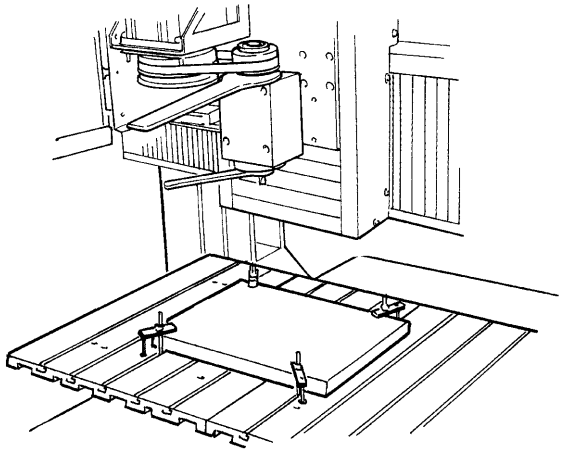
1 Press the [COPY] key.
The screen at right appears.

```
>1 COPY START
 2 CLEAR COPY BUFFER
```

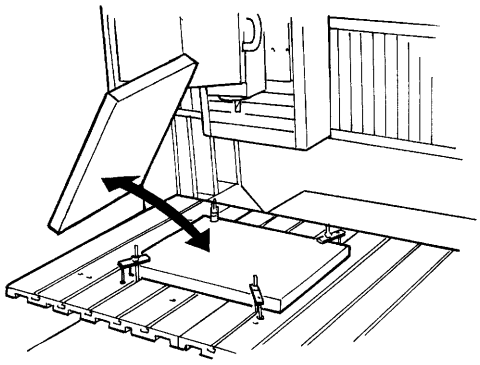
2 Turn the dial to move the arrow to [CLEAR COPY BUFFER], then press the [ENTER] key.
Cutting data in the data buffer is lost.

```
>2 CLEAR COPY BUFFER
                -<END>-
```

3 Install the tool (blade) and load the material. Use the software to send the cutting data.



4 After cutting has finished, remove the cut material and load a new piece. Set the origin point if necessary.



5 Press the [COPY] key.
Turn the dial to move the arrow to [COPY START], then press the [ENTER] key.

```
>1 COPY START
 2 CLEAR COPY BUFFER
```

Pausing Operation and Adjusting the Rotating Speed and Feed Speed

The programmed rotating speed and feed speed can be adjusted while cutting is in progress. Below is a list of the items that can be adjusted.

Positioning override	This sets the G00 operating speed, with the maximum speed taken to be 100%.
Cutting override	This specifies a percentage of the feed rate set by programming (F code). According to this setting, the feed rate is set to a percentage of all feed rates specified by F codes.
Cutting speed	This specifies the feed rate for cutting in “mm/min.” units. If an F code is specified after restarting cutting, the speed specified by the F code is used.
Spindle speed	This sets the speed of the spindle. If an S code is specified after restarting cutting, the speed specified by the S code is used.

1 While operation is in progress, press the [PAUSE] key. Movement of the tool and table stops. Note that because this is not an emergency stop, movement may continue for two or three seconds before stopping.
The screen at right appears.

```
PAUSE>CONTINUE
      STOP
```

2 Turn the dial to move the arrow to the desired item, then press the [ENTER] key.

```
PAUSE>CUT OVER RIDE
      CUT SPEED
```

3 Turn the dial to change the value, then press the [ENTER] key.
To change another item, three the [EXIT] key, then repeat steps 2 and 3.

```
PAUSE:CUT OVER RIDE
             <100 %>
```

4 Press the [EXIT] key to return to the screen at right.

```
PAUSE>CONTINUE
      STOP
```

5 Turn the dial to move the arrow to [CONTINUE], then press the [ENTER] key.
The paused state is canceled and cutting resumes.

Terminating Cutting

If you wish to correct the program and restart cutting from the beginning, or if the cutting data was different from what was desired, carry out the procedure below.

1 While operation is in progress, press the [PAUSE] key. Movement of the tool and table stops. Note that because this is not an emergency stop, movement may continue for two or three seconds before stopping.
The screen at right appears.

```
PAUSE>CONTINUE
      STOP
```

2 Stop sending data from the computer

3 Turn the dial to move the arrow to [STOP], then press the [ENTER] key.
Stop execution of the program.

```
PAUSE>STOP
      CUT OVER RIDE
```

2-3 Descriptions of the Display Menus

1 SPEED OVER RIDE

1-1 CUT OVER RIDE

```
1-1 CUT OVER RIDE
            <100 %>
```

Stored in Memory
Yes

Factory Default
100

Setting Range
0 to 200

Steps
1

Description

This adjusts the cutting speed on the machine. This setting is made as a percentage, with the feed rate specified by program F-codes taken to be 100%. All codes that function at the cutting speed are affected.
This setting can be used to make fine adjustments in the feed rate without altering programming.
Note, however, that operation outside the machine's maximum or minimum speed is not possible, regardless of this setting. Also note that the feed rate changes in units of 60 mm/min. (1 mm/sec.). (However, the minimum speed can be set at 30 mm/min.)
This setting can be made even while operation is paused.

1-2 CUT SPEED

```
1-2 CUT SPEED
      < 120 mm/min>
```

Stored in Memory
Yes

Factory Default
120

Setting Range
30, 60 to 5100

Steps
60

Description

This sets the default value for cutting speed. When an operation command using cutting speed is input when no cutting speed has been specified using F-codes, operation is carried out at this cutting speed.
This setting can be made even while operation is paused. Cutting restarts at the speed that has been set, but if an F-code speed setting is encountered, the speed then changes to the value of the F-code setting.

1-3 MOVE OVER RIDE

```
1-3 MOVE OVER RIDE
            <100 %>
```

Stored in Memory
Yes

Factory Default
100

Setting Range
0 to 100

Steps
1

Description

This sets a limit for positioning speed. When shipped from the factory, the machine is set at maximum speed. This setting is made as a percentage, with maximum speed taken to be 100%.
This setting is applied to all codes which function at the positioning speed: G00, G80, G81, G82, G85, G86, and G89 are affected.
This limitation of positioning speed causes cutting times to become longer. Positioning speed is not restricted by the limit speed setting for cutting speed.
This setting can be made even while operation is paused.

2 SPINDLE RPM

```
 2 SPINDLE RPM
        < 5000 RPM>
```

Stored in Memory
Yes

Factory Default
3000
(High Torque)

5000
(High Speed)

Setting Range
3000 to 12000
(High Torque)

5000 to 20000
(High Speed)

Steps
1

Description

The speed of rotation specified by this setting is used when a spindle rotation command (M03) is input in a state where rotation has not been specified by an S-code.
This setting can also be made while operation is paused. Cutting restarts at the speed that has been set, but if an S-code speed setting is encountered, the speed then changes to the value of the S-code setting.

3 MECHA MOVING

```
3 >1 Go EXOFS(XY)
  2 Go EZOFS(Z)
```

```
3 17 Go LIMIT Pos.
  18 To Main MENU
```

Description

This performs movement to the specified location at the machine's highest speed. In the case of [Go EXOFS (Z)], however, the speed is as follows.

(1) The speed specified by an F code in the program and by the machine's [CUT OVER RIDE] setting
(2) The speed specified by the machine's [CUT SPEED] and [CUT OVER RIDE] settings

When both of these have been set, the one set last is valid.

4 TOOL DIAMETER

```
4 TOOL No.1
         <    0 um>
```

```
4 TOOL No.2
         <    0 um>
```

```
4 TOOL No.3
         <    0 um>
```

Stored in Memory
Yes

Factory Default
0

Setting Range
0 to 10000

Steps
10

Description

This specifies the amount of offset set for cutter compensation and tool-length compensation.
When G41 or G42 (cutter compensation) specifies an offset number which has not been set with G10, the value set on the machine is used.

5 MOVING MODE

5-1 SPINDLE CONTROL

```
5-1 SPINDLE CONTROL
                <ON>
```

Stored in Memory
Yes

Factory Default
ON

Selection Choices
ON, OFF,
EXTERNAL
ONLY

Description

This chooses the method of control for the spindle motor.

ON	When a command to rotate the spindle is received, a rotate signal is issued to the internal spindle. A signal is also issued to the EXT2 connector at the same time.
OFF	Even when a command to rotate the spindle is received, not rotate signal is issued to the internal spindle circuit. Similarly, no signal is issued to the EXT2 connector.
EXTERNAL ONLY	Even when a command to rotate the spindle is received, not rotate signal is issued to the internal spindle circuit. A signal is issued only to the EXT2 connector.

5-2 CALC.TYPE VALUE

```
5-2 CALC.TYPE VALUE
            <NOT USE>
```

Stored in Memory
Yes

Factory Default
NOT USE

Selection Choices
NOT USE,
F & IJKRXYZ CODE,
F CODE,
IJKRXYZ CODE

Description

This selects how numeric values are interpreted during real-number entry or integer entry.
When set to [NOT USE], values during real-number entry are interpreted as millimeters or inches, and values during integer entry are interpreted as minimum-unit millimeters or inches.
When set to [F & IJKRXYZ CODE], [F CODE], or [IJKRXYZ CODE], numerical values are always interpreted as millimeters or inches regardless of whether real-number entry or integer entry has been selected for the respective code.

