

CAMM-3

Modeling Machine by Roland DG Corporation

PNC-3200

USER'S MANUAL

3

Cutting Using NC Code

Thank you very much for purchasing the PNC-3200.

- 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.

Table of Contents

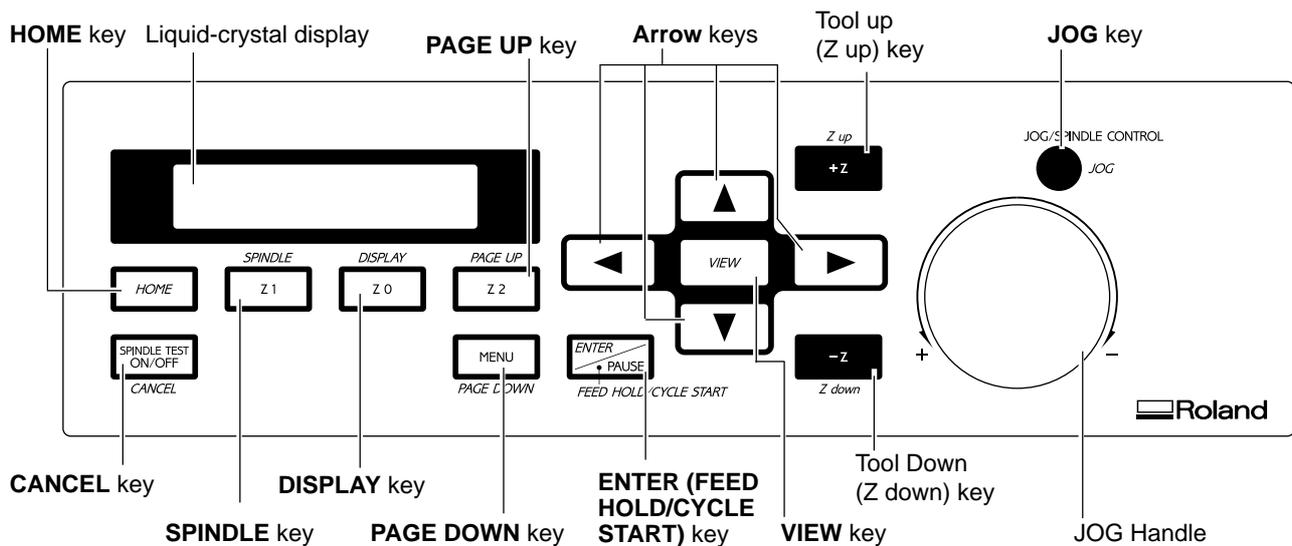
Introduction	2
 Part 1 Basic Operations	
1 The Switch Panel	3
Making Settings Using the Liquid-crystal Display	4
2 Setting a Workpiece Coordinate System	5
Setting a Workpiece Coordinate System Using G92	7
Setting a Workpiece Coordinate System Using G54 to G59	8
Setting the Amount of Shift from the EXOFS As a Numerical Value	9
Setting the Current Tool Location As the Workpiece Coordinate System (G54 to G59) Origin Point ...	10
3 Sending Cutting Data	12
Selecting the Character Code	12
Cutting Procedure	13
Performing Cutting As Data Is Being Received	13
Saving Cutting Data in the Buffer, Then Calling Up the Data and Performing Cutting	14
Settings Affecting Operation While Cutting	15
Checking the Code Being Executed on the Display	15
Pausing Operation and Adjusting the Rotating Speed and Feed Speed	16
Terminating Cutting	17
4 Finishing	18
 Part 2 User's Reference	
Cutting Area	19
Operations Using the Display	20
Changing to Other-language Messages on the Liquid-crystal Display	21
Descriptions of the Display Menus	22
Coordinate-value Displays	23
Main Menus	24
Submenus	25
What to Do If...	38
Error Message	39
Errors Occurring During Program Execution	39
Other Error Messages	40
Display Menus Flowchart	42

Introduction

This document describes operation when performing cutting with the PNC-3200 using NC codes. To perform cutting using NC codes, the command selection must be set to NC Code when the PNC-3200 is switched on. For information on how to select commands, see User's Manual 1 -- "Setup and Maintenance."

Part 1 Basic Operations

1 The Switch Panel

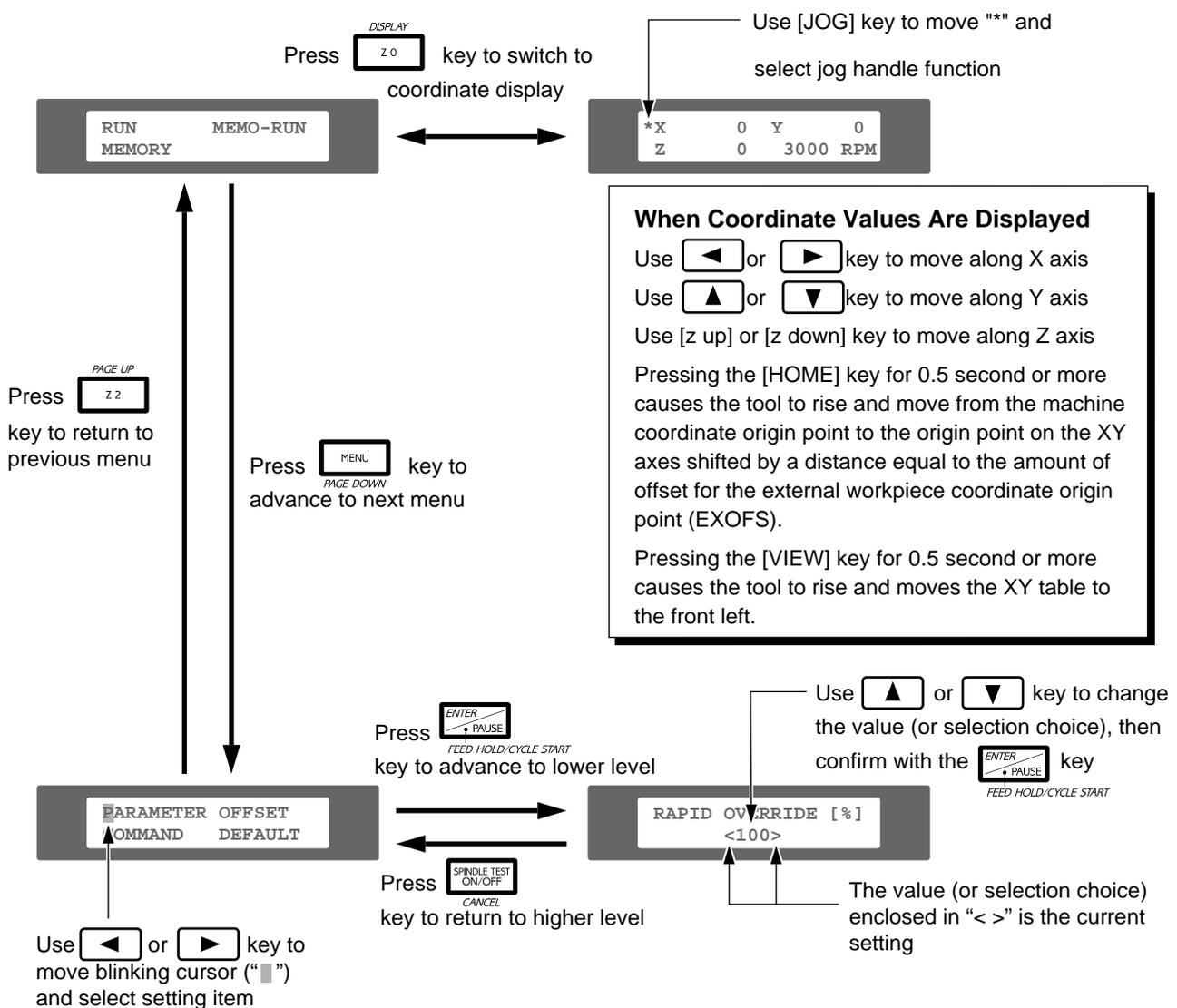


* Valid key names when NC code has been selected are shown in italic type.