5-3 SINGLE BLOCK

```
5-3 SINGLE BLOCK
                <OFF>
```

Stored in Memory
Yes

Factory Default
OFF

Selection Choices
OFF, ON

Description

When [ON] is specified, the program executes one block and then goes into a standby state. Pressing the [ENTER] key causes the next block to be executed.
This makes it possible to carry out operations while change the contents of the program at each step.
During single-block execution, it is not possible to switch the single block on or off.

5-4 OP. BLOCK SKIP

```
5-4 OP. BLOCK SKIP
                <ON>
```

Stored in Memory
Yes

Factory Default
ON

Selection Choices
ON, OFF

Description

Optional block skip is a function that skips over desired blocks in a program (see the NC Code Programmer's Manual).
When set to [OFF], no blocks specified by optional block skip are skipped.

5-5 OFFSET TYPE

```
5-5 OFFSET TYPE
          <TYPE A>
```

Stored in Memory
Yes

Factory Default
TYPE A

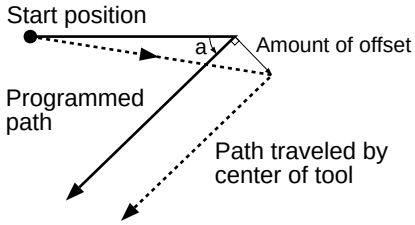
Selection Choices
TYPE A, TYPE B

Description

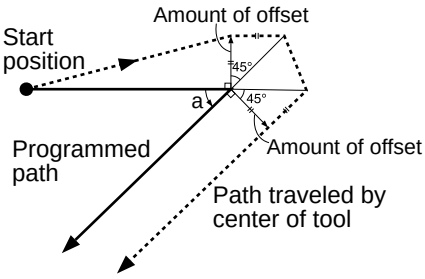
This selects the program path for outer-side travel when starting or ending cutter compensation.
A case of shifting from a line to another line with travel on the outer side of an acute angle when cutter compensation starts is given as an example. For more information, refer to the NC Code Programmer's Manual.

Outer-side Acute Angle **(a < 90°)**

From a line to a line -- Type A



From a line to a line -- Type B



5-6 PROGRAM NUMBER

```
5-6 PROGRAM NUMBER
          <4 DIGITS>
```

Stored in Memory
Yes

Factory Default
4

Selection Choices
4, 8

Description

This chooses the number of digits for program numbers and sequence numbers.
Choose the number of digits employed in the program. If the number of digits specified by the program differs from the number of digits specified by the machine, unexpected operation or errors may occur. If you are using subprograms, all programs used must employ the same number of digits. Programs that specify different numbers of digits must not be run at the same time.
For more information about how to specify program numbers and sequence numbers, see the entries for M98, O, and N in the NC Code Programmer's Manual.

5-7 ACCELERATION

```
5-7 ACCELERATION
              <0.3G>
```

Stored in Memory
Yes

Factory Default
0.3

Selection Choices
0.05, 0.1, 0.3

Description

This chooses the acceleration when moving the tool and table. Normally the default value (0.3 G) can be left unchanged. When cutting material that creates a high load, on rare occasions the acceleration may make it impossible to perform cutting. In such cases, the value should be changed.

6 USE CODE

```
6 USE CODE
              <AUTO>
```

Stored in Memory
Yes

Factory Default
AUTO

Selection Choices
ASCII, ISO, EIA, AUTO

Description

This selects the character code system of the program to be received. When set to [AUTO], the character code (ASCII, ISO, or EIA) is determined automatically. Normally this should be left set to [AUTO].
If the character code cannot easily be determined when set to [AUTO], the setting should be made manually. Receiving a program when the character code setting is incorrect may result in unexpected operation.

7 CONNECTION

```
7 CONNECTION
              <AUTO>
```

Stored in Memory
Yes

Factory Default
AUTO

Selection Choices
AUTO, SERIAL, PARALLEL

Description

This sets the type of interface used for connection to the computer. When set to [AUTO], the port is determined automatically. The communication parameters in effect when a serial connection is used are according to the parameters of the panel settings.
If the interface cannot easily be determined when set to [AUTO], the setting should be made manually.

8 SERIAL PARAMETER

8-1 STOP BIT

```
8-1 STOP BIT
                <1>
```

Stored in Memory
Yes

Factory Default
1

Selection Choices
1, 2

Description

This sets the number of stop bits for the communication parameters.

8-2 DATA BIT

```
8-2 DATA BIT
                <8>
```

Stored in Memory
Yes

Factory Default
8

Selection Choices
7, 8

Description

This sets the data bit length for the communication parameters.

8-3 PARITY BIT

```
8-3 PARITY BIT
              <NONE>
```

Stored in Memory
Yes

Factory Default
NONE

Selection Choices
NONE, ODD, EVEN

Description

This makes the setting for parity checking for the communication parameters.

8-4 BAUD RATE

```
8-4 BAUD RATE
            <9600>
```

Stored in Memory
Yes

Factory Default
9600

Selection Choices
4800, 9600,
19200, 38400

Description

This sets the bit rate (transmission speed) for the communication parameters.

8-5 HAND SHAKE

```
8-5 HAND SHAKE
        <HARD-WIRE>
```

Stored in Memory
Yes

Factory Default
HARD-WIRE

Selection Choices
HARD-WIRE,
XON/XOFF

Description

This sets the type of hand shake for the communication parameters.

9 COMPENSATE

```
9-1 X-COMPENSATE
        <100.00 %>
```

```
9-2 Y-COMPENSATE
        <100.00 %>
```

```
9-3 Z-COMPENSATE
        <100.00 %>
```

Stored in Memory
Yes

Factory Default
100

Setting Range
99.90 to 100.10

Steps
0.01

Description

This compensates for differences between the length specified by the program and the actual cutting length. This can correct for error due to temperature or humidity, as well as error due to individual differences from one machine to another.
If you change the compensation value, then switch the power off and back on. The changed compensation value is enabled after the power has been reset.

10 SUB-PROGRAM

10-1 ENTRY SUB-PROG.

```
10-1 ENTRY SUB-PROG.
  Send Sub-Program
```

Description

This registers a subprogram.
When programming using subprograms, you must register the subprograms with the machine before you send the main program. Register all subprograms called by the main program. If not registered, and error is displayed during cutting.
Subprograms are stored in a buffer (temporary memory) , so they disappear when you switch off the power.
For more information about subprograms, refer to the separate "NC Code Programmer's Manual."

10-2 SUB-PROG. SIZE

```
10-2 SUB-PROG. SIZE
        < 512 KByte>
```

Stored in Memory
Yes

Factory Default
512

Setting Range
0 to 1536

Steps
1

Description

This determines the amount of data space for registering subprograms. If an error message about insufficient data space is displayed during registration, increase the amount of space.

11 OTHERS

11-1 SENSOR MODE

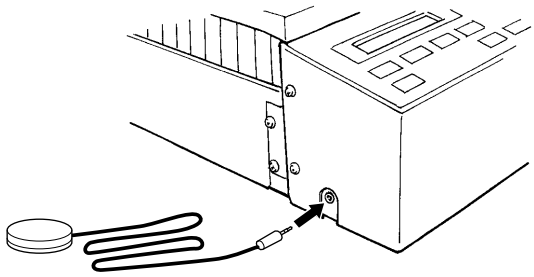
```
11-1 SENSOR MODE
 Please Cursor Move
```

Description

This sets the amount of Z-axis shift for the EXOFS (external workpiece origin point amount of offset) using the Z0 sensor. Changing the amount of shift causes the setting for [EXOFS Z] to change.

Connect the Z0 sensor before entering [SENSOR MODE]. An error is generated if the cord is loose or the Z0 sensor is not connected.

NOTICE Do not connect the Z0 sensor to the EXT2 connector. Doing so may damage the sensor.



Turn the dial to move the arrow to [OTHERS], then press the [ENTER] key.

```
>11 OTHERS
 12 To Coordinate
```

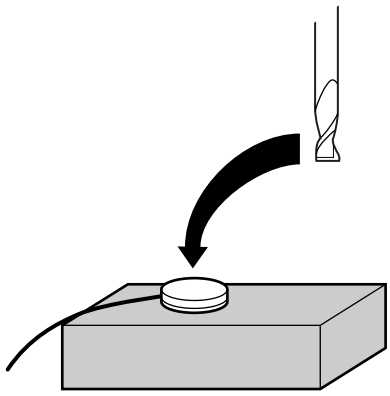
Turn the dial to move the arrow to [SENSOR MODE], then press the [ENTER] key.

```
11>1 SENSOR MODE
   2 SPINDLE UNIT
```

The following screen appears.

```
11-1 SENSOR MODE
 Please Cursor Move
```

Position the Z0 sensor at the area where the amount of Z-axis shift for the EXOFS is to be set, and move the tool until the tip of the tool touches the Z0 sensor.



When the tool touches the sensor, the following message appears on the display and the amount of shift is set.
After making the setting, the tool rises automatically.

```
11-1 SENSOR MODE
      SET Z ORIGIN!
```

Wait for the tool to stop, then detach the Z0 sensor.

11-2 SPINDLE UNIT

```
11-2 SPINDLE UNIT
       <HIGH TORQUE>
```

Stored in Memory
Yes

Factory Default
HIGH TORQUE

Selection Choices
HIGH TORQUE,
HIGH SPEED

Description

This sets the type of the installed spindle.
If a high-torque spindle is installed, choose [HIGH TORQUE]. If a high-speed spindle is installed, choose [HIGH SPEED].