- Liquid-crystal display** This displays the various setting items and selection choices (or numerical values) for the PNC-3200, as well as tool coordinates. Error messages also appear here in the event of a problem.
- Arrow keys** When coordinate values are displayed on the liquid-crystal display, pressing an arrow key causes the XY table to move in the corresponding direction. These keys are also used to make selections and change values when making settings at other than the coordinate-display menu.
- Tool up (Z up) key** When coordinate values are displayed on the liquid-crystal display, pressing this key causes the tool to move in the positive direction of the Z axis (i.e., upward). Movement is always at a constant speed.
- Tool down (Z down) key** When coordinate values are displayed on the liquid-crystal display, pressing this key causes the tool to move in the negative direction of the Z axis (i.e., downward). Movement is always at a constant speed.
- HOME key** This key is functional when coordinate values are displayed on the liquid-crystal display. Pressing this key causes the tool to rise to the topmost point and move to the current home position. The home position is an origin point on the X and Y axes which is shifted from the machine-coordinate origin point by a distance equal to the amount of offset for the external workpiece origin point (EXOFS). See "2 Setting a Workpiece Coordinate System" for more information on the EXOFS.
- DISPLAY key** This toggles the liquid-crystal display between "setting display" and "coordinate-value display". When cutting operations are in progress, this key toggles the liquid-crystal display between "coordinate-value display" and "display of the block being executed".
- PAGE DOWN key** This advances the liquid-crystal display to the next menu (i.e., the panel display changes).
- PAGE UP key** This moves the liquid-crystal display back to the previous menu (i.e., the panel display changes).
- ENTER (FEED HOLD /CYCLE START) key** This is used to confirm a value that has been set or a selection that has been made when making settings using the liquid-crystal display. When pressed while machining is in progress, it pauses operation.
- CANCEL key** When making settings at other than the coordinate-display menu, pressing this key either returns the display to the menu one level higher, or cancels the change for a setting that is in progress. However, once the [ENTER] key has been pressed to confirm a change, the setting cannot be canceled.

- SPINDLE key** Pressing this key starts or stops the spindle motor.
- VIEW key** This key is functional when coordinate values are displayed on the liquid-crystal display. Pressing this key causes the tool to rise to the topmost point and moves the XY table to the front left.
- JOG handle** This is used for inching the XY table and cutting tool (in steps of 0.01 mm (0.00039")), and also to set the speed of the spindle motor.
- JOG key** This key is used together with the liquid-crystal display to select the function of the JOG handle.

Making Settings Using the Liquid-crystal Display



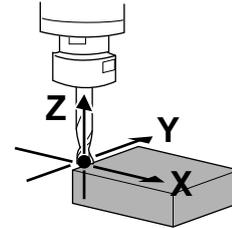
2 Setting a Workpiece Coordinate System

A workpiece coordinate system is a coordinate system for workpiece machining.

When performing output from the included software (MODELA Player, Dr. Engrave or the 2.5D DRIVER), or when you don't know what coordinate settings have been made with the program you're using, follow the steps below to make the settings.

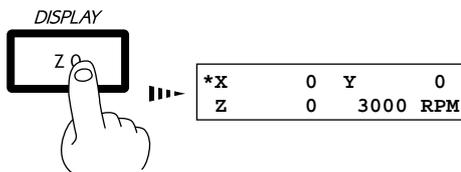
When the coordinate settings of the NC codes that are actually sent are clearly known, see "Setting a Workpiece Coordinate System with G92" or "Setting a Workpiece Coordinate System with G54 to G59." In such cases, make sure which method for setting the coordinate system has been used for the program you want to run.

Set the present tool position to the workpiece coordinate origin point.

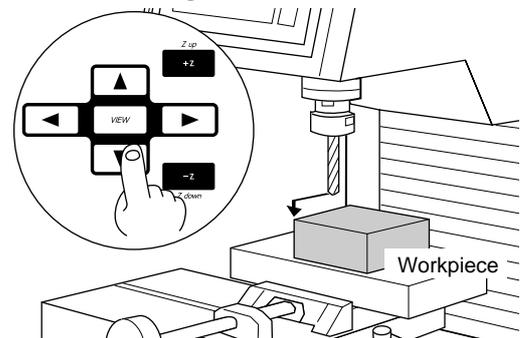


Move the tool to the starting point assumed by the program. This example assumes that the front-left corner of the workpiece is to be specified as the origin point for the workpiece coordinate system. The same procedure is used to move the tool to a desired location.

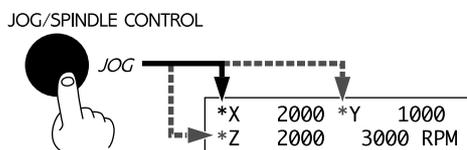
- 1 Press the [DISPLAY] key to make the following screen appear on the display.



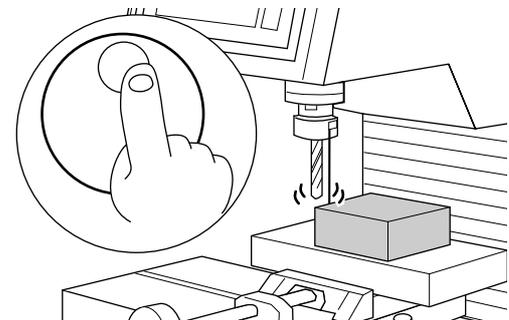
- 2 Press the arrow keys and the Z up/down keys to move the cutting tool to a position close to the front left corner of the workpiece.



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

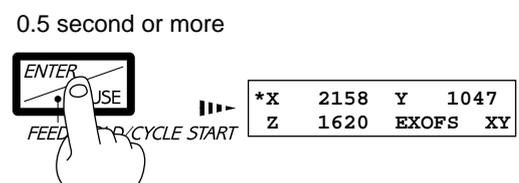


- 4 Use the jog handle to move the tool a little at a time.

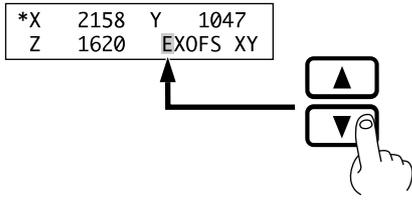


- 5 Repeat steps 3 and 4 to align the tool at the front-left corner of the material to be machined.

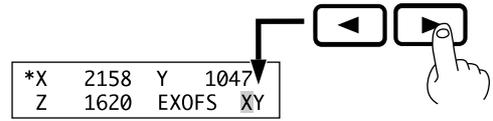
- 6 Hold down the [ENTER] key for 0.5 second or more.



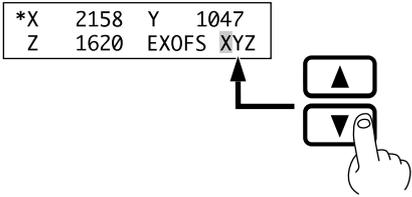
7 Press the [▲] or [▼] key to display “EX0FS”



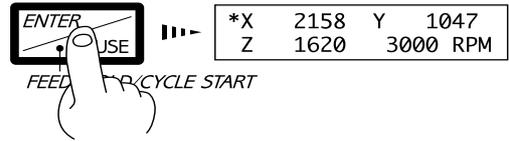
8 Press the [▶] key to move the blinking cursor (“█”) to the displayed letter for the axis.



9 Press the [▲] or [▼] key to display “XYZ.”

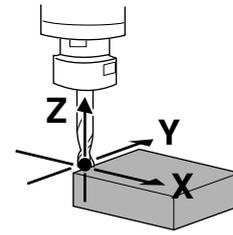
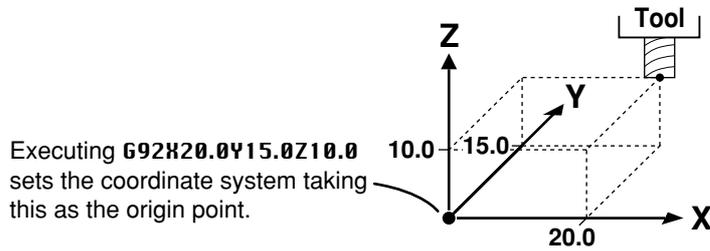


10 Press the [ENTER] key. The display flashes once, then returns to displaying the coordinate values.



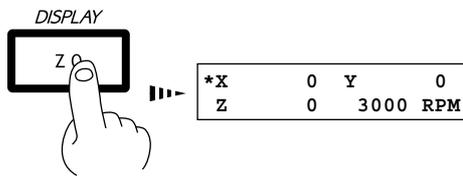
Setting a Workpiece Coordinate System Using G92

G92 sets the current tool location to a desired location on a workpiece coordinate system.

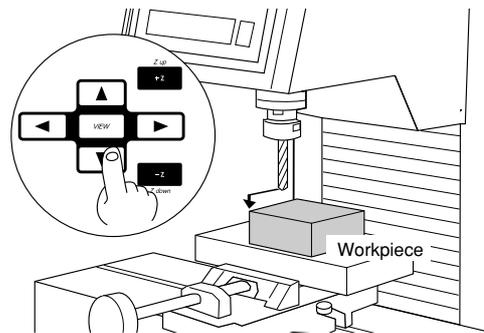


Move the tool to the starting point assumed by the program. This example assumes that the front-left corner of the workpiece is to be specified as the origin point for the workpiece coordinate system. The same procedure is used to move the tool to a desired location.

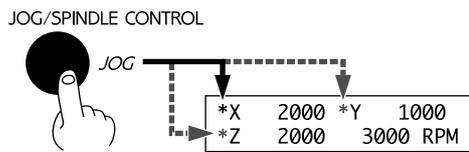
- 1 Press the [DISPLAY] key to make the following screen appear on the display.



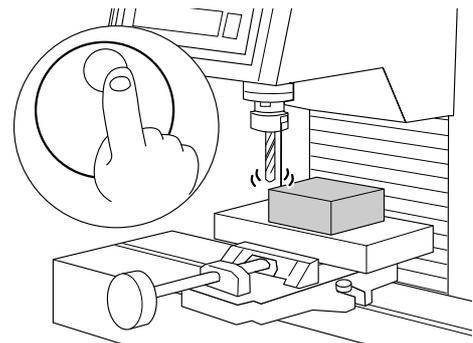
- 2 Press the arrow keys and the Z up/down keys to move the cutting tool to a position close to the front left corner of the workpiece.



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



- 4 Use the jog handle to move the tool a little at a time.

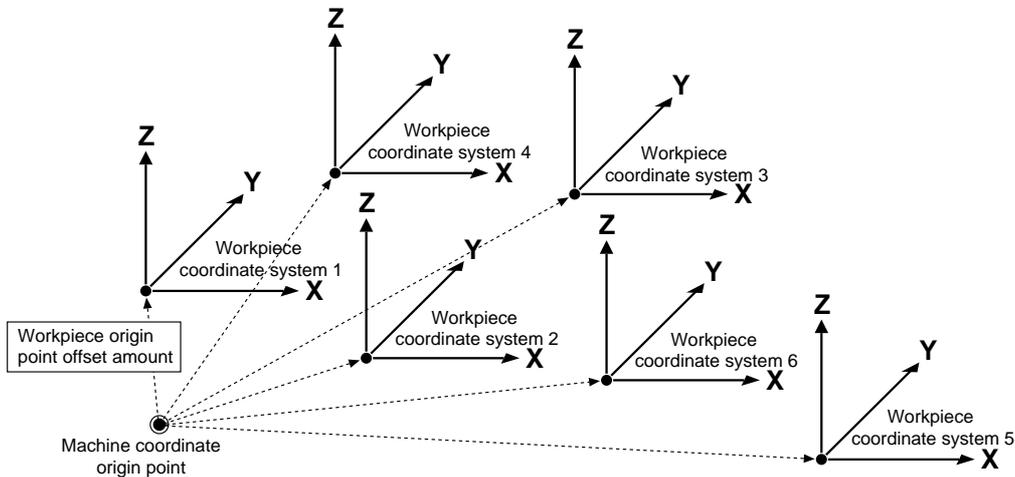


- 5 Repeat steps 3 and 4 to align the tool at the front-left corner of the material to be machined.

Setting a Workpiece Coordinate System Using G54 to G59

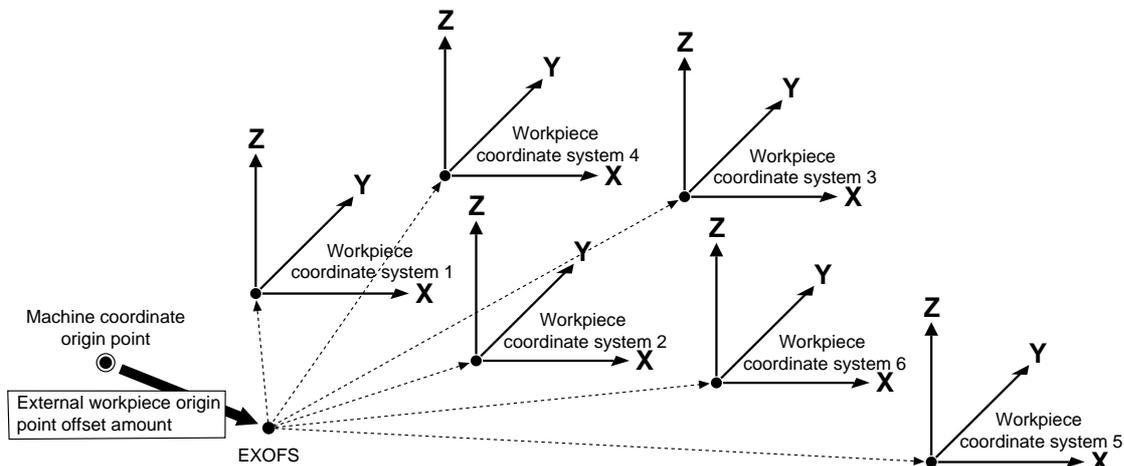
G54 to G59 can be used to set up to six workpiece coordinate systems on the PNC-3200, and select a coordinate system from among these by means of the program.

The workpiece coordinate systems 1 through 6 are set by specifying the amount of shift (the amount of workpiece origin point offset) from the machine coordinate origin point to the workpiece coordinate origin point.



It is also possible to shift the six set workpiece coordinates by a desired distance at one time. Because the workpiece origin point is offset, this is called EXOFS (external workpiece origin point offset amount).

When an external workpiece origin point offset amount (EXOFS) has been set, this specifies the amount of shift from the EXOFS



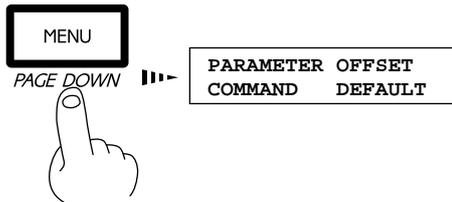
Set the workpiece coordinate system assumed by the program.

Either of two methods can be used to set G54 to G59: setting the amount of shift from the EXOFS as a numerical value, and setting the current tool location as the workpiece coordinate system origin point.

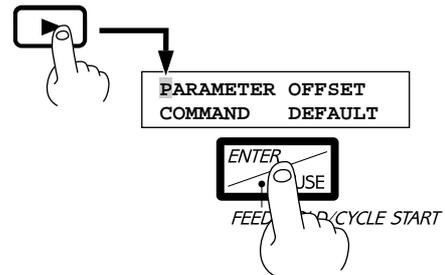
Setting the Amount of Shift from the EXOFS As a Numerical Value

This explanation assumes that G54 (workpiece coordinate system 1) is to be set to a location that is shifted from the machine coordinate origin point by 50 mm on the X axis, 50 mm on the Y axis, and -70 mm on the Z axis. The EXOFS is assumed to be "0" (zero). The settings for G56 to G59 are made in the same way.

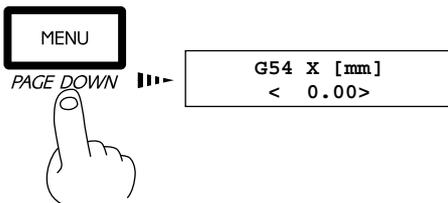
- 1** Press the [PAGE DOWN] key to make the following screen appear on the display.



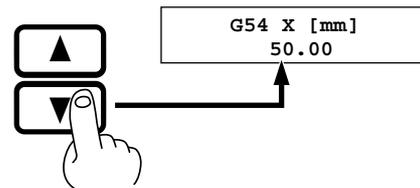
- 2** Press the [▶] key to move the blinking cursor ("█") to "PARAMETER," and then press the [ENTER] key.



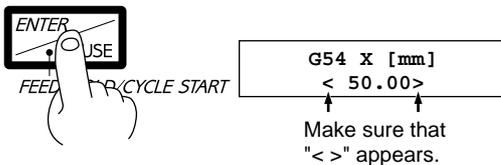
- 3** Press the [PAGE DOWN] key to make the following screen appear on the display.



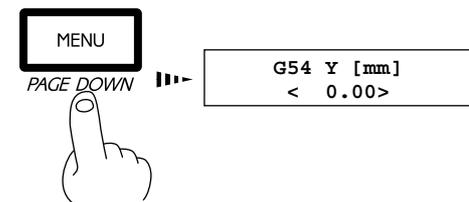
- 4** Use the [▲] or [▼] key to set the amount of shift of the X axis to 50 mm.



- 5** Press the [ENTER] key.



- 6** Press the [PAGE DOWN] key once, and set the amount of shift of the Y axis to 50 mm in the same way.

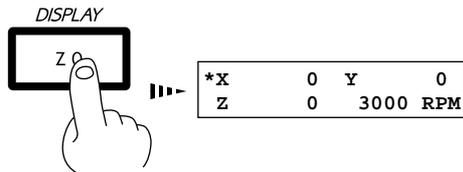


- 7** Set the amount of shift of the Z axis in the same way.