11-3 BUZZER

```
11-3 BUZZER
                 <ON>
```

Stored in Memory
Yes

Factory Default
ON

Selection Choices
ON, OFF

Description

This switches on or switches off the confirmation sound heard when a control key is pressed.

11-4 SENSOR HEIGHT

```
11-4 SENSOR HEIGHT
          <15000 um>
```

Stored in Memory
Yes

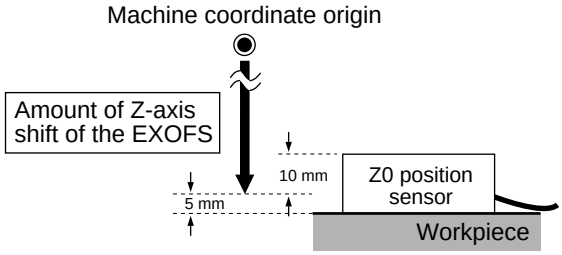
Factory Default*
15000

Setting Range
0 to 30000

Steps
10

Description

This sets the thickness of the Z0 sensor. The thickness of the Z0 sensor is 15.00 mm.
The location where the tool has descended from the surface location of the Z0 sensor by a distance equal to the set thickness is set to the origin of Z-axis.
The origin of Z-axis can be set to a location which is a fixed distance from the location where the Z0 sensor is placed. For example, to place the Z0 sensor on the surface of the material and set the amount of shift for the Z axis at a location 5 mm from the surface, the thickness should be set to a value of [10000].



The thickness of the Z0 sensor may vary slightly due to conditions of temperature and humidity. The sensor thickness can also be used to make fine adjustments when high accuracy is required.
Because cutting depth is affected, an incorrect setting may result in breakage of the tool.

***** The value that is set when shipped from the factory varies from one unit to another.

11-5 REVOLUTION TIME

Description

This displays the total rotation time of the spindle. The rotation time cannot be returned to [0] (zero).
The displayed time should be used as an indicator for performing periodic maintenance. (For more information about maintenance, refer to the separate "Setup & Maintenance.")

12 Rotary Control (MDX-650A/650 only)

```
12 ROTARY CONTROL
```

Description

This menu item is available when you are employ the optional rotary axis unit. If the rotary axis unit is not installed, you cannot enter this menu.

13 ATC (MDX-650A only)

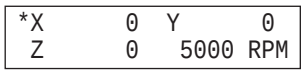
```
13 ATC
```

Description

This menu item is available when you are employ the optional ATC unit. If the ATC unit is not installed, you cannot enter this menu.

14 To Coordinate

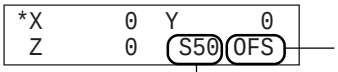
< MDX-500 >



Description

This displays the current tool location and spindle speed.
The units of measurement are in machine steps (1 step = 0.01 mm).

< MDX-650A/650 >



Current coordinate system

Current spindle speed
(The actual spindle speed is 100 times the displayed value.)

2-4 What to Do If...

When the machine does not work...

Is operation paused?	Cancel the paused state.
Is the power switched on?	Make sure the machine is powered up.
Is single-block operation on?	When single-block operation is on, the system executes one block, then stops. To execute the next block, press the [ENTER] key.
Is RML-1 chosen as the instruction set?	Set the instruction set to NC code. For information on how to make the setting, refer "Introduction" in this document.
Does the program use any unsupported codes?	Unsupported codes are ignored. For information about NC codes that the machine supports, refer to the separate "NC Code Programmer's Manual." For information about postprocessing when creating programs with general-use CAM software, refer to "Downloading Cutting Data - Postprocessing" in this document.
Is the setting for the character code incorrect?	Make the setting for the same character code as the program's character code. If the character code cannot be determined without difficulty when set to "AUTO," then specify the character code. For more information on how to make the setting, refer to "Descriptions of the Display Menus - 6 USE CODE."

When the spindle does not rotate ...

Is the SPINDLE COVER open?	Close the SPINDLE COVER.
Is [SPINDLE CONTROL] set to [OFF] or [EXTERNAL ONLY]?	Refer to "Descriptions of the Display Menus" and change the setting for "SPINDLE CONTROL" to "ON."

Data cannot be sent

Do the machine's connection parameter settings match the settings for the computer?	Refer to “1-2 Setting the Connection Parameters” to make the correct settings. When you send a file using the MS-DOS prompt in Windows, you make the settings for the computer's communication parameters at the MS-DOS prompt. For more information about how to make the settings, refer to "Downloading Cutting Data - Using the MS-DOS Prompt in Windows."
Has the connection cable come loose?	Make sure the connection cable is plugged in securely with no looseness at either end.
Is the correct connection cable being used?	The type of connection cable varies according to the computer being used. Also, some application software requires the use of a special cable. Make sure the correct cable is being used.

The power does not come on...

Has the power cord come loose?	Make sure the power cord is plugged in securely with no looseness at either end.

2-5 Error Messages

An error message appears on the display when any type of error occurs on the machine.
When an error occurs, operation pauses and an error message is displayed. When this happens, press the [ENTER] key to display the menu in pause.
It is possible to ignore the error and continue cutting, but doing so is not recommended. Operation after occurrence of an error may not be correct.
Instead, stop program execution and correct the place where the error was generated.

Error message	Description
Bad Parameter	The value of a parameter exceeds the allowable range, or the value of the radius for circular interpolation or the amount of offset is not correct.
Address Undefined	Only a parameter has been set. The code which specifies the parameter has not been set.
Parameter Undefined	A parameter has not been set.
Code Cannot Execute	This is displayed when an attempt was made to execute an unrecognizable command, when cutter compensation was started while in the circular interpolation mode, or when an attempt was made to execute a command which cannot be used during tool-diameter or tool-length compensation.
Program Number Not found	The program number specified by M98 could not be found.
Sub-Program Nest Over	An attempt was made to call a fifth-level subprogram from a fourth-level subprogram of a main program.
Parity Error	This appears when auto-recognition of the character code fails, or when character codes that differ from the character-code setting on the machine are received. Make the setting for the character code again.
I/O Err:Framing/Parity Error	This message is displayed when a framing error, parity error, or overrun error occurs when receiving data. It is caused by an incorrect setting in communication parameters (bit rate, parity, stop bit, or data length). Stop sending data from the computer, and make the correct settings for the communication parameters.
I/O Err: Buffer Overflow	Appears if the I/O buffer has overflowed. (There is a problem with the connecting cable, or the settings for Handshaking. Make sure you are using a cable appropriate for the computer being used. Also, check that the setting for Handshaking is correct.)
I/O Err:Indeterminate Error	Appears if a communication error, one uninterpretable by the machine, occurs during data communications.

In the following cases, the display is made when program integrity is analyzed after registering the subprograms.
At such times, the message appears for two or three seconds, then reverts to the original screen. Correct the program, then send it again.

Error message	Description
Sub-Program Regist Error	There are more than ten subprograms. The maximum number of subprograms that can be specified in one set of data is ten. After correcting the program, resend the data.
Duplicate Sub-Program Number	The program contains multiple subprograms with the same program number. The same program number may not be set for more than one subprogram within a single set of data. After correcting the program, resend the data.