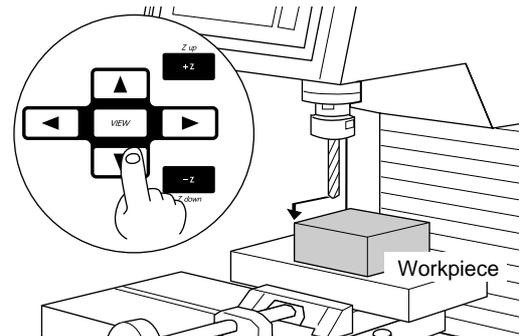
Setting the Current Tool Location As the Workpiece Coordinate System (G54 to G59) Origin Point

This explanation assumes that the front-left corner of the workpiece is to be set to G54 (workpiece coordinate system 1). The EXOFS is assumed to be "0" (zero). The settings for G56 to G59 are made in the same way. A similar procedure is also used when setting the origin point to a desired location.

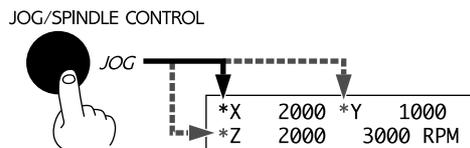
- 1 Press the [DISPLAY] key to make the following screen appear on the display.



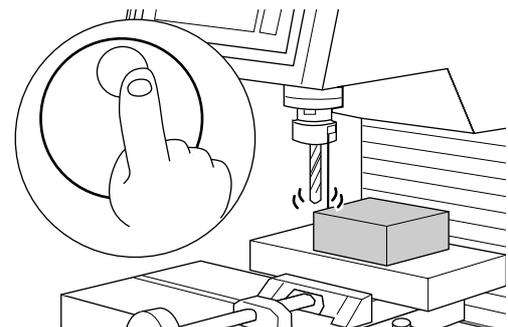
- 2 Press the arrow keys and the Z up/down keys to move the cutting tool to a position close to the front left corner of the workpiece.



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

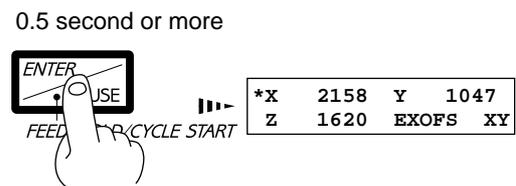


- 4 Use the jog handle to move the tool a little at a time.

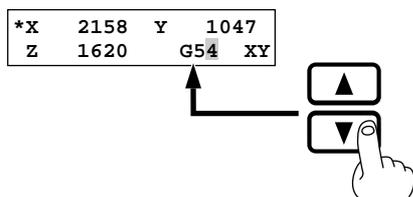


- 5 Repeat steps 3 and 4 to align the tool at the front-left corner of the material to be machined.

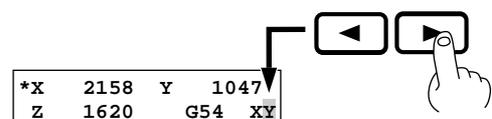
- 6 Hold down the [ENTER] key for 0.5 second or more.



- 7 Press the [▲] or [▼] key to display "G54."

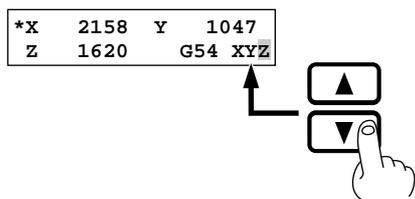


- 8 Press the [▶] key to move the blinking cursor ("█") to the displayed letter for the axis.



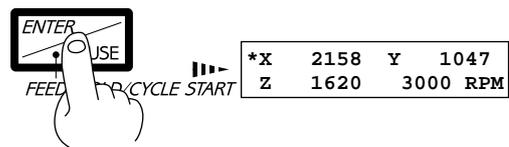
9

Press the [▲] or [▼] key to display “XYZ.”



10

Press the [ENTER] key. The display flashes once, then returns to displaying the coordinate values.



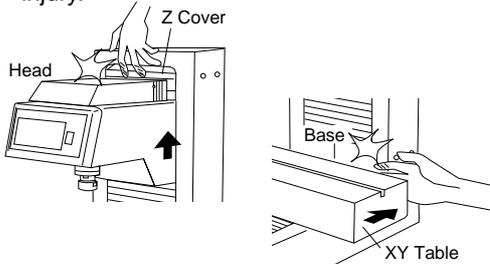
* The distance between the EXOFS and the tool location is set to the amount of shift. In this case, the “G54 X,” “G54 Y,” and “G54 Z” amounts of shift are changed.

3 Sending Cutting Data

⚠ CAUTION

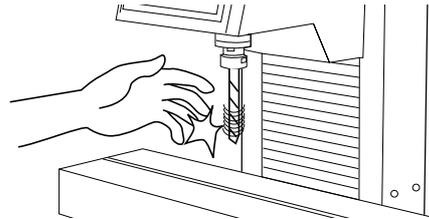
Do not insert the fingers between the XY table and base or between the head and Z cover.

The fingers may be pinched, resulting in injury.



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 an attempt is made by mistake to perform cutting beyond capacity, immediately switch off the spindle switch.



To keep cutting waste from being scattered, we recommend using a commercially available vacuum cleaner to take up cutting waste during cutting. When attaching a vacuum cleaner to the PNC-3200, use a brush adapter for chip cleaning (ZAD-10/ZAD-20), sold separately.

Selecting the Character Code

The PNC-3200 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 - Submenus" and make the setting for the character code.

For more information about parity errors, see "Error Messages."

Cutting Procedure

There are two methods of processing cutting data.
Before sending the cutting data, choose the processing method.

1. Performing Cutting As Data Is Received, Without Saving the Sent Data

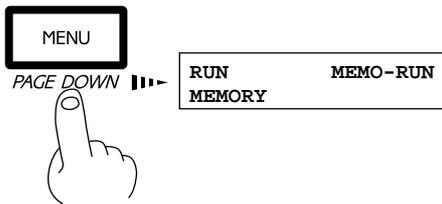
Cutting is performed while the cutting data is being received. The data is not saved in the buffer (temporary memory).
If the amount of cutting data exceeds the PNC-3200's buffer size, choose this processing method.
It is also a good idea to choose this method when you don't know the amount of cutting data to be sent.

2. Saving the Sent Cutting Data in the Buffer, Then Calling Up the Data and Performing Cutting

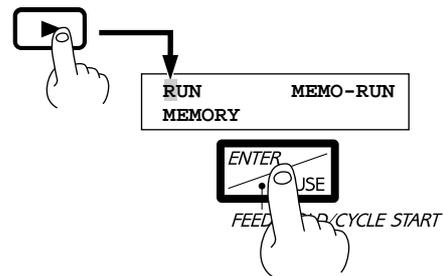
Save the cutting data in the PNC-3200's buffer, then use the panel to choose the data to execute.
If the cutting data includes subprograms, choose this processing method. Choosing the other method may result in an error during cutting. (For more information about subprograms, see the NC Code Programmer's Manual.)
It is also a good idea to choose this processing method when executing the same cutting data a number of times, or when you don't know whether the cutting data includes subprograms.
The saved data disappears when it is overwritten, when the unit is switched to "RUN," or when the power is switched off.

Performing Cutting As Data Is Being Received

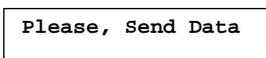
- 1** Press the [PAGE DOWN] key to make the following screen appear on the display.



- 2** Press the [▶] key to move the blinking cursor ("█") to "RUN," and then press the [ENTER] key.



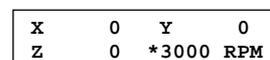
- 3** The display shows the following message.



Send the data from the computer.
The data is output using a text file entered with NC codes.

For more information about programming with NC codes, see the PNC-3200 NC Code Programmer's Manual.

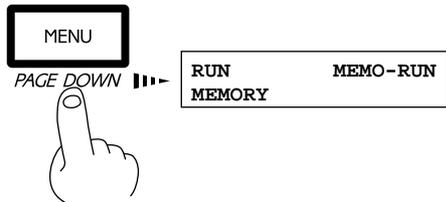
- 4** Cutting starts and the display changes to show the following screen.



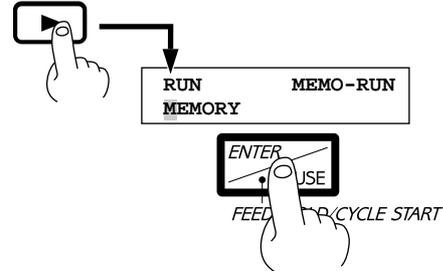
Saving Cutting Data in the Buffer, Then Calling Up the Data and Performing Cutting

Save the data in the buffer.

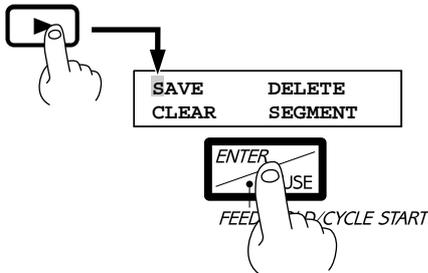
- 1** Press the [PAGE DOWN] key to make the following screen appear on the display.



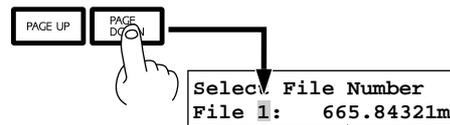
- 2** Press the [▶] key to move the blinking cursor ("█") to "MEMORY," and then press the [ENTER] key.



- 3** Press the [▶] key to move the blinking cursor ("█") to "SAVE," and then press the [ENTER] key.

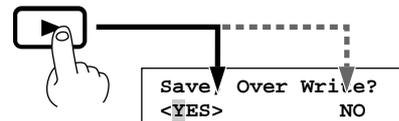


- 4** Press the [PAGE DOWN] or [PAGE UP] key to choose the storage area (data number) for the cutting data.

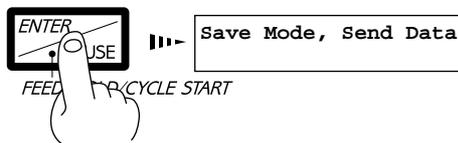


Check the tape length (amount of data) for which storage is possible.

* If data has already been saved, the following message appears. Select "Yes" to overwrite the old data, or "No" not to overwrite it.



- 5** Press the [ENTER] key. Operate the computer to send the cutting data.



- 6** When the data-receiving process has finished, the following message appears on the display. Press any key at this time.



This is the tape length (amount of data) that has been stored

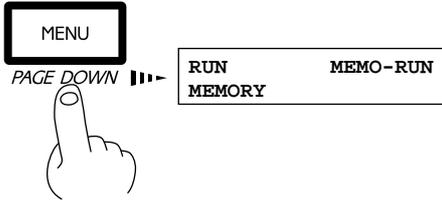
* If the program does not end with %, press the [CANCEL] key to call up this display.

If there is not enough space in the storage area to store all of the data, the following message appears when the storage space runs out. If this happens, stop sending data and change the amount of data space (see "Descriptions of the Display Menus").

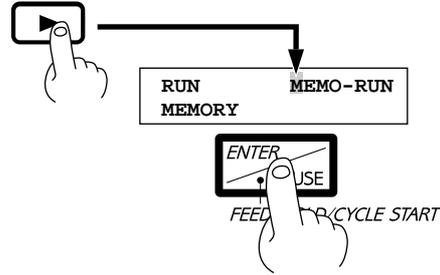
Memory Full!
Stop Sending Data

Perform cutting

- 1 Press the [PAGE DOWN] key to make the following screen appear on the display.



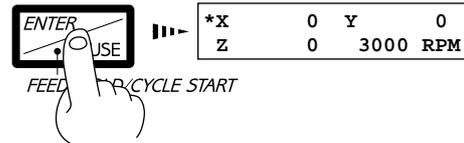
- 2 Press the [▶] key to move the blinking cursor ("█") to "MEMO-RUN," and then press the [ENTER] key.



- 3 Use the [PAGE DOWN] or [PAGE UP] key to choose the cutting data (data number) to execute.

- 4 Press the [ENTER] key. The display changes to show the coordinates, and cutting starts.

* When the spindle has been subjected to an excessive load or when the spindle switch is off, operation stops immediately.

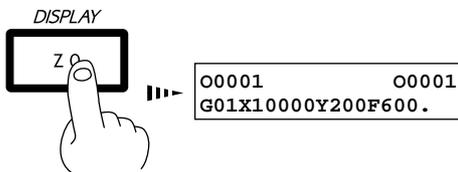


Settings Affecting Operation While Cutting

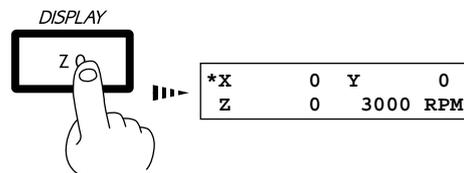
Checking the Code Being Executed on the Display

The block currently being executed can be made to appear on the display.

- 1 Press the [DISPLAY] key while operation is in progress.



- 2 Pressing the [DISPLAY] key again returns to the normal display of the coordinate values.

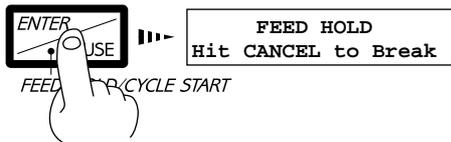


Pausing Operation and Adjusting the Rotating Speed and Feed Speed

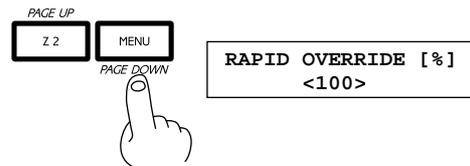
The programmed rotating speed and feed speed can be adjusted while cutting is in progress. Below is a list of the items that can be adjusted.

Item	Description
Positioning override	This sets the G00 operating speed, with the maximum speed of the PNC-3200 (1,800 mm/min.) 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.
Restriction of cutting speed	This sets an upper limit for cutting speed. Once set, this upper limit is not exceeded, even if a faster cutting speed is specified.
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.

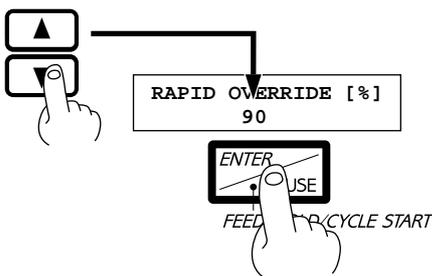
1 Press the [ENTER] key during operation. The cutting operation currently in progress is performed once, then operation stops. The display changes to show the following message.



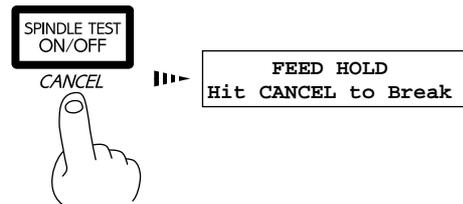
2 Press the [PAGE UP] or [PAGE DOWN] key to display the desired item.



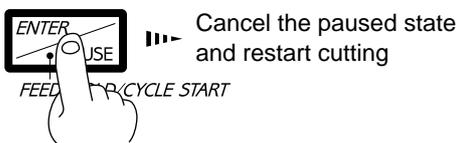
3 Use the [▲] or [▼] key to change the value, then press the [ENTER] key. To change other items, repeat steps 2 and 3.



4 Press the [CANCEL] key once.



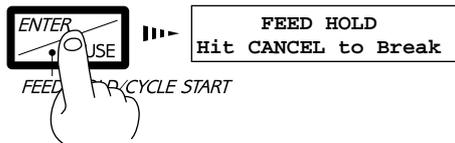
5 Press the [ENTER] key. Cutting is restarted from the operation following the pause.



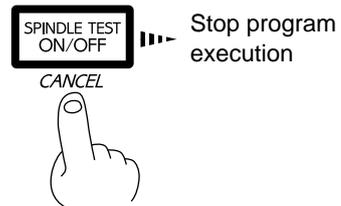
Terminating Cutting

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** Press the [ENTER] key during operation. The cutting operation currently in progress is performed once, then operation stops. The display changes to show the following message.



- 2** Press the [CANCEL] key. Execution of the program stops.



4 Finishing

⚠ CAUTION



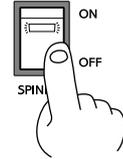
Do not touch the tip of the blade with your fingers.

Doing so may result in injury.



When mounting or removing a tool, first switch off the spindle switch.

Doing so may result in injury.



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.

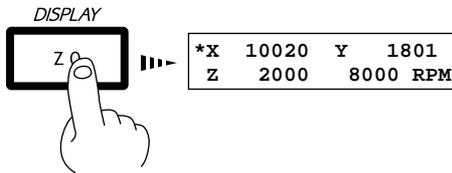


Please use a vacuum cleaner to remove cutting dust.

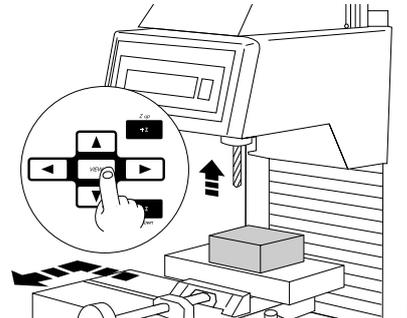
Do not use any blower like airbrush. Otherwise, dust spread in the air may harm your health.