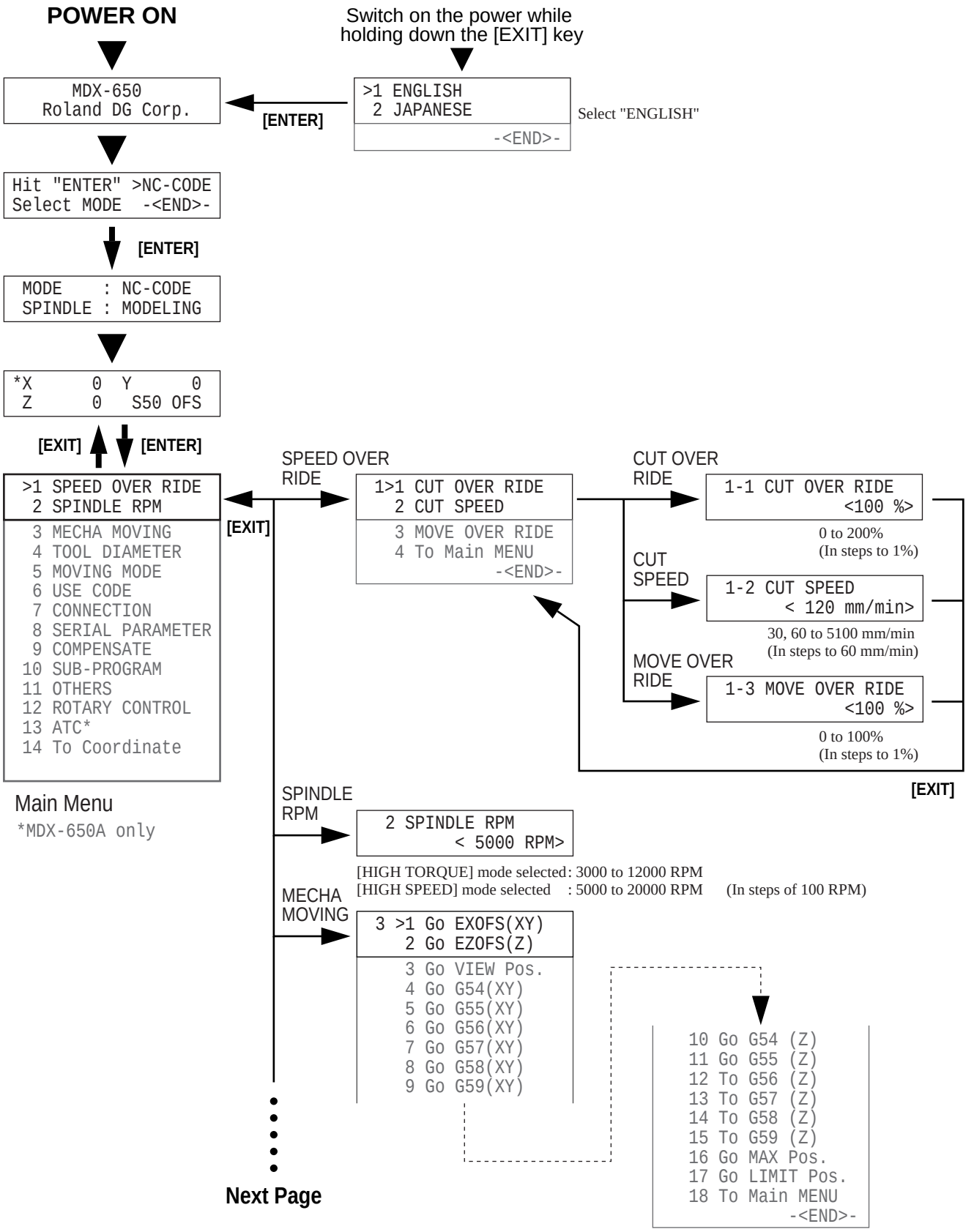
2-6 Other Messages

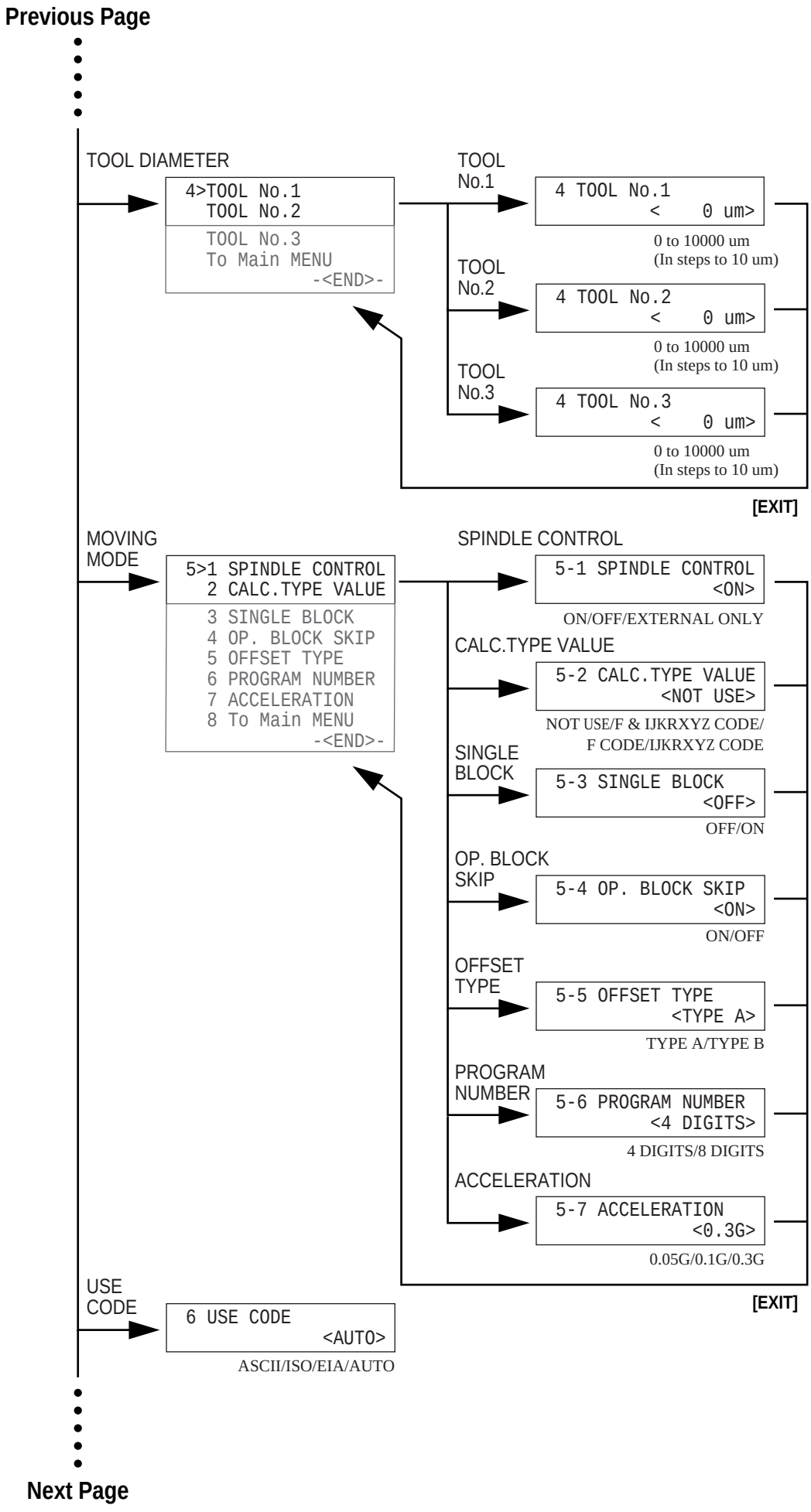
Besides error messages related to commands or communication parameters, the following messages may also appear on the display.

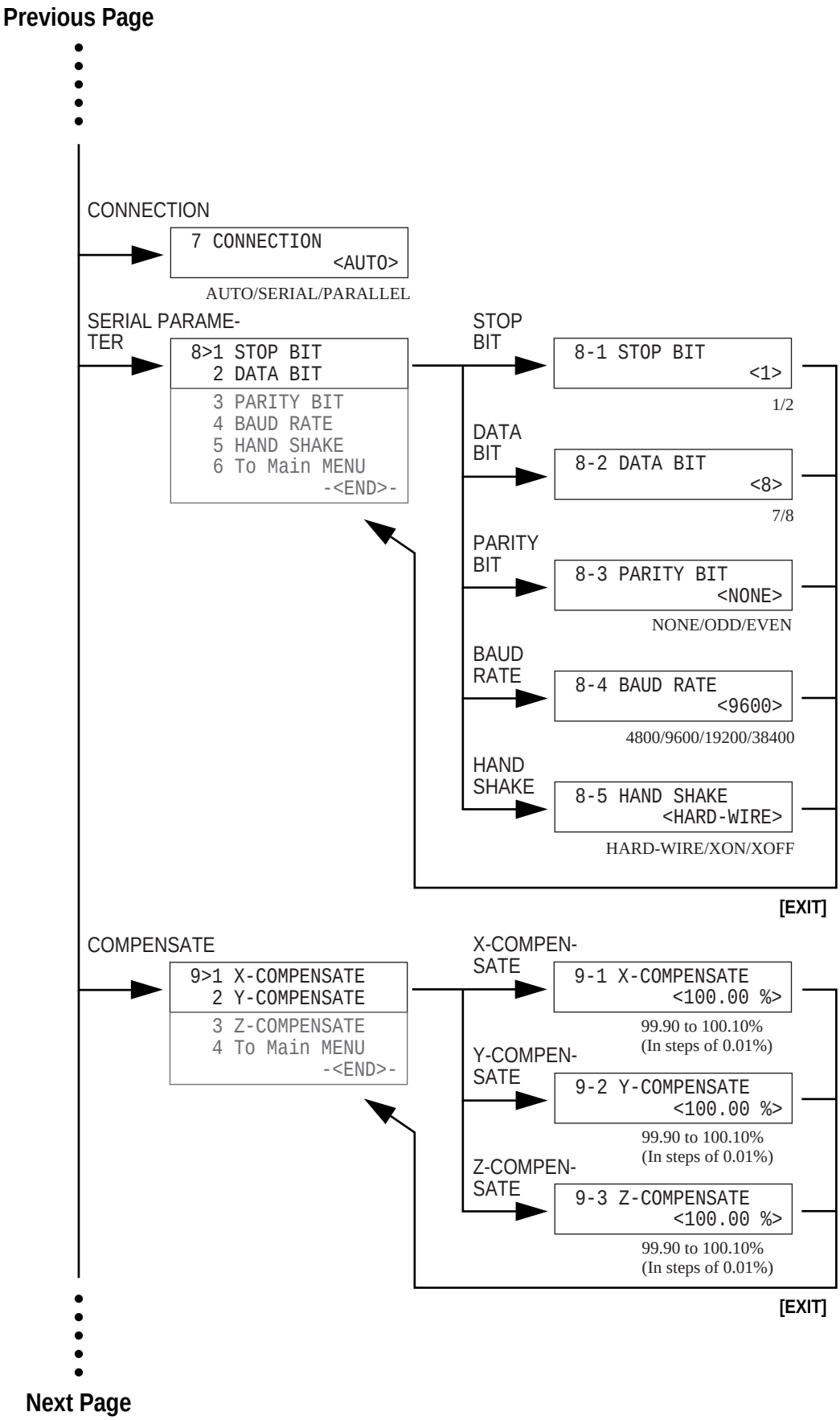
Error message	Description
EMERGENCY STOP EXT1 IS NOT CONNECT	This appears when either the cable connecting the safety cover and the main unit or the key connector is disconnected. Switching off the power clears the message. Make the connections correctly, then switch on the power.
EMERGENCY STOP MOTOR LOCK XYZS	The machine stops automatically if an excessive load is placed on the spindle or the X, Y, or Z axis during cutting. The message shown at left appears at this time. Switching off the power clears the message. The overload may be due to excessive hardness of the material, an excessive amount of cutting-in, a feed rate that is too fast, or operation being impeded by cuttings. Take action such as changing the cutting parameters or cleaning the machine to eliminate the cause of the over load.
EMERGENCY STOP SP/SFTY COVER OPEN	This appears when the spindle cover or the safety cover is opened during operation. SP: Spindle cover SFTY: Safety cover Switching off the power clears the message. Close the covers, then switch on the power.
CAUTION! SP COVER OPEN	This appears when the spindle cover is opened during standby. SP: Spindle cover Closing the cover clears the message and returns the display to the coordinate view screen.
11-1 SENSOR MODE Please Cursor Move	This appears when the sensor mode is entered.
11-1 SENSOR MODE Z0SENSOR NOTHING	This error appears if the Z0 sensor is not connected when entering the sensor mode. The display shows the message for several seconds, then returns to the previous screen. Connection the Z0 sensor before entering the sense mode.
11-1 SENSOR MODE SET Z ORIGIN!	This appears when Z0 is set in the sensor mode.
Comp. Effect After Power On Again	This appears when the value for [COMPENSATE] is changed. After setting the distance compensation value, switch the power off and on again to activate the change.
CAN'T COPY TOO BIG DATA	When the amount of cutting data exceeds the capacity of the data buffer, this message appear when an attempt is made to perform recutting with this data. The data cannot all fit in the data buffer, so repeat cutting cannot be performed.
CAN'T COPY BUFFER EMPTY	This message appears if repeat cutting is attempted when the data buffer is empty. Send cutting data before performing repeat cutting.
CAN'T COPY COVER OPEN	This appears when the [COPY] key is pressed to attempt to perform copying while the spindle cover or the safety cover is open. Closing the cover pauses operation. To perform copying, choose [CONTINUE]. To stop copying, choose [STOP].