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

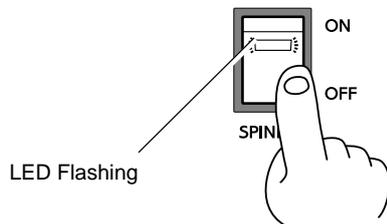
- 1 Press the [DISPLAY] key to make the following screen appear on the display.



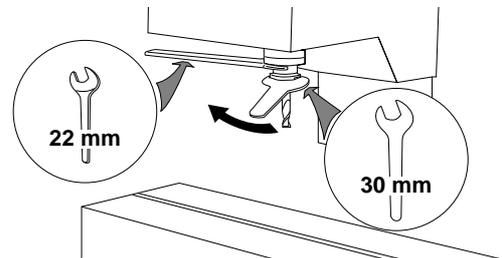
- 2 Press the [VIEW] key for at least 0.5 seconds.



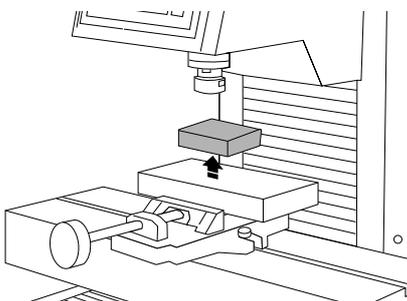
- 3 Turn off the spindle switch on the front of the unit.



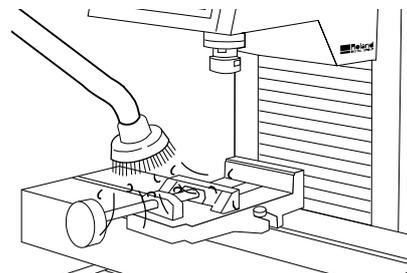
- 4 Open the cover and detach the tool.



- 5 Remove the material.



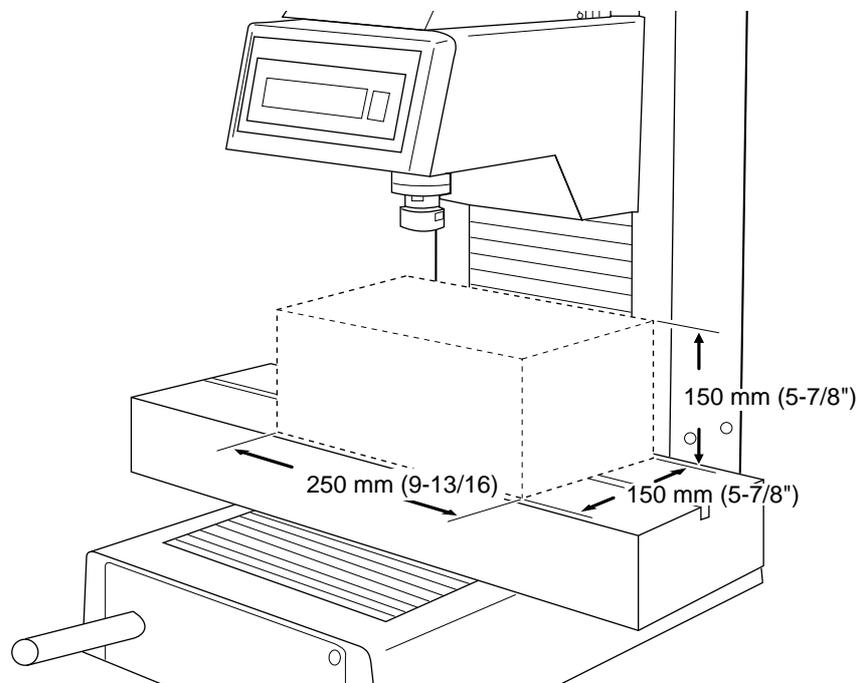
- 6 Use a commercially available vacuum cleaner to remove chips inside the box.



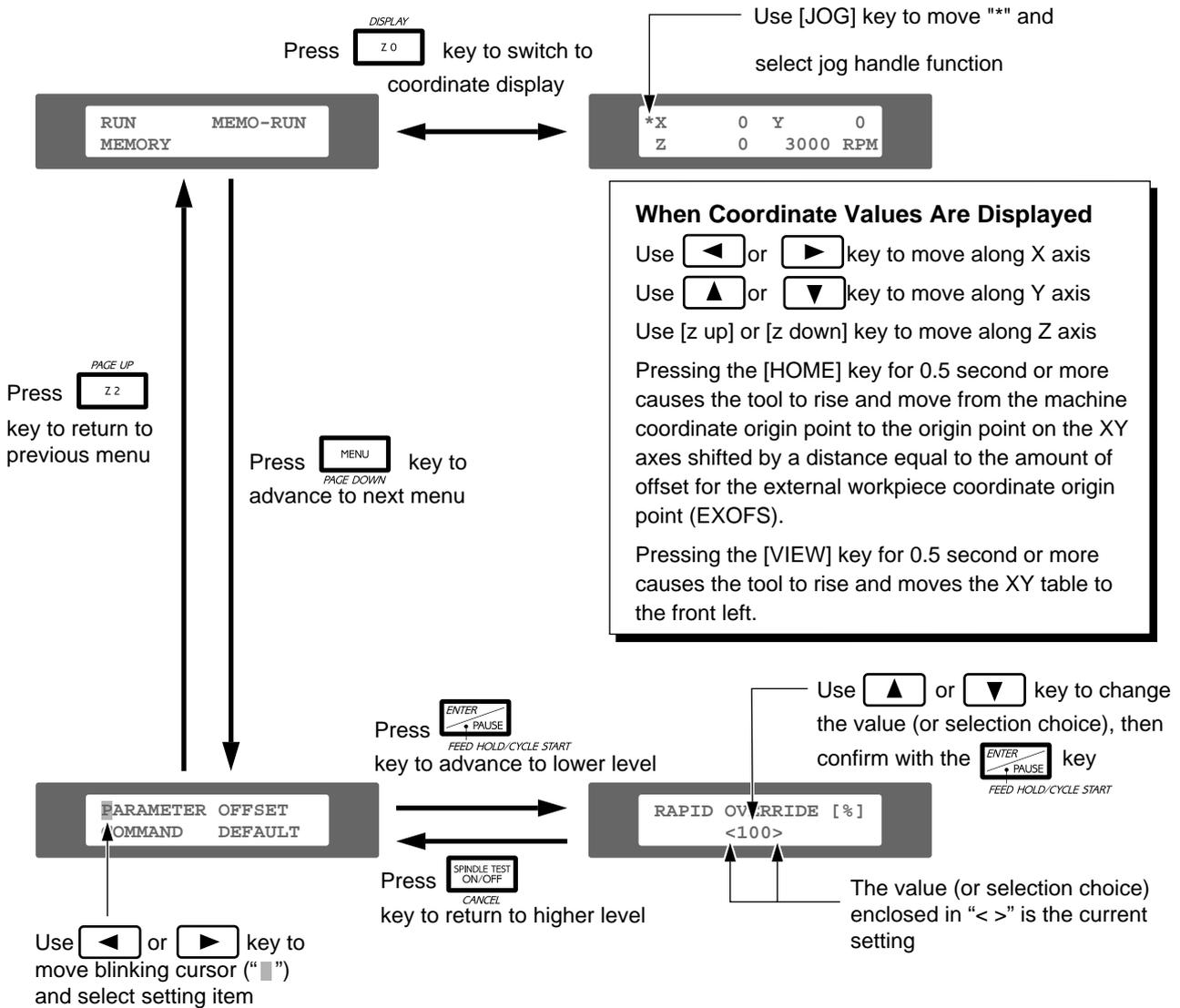
Part 2 *User's Reference*

Cutting Area

The maximum cutting area of the PNC-3200 is 250 mm × 150 mm × 150 mm ("9-13/16" × 5-7/8" × 5-7/8"). The actual available cutting area is subject to restrictions according to the length of the attached tool, the X table position at which the workpiece is fixed, and the vice height (in the case that the vice is used); and in some cases it may be larger than the maximum operating area.



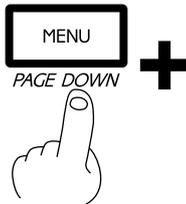
Operations Using the Display



Changing to Other-language Messages on the Liquid-crystal Display

- 1** Switch on the power while holding down the [PAGE DOWN] key.

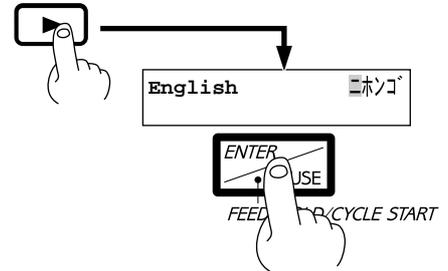
While pressing the
PAGE DOWN key



Turn the power on



- 2** Press the [▶] key to move the blinking cursor (“■”) to “ニホンゴ” and then press the [ENTER] key.



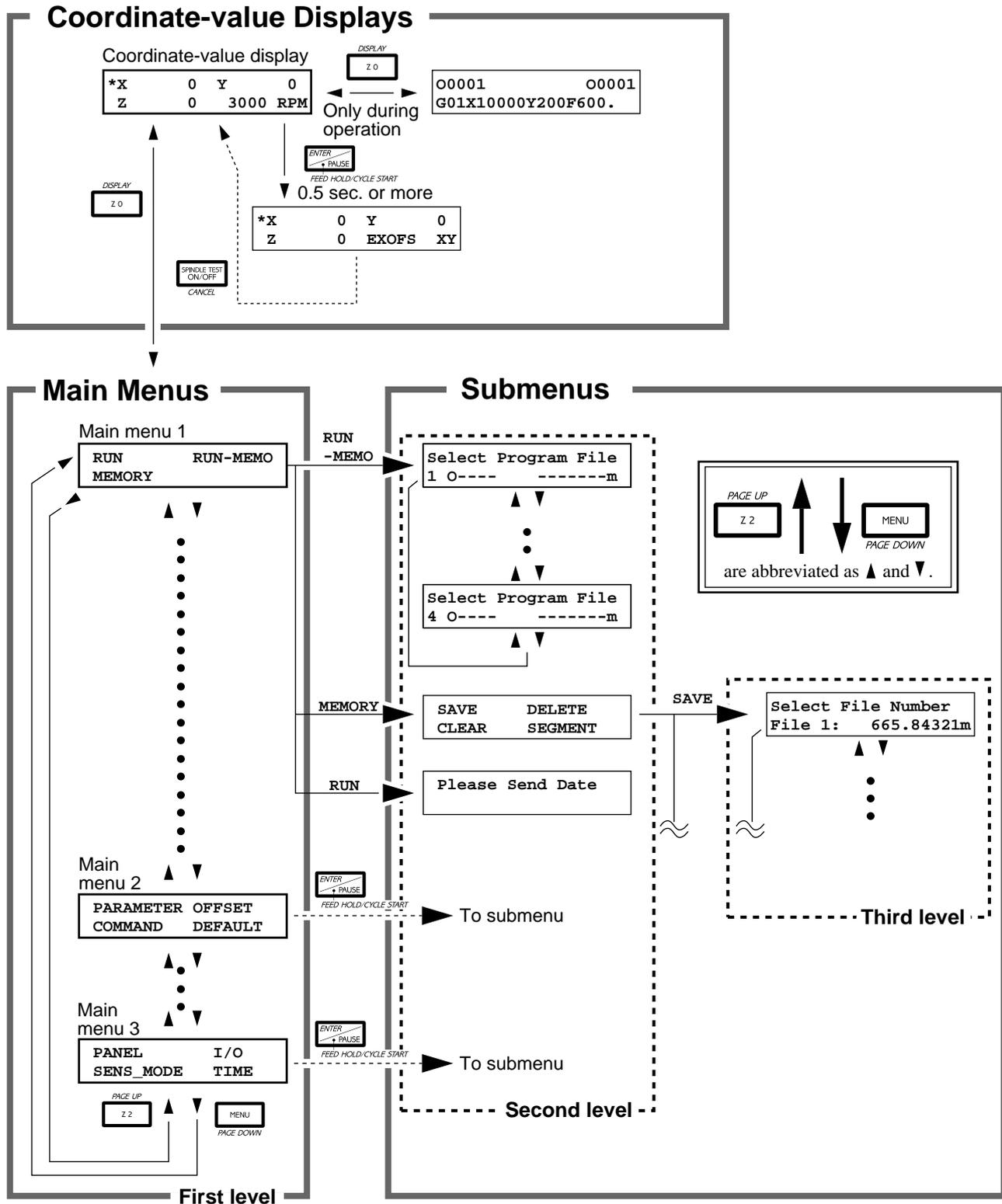
- 3** Messages on the display now appear in Japanese.

ENTERキー	ヲ	オンテクタ [*]	サイ
RML-1		NC-CODE	

- * 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 cursor to “English” and press the [ENTER] key.

Descriptions of the Display Menus

The display menus on the PNC-3200 are arranged in a hierarchical format. The menus can be broadly grouped as coordinate-value displays, main menus, and submenus (menus under the main menus). A partial flowchart of the display menus is shown below.



The coordinate-value displays, main menus, and submenus are described separately.

Coordinate-value Displays

*X	0	Y	0
Z	0	3000	RPM

Description

This displays the current tool location and spindle speed.

When No Program Is Being Executed

Coordinate values are displayed with the point shifted from the machine coordinate origin point by a distance equal to the EXOFS as the origin point.

This display can be used to move the XY table or tool, and to initialize the rotating speed.

The [◀] and [▶] keys are used to move along the X axis.

The [▲] and [▼] keys are used to move along the Y axis.

The [Z up] and [Z down] keys are used to move along the Z axis.

The [JOG] key is used to move the “*” and select the function of the jog handle.

When moved to X, Y, or Z Inching along the selected axis is performed.

When moved to the speed value The speed is initialized. Change the speed with the jog handle, then hold down the [ENTER] key for 0.5 second or more to set the speed. This is the same as the speed set with “SPINDLE SPEED” .

Holding down the [HOME] key for 0.5 second or more causes the tool to rise and move to the EXOFS origin point of the X and Y axes.

Holding down the [VIEW] key for 0.5 second or more causes the tool to rise and moves the XY table to the front left.

Holding down the [ENTER] key for 0.5 second or more causes the display to change to the setting menu for the workpiece coordinate system.

Holding down the [SPINDLE] key for 0.5 second or more causes the spindle to start rotating (However, the spindle does not rotate when the spindle switch is off.) Pressing this while the spindle is rotating stops the rotation.

When Program Execution Is in Progress

Coordinate values are displayed with the point shifted from the machine coordinate origin point by a distance equal to the EXOFS as the origin point.

Pressing the [DISPLAY] key causes the block being executed to appear.

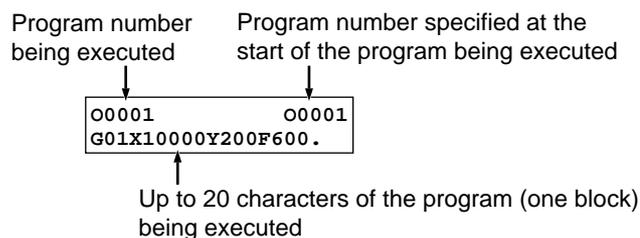
Pressing the [ENTER] key causes operation to pause. The feed rate and spindle speed can be adjusted while paused.

The speed can be varied at all times using the jog dial.

O0001	O0001
G01X1000Y200F600.	

Description

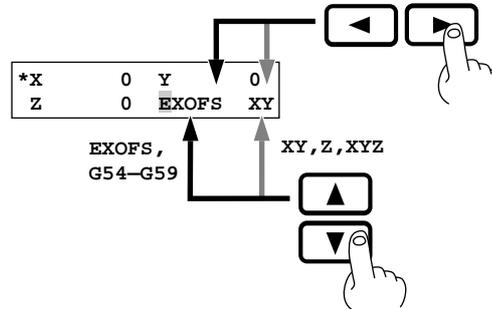
This screen is displayed only when program execution is in progress. It displays the block which is currently being executed.



*X	0	Y	0
Z	0	EXOFS	XY

Description

When a program is not being executed, holding down the [ENTER] key for 0.5 second or more while at the coordinate-value display causes this screen to be displayed. An EXOFS value and workpiece coordinate systems for G54 to G59 can be set. This is done by first moving the tool while in the coordinate-value display, then setting that point to a workpiece coordinate system or EXOFS. Movement of the XY table and tool is not possible.



Main Menus

RUN	RUN-MEMO
MEMORY	

Description

“RUN”

The screen changes to the submenu for cutting while cutting data is being received.

“RUN-MEMO”

This changes to the submenu for calling up and executing stored cutting data.

“MEMORY”

This changes to the submenu for storing cutting data, deleting stored cutting data, and allocating data capacity.

PARAMETER	OFFSET
COMMAND	DEFAULT

Description

“PARAMETER”

This changes to the submenu for setting a variety of parameters, including feed rate, workpiece coordinate system, and circular interpolation.

“OFFSET”

This changes to the submenu for setting cutter compensation.

“COMMAND”

This changes to the submenu for setting the execution location (end or start of block) for F, S, and M codes.

“DEFAULT”

This changes to the submenu for setting the default values for code groups.

PANEL	I/O
SENS_MODE	TIME

Description

“PANEL”

This changes to the submenu for setting single block, machine lock, optional stop, and optional block skip.

“I/O”

This changes to the submenu for setting the program character code and the interface with the computer.