2-7 Display Menus Flowchart

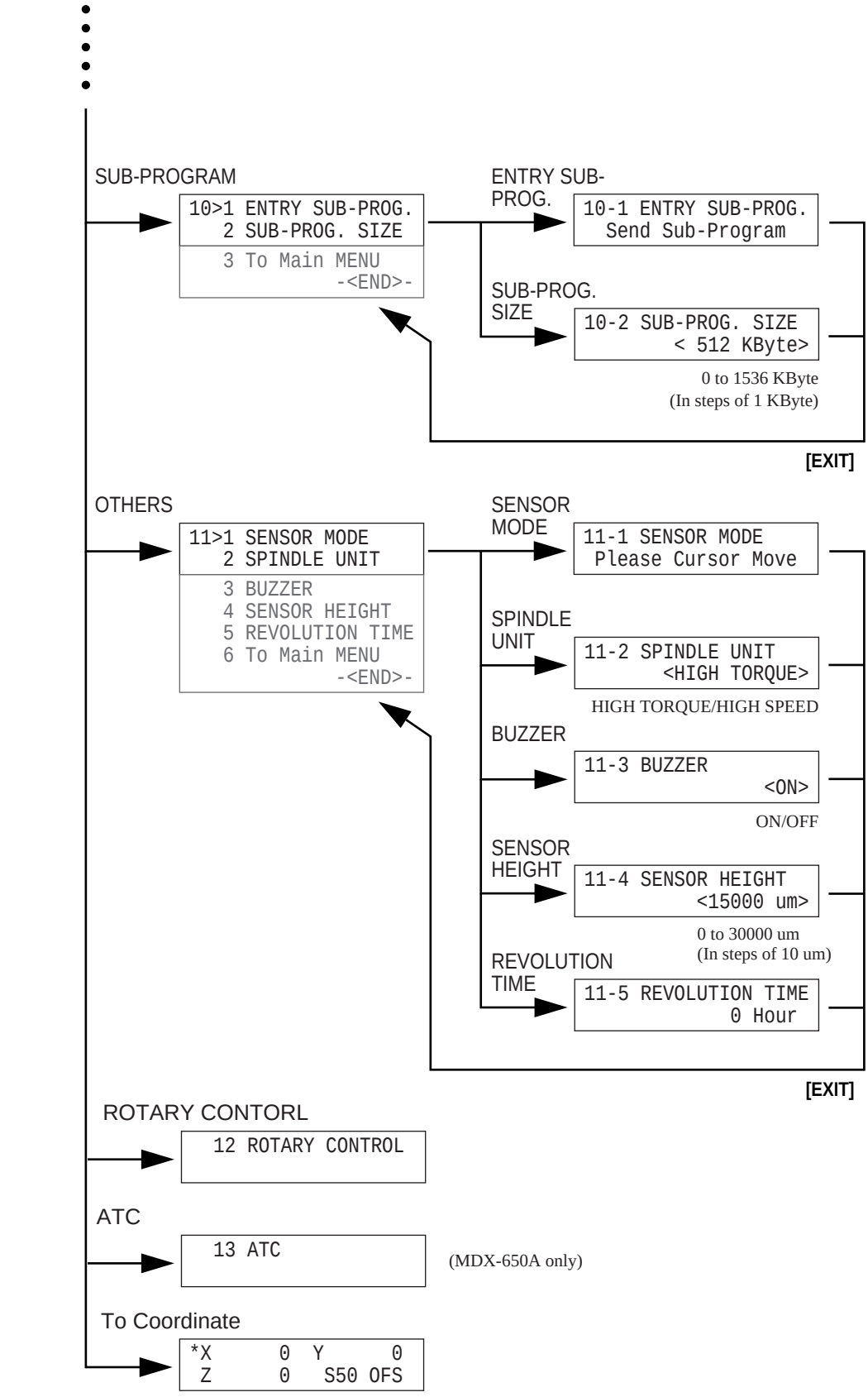
MDX-650A/650



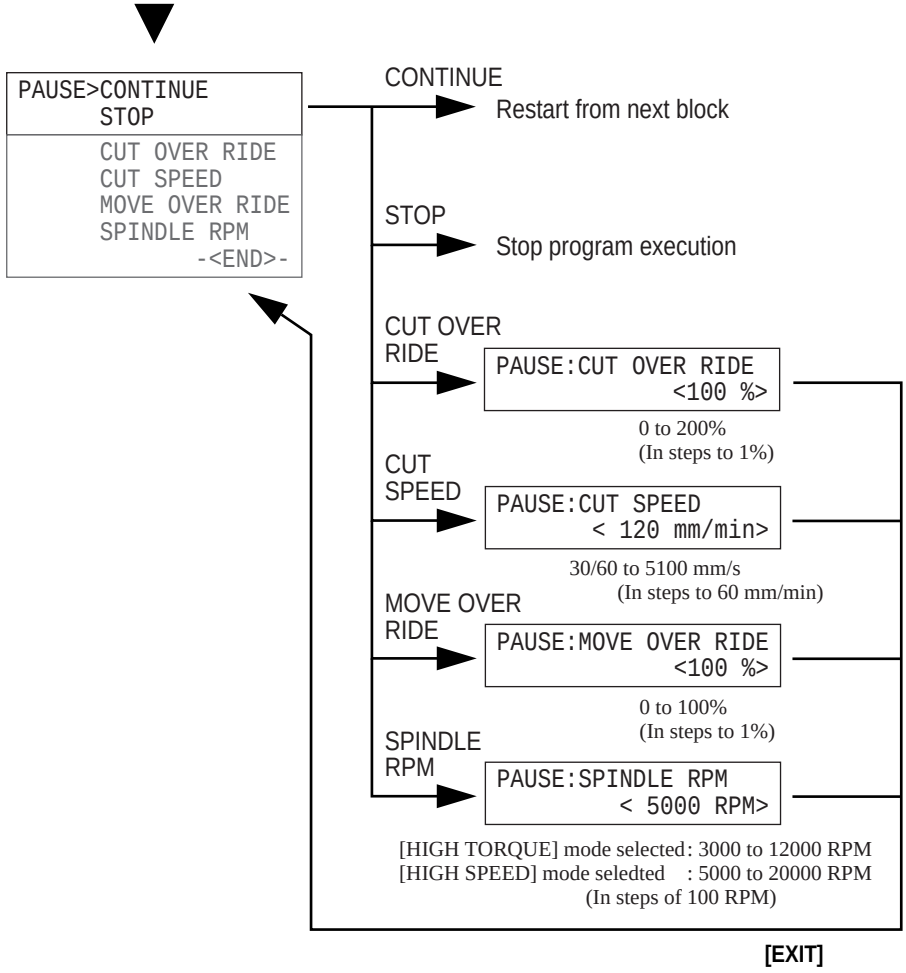

Previous Page
TOOL DIAMETER
4>TOOL No.1
TOOL No.2
TOOL No.3
To Main MENU
-<END>-
TOOL No.1
4 TOOL No.1
< 0 um>
0 to 10000 um
(In steps to 10 um)
TOOL No.2
4 TOOL No.2
< 0 um>
0 to 10000 um
(In steps to 10 um)
TOOL No.3
4 TOOL No.3
< 0 um>
0 to 10000 um
(In steps to 10 um)
[EXIT]
MOVING MODE
5>1 SPINDLE CONTROL
2 CALC.TYPE VALUE
3 SINGLE BLOCK
4 OP. BLOCK SKIP
5 OFFSET TYPE
6 PROGRAM NUMBER
7 ACCELERATION
8 To Main MENU
-<END>-
SPINDLE CONTROL
5-1 SPINDLE CONTROL
<ON>
ON/OFF/EXTERNAL ONLY
CALC.TYPE VALUE
5-2 CALC.TYPE VALUE
<NOT USE>
NOT USE/F & IJKRXYZ CODE/
F CODE/IJKRXYZ CODE
SINGLE BLOCK
5-3 SINGLE BLOCK
<OFF>
OFF/ON
OP. BLOCK SKIP
5-4 OP. BLOCK SKIP
<ON>
ON/OFF
OFFSET TYPE
5-5 OFFSET TYPE
<TYPE A>
TYPE A/TYPE B
PROGRAM NUMBER
5-6 PROGRAM NUMBER
<4 DIGITS>
4 DIGITS/8 DIGITS
ACCELERATION
5-7 ACCELERATION
<0.3G>
0.05G/0.1G/0.3G
[EXIT]
USE CODE
6 USE CODE
<AUTO>
ASCII/ISO/EIA/AUTO
Next Page


Previous Page
CONNECTION
7 CONNECTION
<AUTO>
AUTO/SERIAL/PARALLEL
SERIAL PARAME-
TER
8>1 STOP BIT
2 DATA BIT
3 PARITY BIT
4 BAUD RATE
5 HAND SHAKE
6 To Main MENU
-<END>-
STOP
BIT
8-1 STOP BIT
<1>
1/2
DATA
BIT
8-2 DATA BIT
<8>
7/8
PARITY
BIT
8-3 PARITY BIT
<NONE>
NONE/ODD/EVEN
BAUD
RATE
8-4 BAUD RATE
<9600>
4800/9600/19200/38400
HAND
SHAKE
8-5 HAND SHAKE
<HARD-WIRE>
HARD-WIRE/XON/XOFF
[EXIT]
COMPENSATE
9>1 X-COMPENSATE
2 Y-COMPENSATE
3 Z-COMPENSATE
4 To Main MENU
-<END>-
X-COMPEN-
SATE
9-1 X-COMPENSATE
<100.00 %>
99.90 to 100.10%
(In steps of 0.01%)
Y-COMPEN-
SATE
9-2 Y-COMPENSATE
<100.00 %>
99.90 to 100.10%
(In steps of 0.01%)
Z-COMPEN-
SATE
9-3 Z-COMPENSATE
<100.00 %>
99.90 to 100.10%
(In steps of 0.01%)
[EXIT]
Next Page

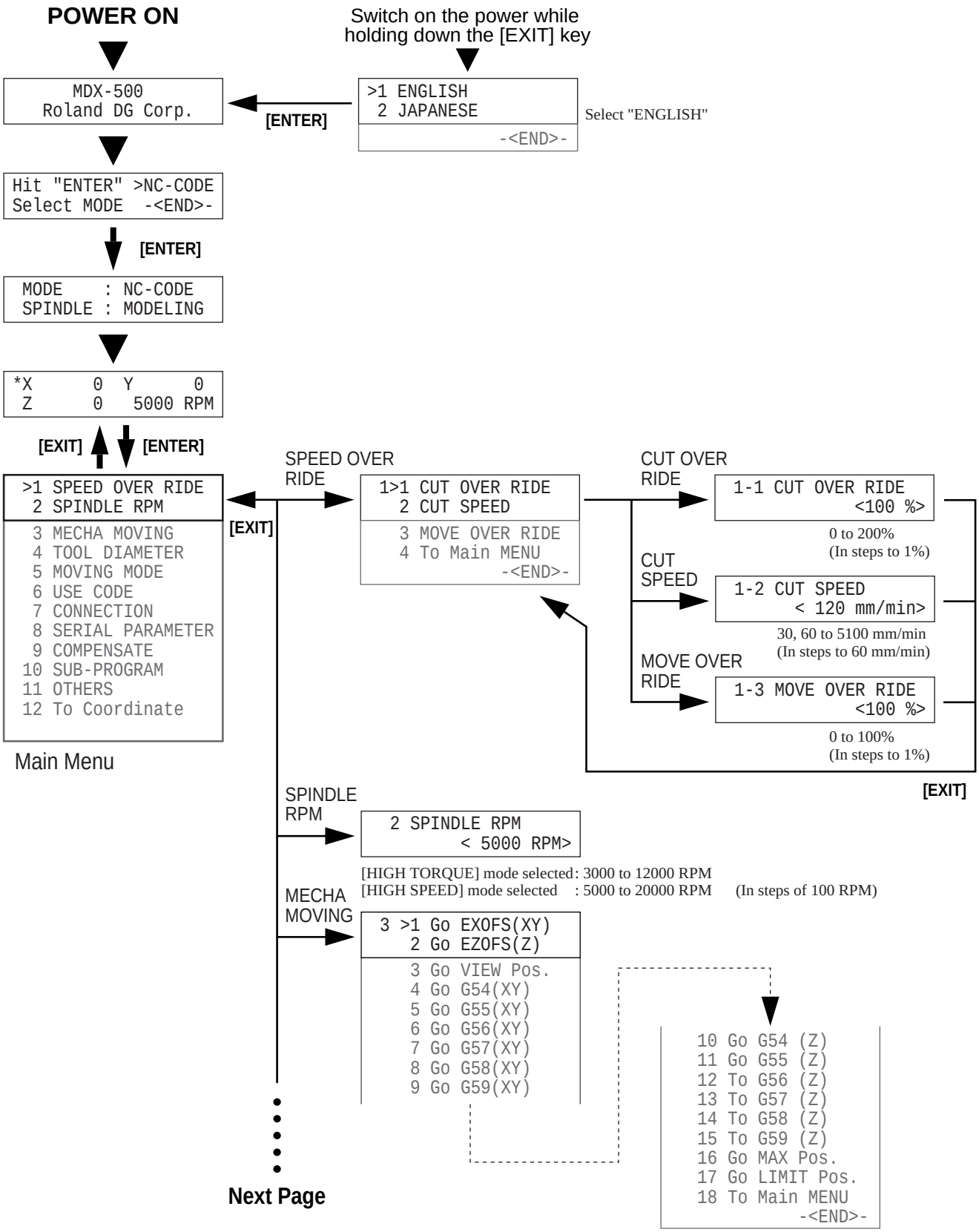
Previous Page

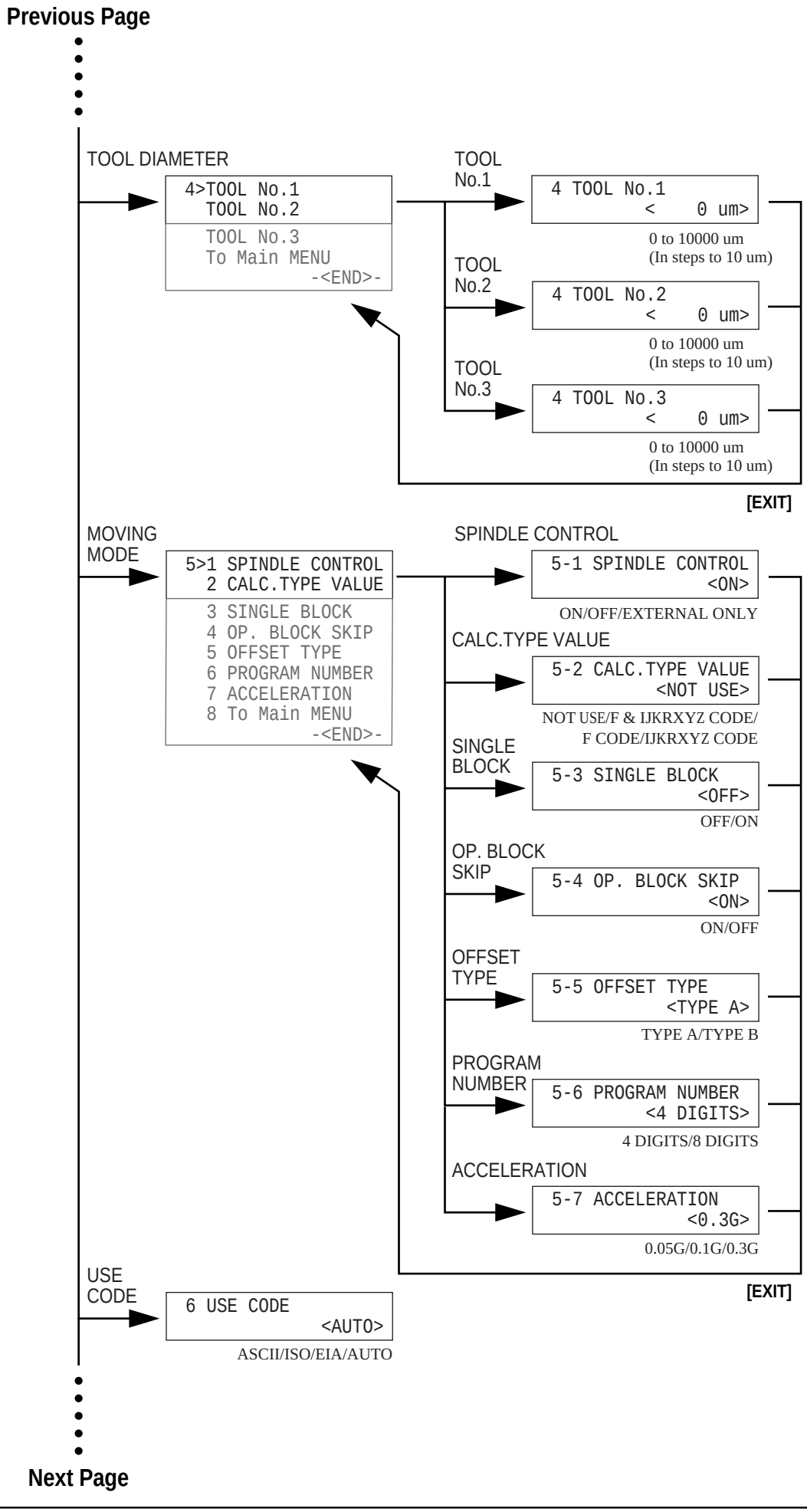


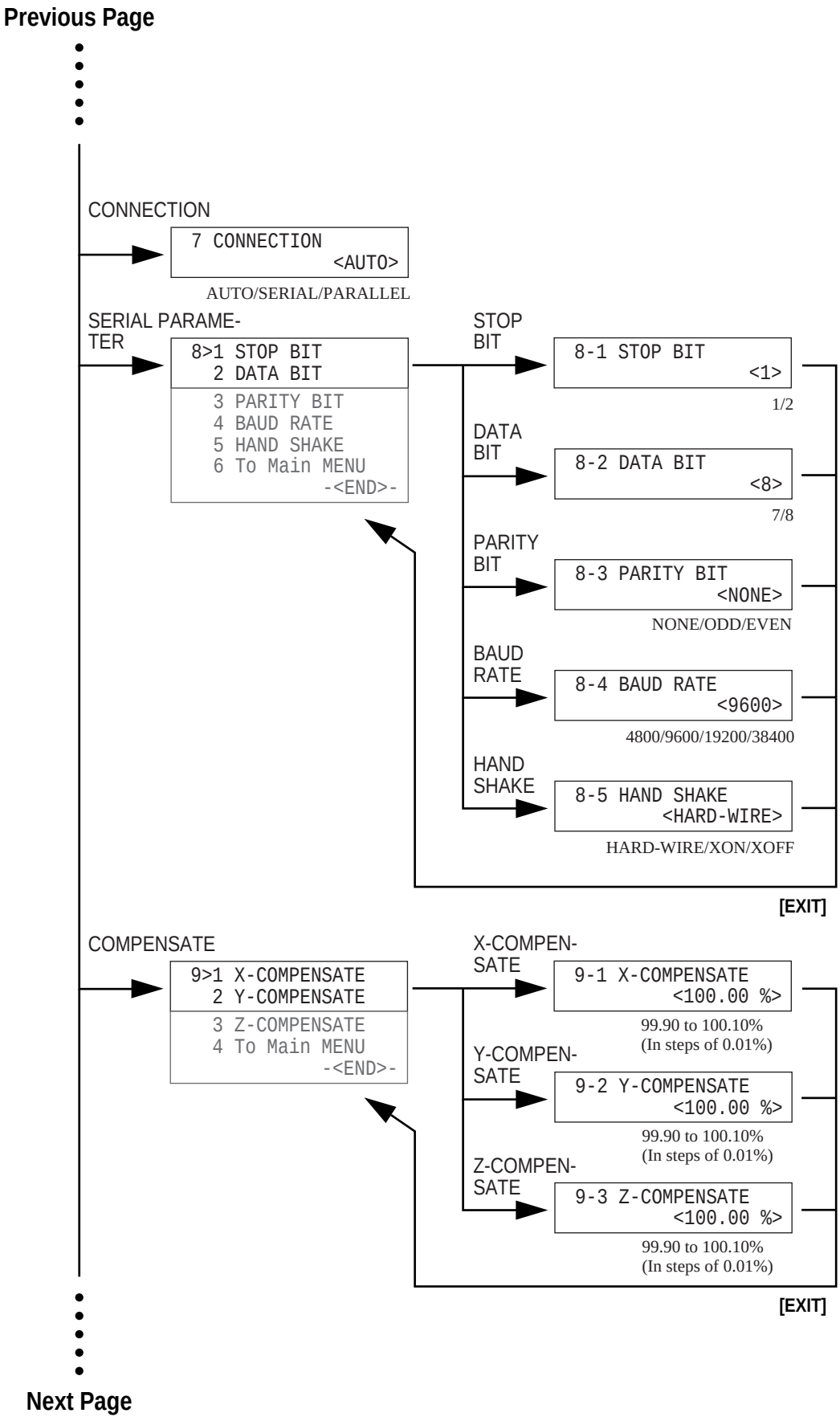
Press the [PAUSE] key

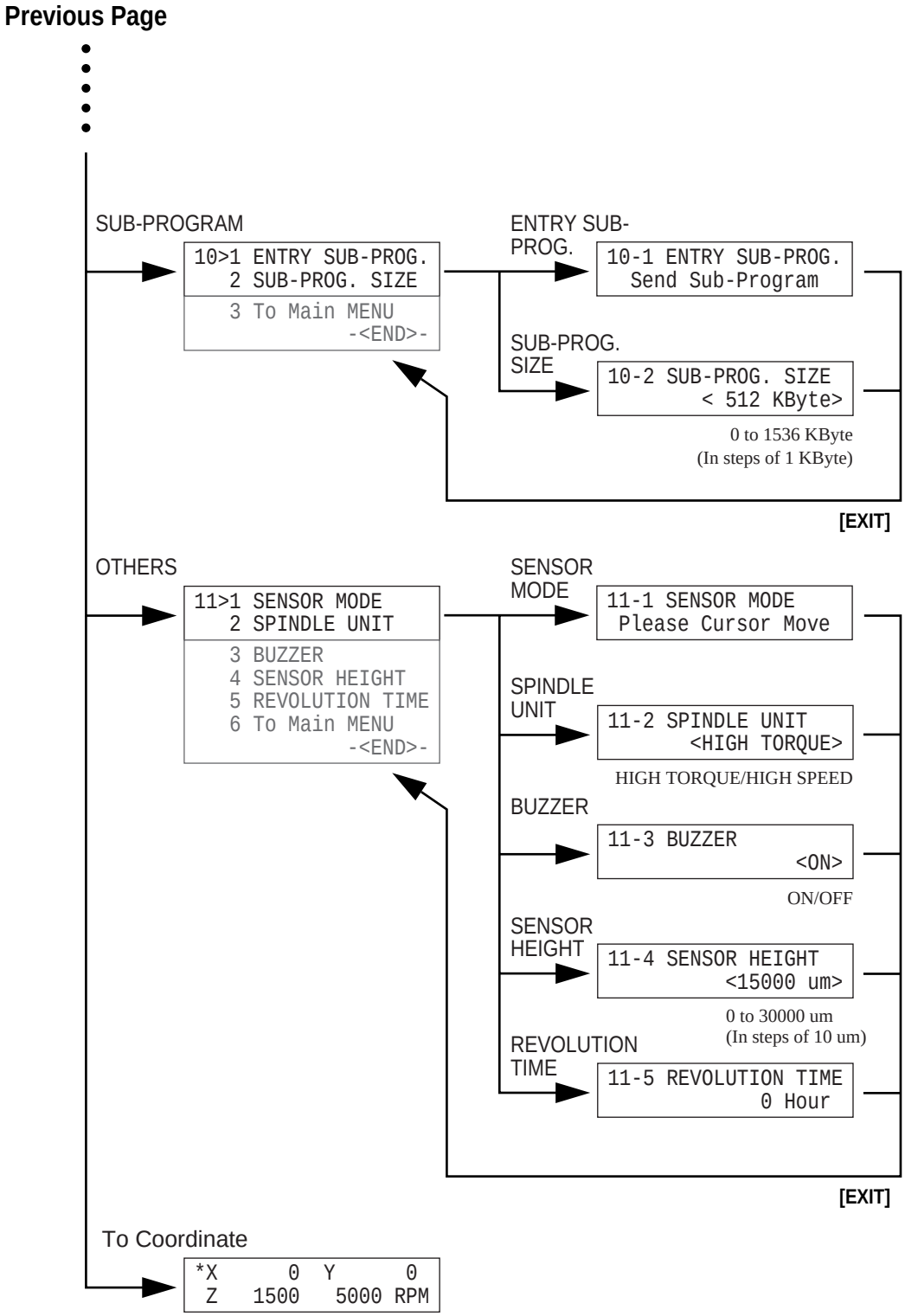


MDX-500

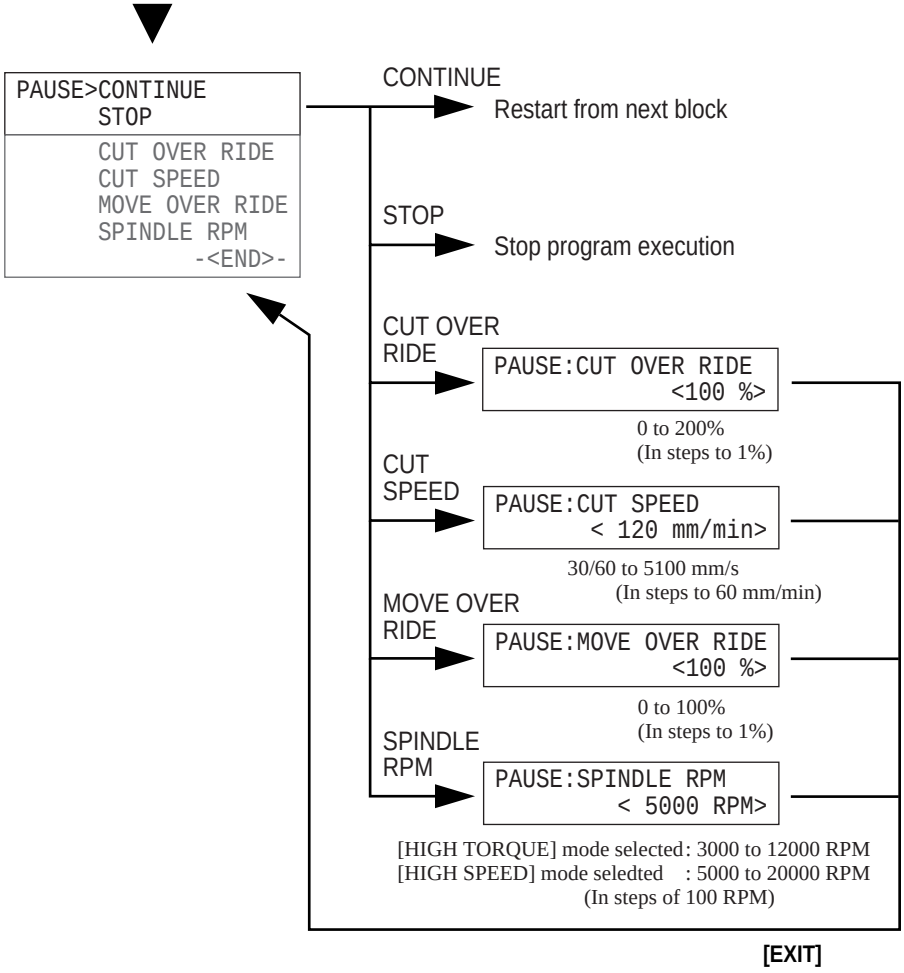

POWER ON
Switch on the power while holding down the [EXIT] key
MDX-500
Roland DG Corp.
[ENTER]
>1 ENGLISH
2 JAPANESE
-<END>-
Select "ENGLISH"
Hit "ENTER" >NC-CODE
Select MODE -<END>-
[ENTER]
MODE : NC-CODE
SPINDLE : MODELING
*X 0 Y 0
Z 0 5000 RPM
[EXIT]
[ENTER]
>1 SPEED OVER RIDE
2 SPINDLE RPM
3 MECHA MOVING
4 TOOL DIAMETER
5 MOVING MODE
6 USE CODE
7 CONNECTION
8 SERIAL PARAMETER
9 COMPENSATE
10 SUB-PROGRAM
11 OTHERS
12 To Coordinate
Main Menu
[EXIT]
SPEED OVER RIDE
1>1 CUT OVER RIDE
2 CUT SPEED
3 MOVE OVER RIDE
4 To Main MENU
-<END>-
CUT OVER RIDE
1-1 CUT OVER RIDE
<100 %>
0 to 200%
(In steps to 1%)
CUT SPEED
1-2 CUT SPEED
< 120 mm/min>
30, 60 to 5100 mm/min
(In steps to 60 mm/min)
MOVE OVER RIDE
1-3 MOVE OVER RIDE
<100 %>
0 to 100%
(In steps to 1%)
[EXIT]
SPINDLE RPM
2 SPINDLE RPM
< 5000 RPM>
[HIGH TORQUE] mode selected: 3000 to 12000 RPM
[HIGH SPEED] mode selected : 5000 to 20000 RPM (In steps of 100 RPM)
MECHA MOVING
3 >1 Go EXOFS(XY)
2 Go EZOFS(Z)
3 Go VIEW Pos.
4 Go G54(XY)
5 Go G55(XY)
6 Go G56(XY)
7 Go G57(XY)
8 Go G58(XY)
9 Go G59(XY)
10 Go G54 (Z)
11 Go G55 (Z)
12 To G56 (Z)
13 To G57 (Z)
14 To G58 (Z)
15 To G59 (Z)
16 Go MAX Pos.
17 Go LIMIT Pos.
18 To Main MENU
-<END>-
Next Page


Previous Page
TOOL DIAMETER
4>TOOL No.1
TOOL No.2
TOOL No.3
To Main MENU
-<END>-
TOOL No.1
4 TOOL No.1 < 0 um>
0 to 10000 um
(In steps to 10 um)
TOOL No.2
4 TOOL No.2 < 0 um>
0 to 10000 um
(In steps to 10 um)
TOOL No.3
4 TOOL No.3 < 0 um>
0 to 10000 um
(In steps to 10 um)
[EXIT]
MOVING MODE
5>1 SPINDLE CONTROL
2 CALC.TYPE VALUE
3 SINGLE BLOCK
4 OP. BLOCK SKIP
5 OFFSET TYPE
6 PROGRAM NUMBER
7 ACCELERATION
8 To Main MENU
-<END>-
SPINDLE CONTROL
5-1 SPINDLE CONTROL <ON>
ON/OFF/EXTERNAL ONLY
CALC.TYPE VALUE
5-2 CALC.TYPE VALUE <NOT USE>
NOT USE/F & IJKRXYZ CODE/
F CODE/IJKRXYZ CODE
SINGLE BLOCK
5-3 SINGLE BLOCK <OFF>
OFF/ON
OP. BLOCK SKIP
5-4 OP. BLOCK SKIP <ON>
ON/OFF
OFFSET TYPE
5-5 OFFSET TYPE <TYPE A>
TYPE A/TYPE B
PROGRAM NUMBER
5-6 PROGRAM NUMBER <4 DIGITS>
4 DIGITS/8 DIGITS
ACCELERATION
5-7 ACCELERATION <0.3G>
0.05G/0.1G/0.3G
[EXIT]
USE CODE
6 USE CODE <AUTO>
ASCII/ISO/EIA/AUTO
Next Page


Previous Page
CONNECTION
7 CONNECTION
<AUTO>
AUTO/SERIAL/PARALLEL
SERIAL PARAME-
TER
8>1 STOP BIT
2 DATA BIT
3 PARITY BIT
4 BAUD RATE
5 HAND SHAKE
6 To Main MENU
-<END>-
STOP
BIT
8-1 STOP BIT
<1>
1/2
DATA
BIT
8-2 DATA BIT
<8>
7/8
PARITY
BIT
8-3 PARITY BIT
<NONE>
NONE/ODD/EVEN
BAUD
RATE
8-4 BAUD RATE
<9600>
4800/9600/19200/38400
HAND
SHAKE
8-5 HAND SHAKE
<HARD-WIRE>
HARD-WIRE/XON/XOFF
[EXIT]
COMPENSATE
9>1 X-COMPENSATE
2 Y-COMPENSATE
3 Z-COMPENSATE
4 To Main MENU
-<END>-
X-COMPEN-
SATE
9-1 X-COMPENSATE
<100.00 %>
99.90 to 100.10%
(In steps of 0.01%)
Y-COMPEN-
SATE
9-2 Y-COMPENSATE
<100.00 %>
99.90 to 100.10%
(In steps of 0.01%)
Z-COMPEN-
SATE
9-3 Z-COMPENSATE
<100.00 %>
99.90 to 100.10%
(In steps of 0.01%)
[EXIT]
Next Page


Previous Page
SUB-PROGRAM
10>1 ENTRY SUB-PROG.
2 SUB-PROG. SIZE
3 To Main MENU
-<END>-
ENTRY SUB-PROG.
10-1 ENTRY SUB-PROG.
Send Sub-Program
SUB-PROG. SIZE
10-2 SUB-PROG. SIZE
< 512 KByte>
0 to 1536 KByte
(In steps of 1 KByte)
[EXIT]
OTHERS
11>1 SENSOR MODE
2 SPINDLE UNIT
3 BUZZER
4 SENSOR HEIGHT
5 REVOLUTION TIME
6 To Main MENU
-<END>-
SENSOR MODE
11-1 SENSOR MODE
Please Cursor Move
SPINDLE UNIT
11-2 SPINDLE UNIT
<HIGH TORQUE>
HIGH TORQUE/HIGH SPEED
BUZZER
11-3 BUZZER
<ON>
ON/OFF
SENSOR HEIGHT
11-4 SENSOR HEIGHT
<15000 um>
0 to 30000 um
(In steps of 10 um)
REVOLUTION TIME
11-5 REVOLUTION TIME
0 Hour
[EXIT]
To Coordinate
*X 0 Y 0
Z 1500 5000 RPM

Press the [PAUSE] key



- MEMO -

26015335-R5

R5-021007