“SENS_MODE”

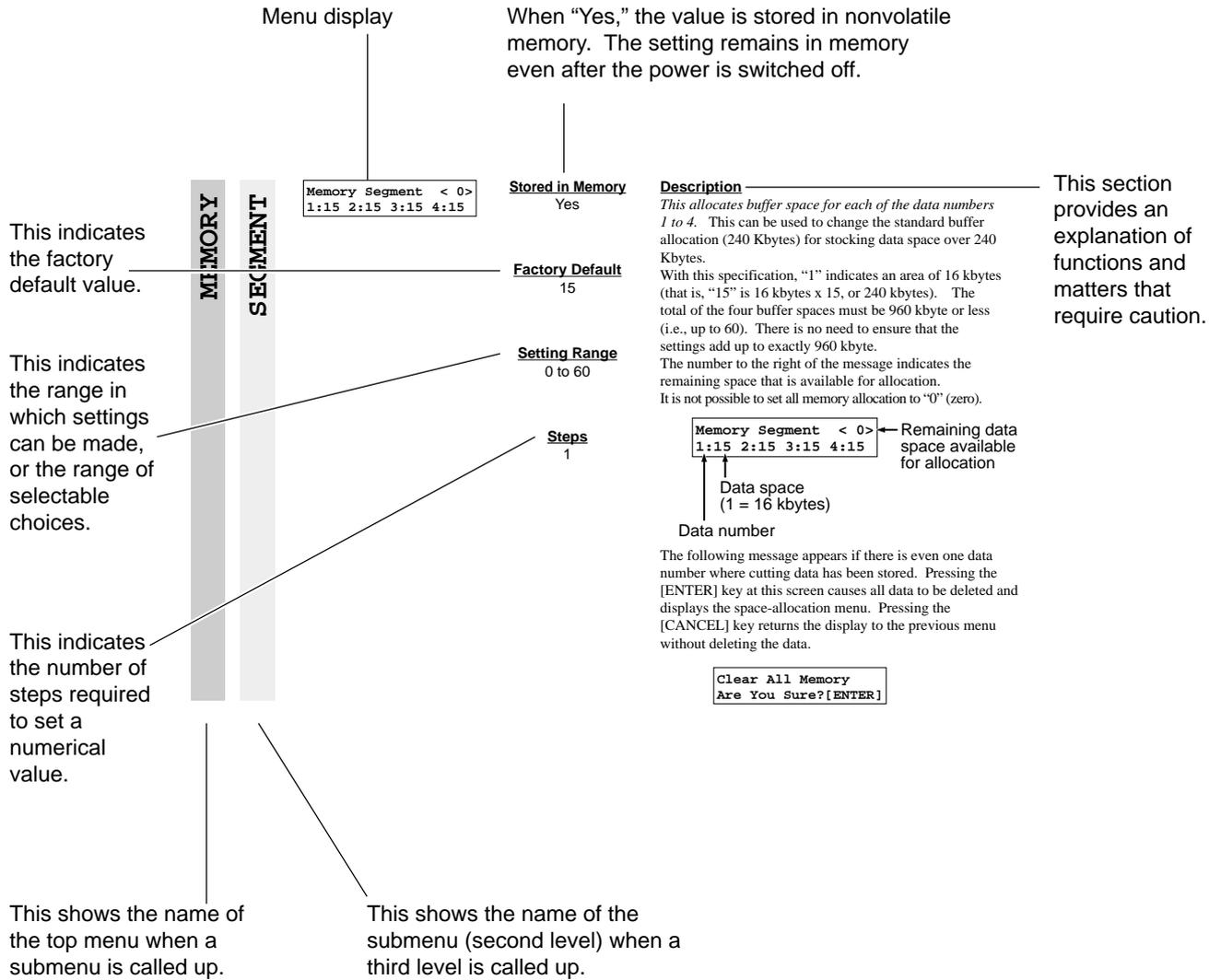
This changes to the mode for using the Z0 sensor.

“SPINDLE”

Displays the total rotation time of the spindle.

Submenus

How to Read the Descriptions of Submenus



RUN

Please, Send Data

Stored in Memory
No
Factory Default
None
Setting Range
None

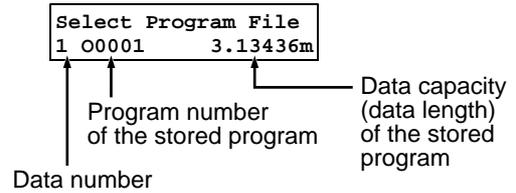
Description
Cutting is performed as the cutting data (NC program) is being received.

Select Program File
1 0---- -m

Select Program File
4 0---- -m

Stored in Memory
No
Factory Default
None
Setting Range
None

Description
This calls up cutting data (an NC program) stored in the PNC-3200's buffer (temporary memory), and performs cutting. The unit of the displayed data capacity (data length) varies according to the setting for "UNIT inch/metric."



MEMORY

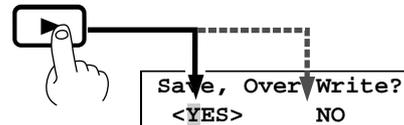
SAVE

Select File Number
File 1: 665.84321m

Select File Number
File 4: 665.84321m

Stored in Memory
No
Factory Default
None
Setting Range
None

Description
This sends data from the computer or NC-code input device and stored the data in the PNC-3200's buffer (temporary memory). Choose the area (data number) where the cutting data is to be stored, press the [ENTER] key, then send the data. If an attempt is made to save new cutting data at a location (data number) where data has already been saved, the following message appears. Selecting "Yes" saves the new data, overwriting the existing data.



The unit of the displayed data capacity (data length) varies according to the setting for "UNIT inch/metric."

DELETE

Hit ENTER to Delete
1 0---- -m

Hit ENTER to Delete
4 0---- -m

Stored in Memory
No
Factory Default
None
Setting Range
None

Description
This allows a data number to be specified to deleted the corresponding cutting data (program) stored in the buffer.

Once data has been deleted, it cannot be restored -- it must be sent from the computer again. To delete all data stored in the buffer, choose "CLEAR." The unit of the displayed data capacity (data length) varies according to the setting for "UNIT inch/metric."

CLEAR

Clear All Memory
Are You Sure?[ENTER]

Stored in Memory
No
Factory Default
None
Setting Range
None

Description
This deletes all cutting data (programs) stored in the buffer. Pressing the [ENTER] key causes the buffer to be cleared. If you do not wish to delete all data, press the [CANCEL] key.

```
Memory Segment < 0>
1:15 2:15 3:15 4:15
```

Stored in Memory

Yes

Factory Default

15

Setting Range

0 to 60

Steps

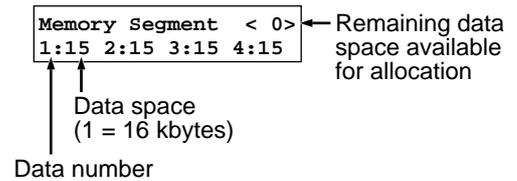
1

Description

This allocates buffer space for each of the data numbers 1 to 4. This can be used to change the standard buffer allocation (240 kbytes) for stocking data space over 240 kbytes.

With this specification, "1" indicates an area of 15 kbytes (that is, "15" is 16 kbytes x 16, or 240 kbytes). The total of the four buffer spaces must be 960 kbyte or less (i.e., up to 60). There is no need to ensure that the settings add up to exactly 960 kbyte.

The number to the right of the message indicates the remaining space that is available for allocation. It is not possible to set all memory allocation to "0" (zero).



The following message appears if there is even one data number where cutting has been stored. Pressing the [ENTER] key at this screen causes all data to be deleted and displays the space-allocation menu. Pressing the [CANCEL] key returns the display to the previous menu without deleting the data.

```
Clear All Memory
Are You Sure?[ENTER]
```

```
RAPID OVERRIDE [%]
<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 PNC-3200 is set at maximum speed (100%, or 1,800 mm/min.). 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.

You can also make the setting while paused or during single-block execution.

```
CUTTING OVERRIDE [%]
<100>
```

Stored in Memory

Yes

Factory Default

100

Setting Range

0 to 200

Steps

1

Description

This adjusts the cutting speed on the PNC-3200. 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.

If the upper limit for cutting speed is exceeded by a speed ratio that has been set, the setting for the limit speed is used. The limit speed can be set using the PNC-3200's display (see the following page). The setting for the limit speed when shipped from the factory is the maximum speed of the PNC-3200.

You can also make the setting while paused or during single-block execution.

CUT SPEED [mm/min] <300>

Stored in Memory

Yes

Factory Default

300

Setting Range

0 to 1800

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.

You can also make the setting while paused or during single-block execution. 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.

The displayed speed is in units of mm/min.

LIMIT SPEED [mm/min] <1800>

Stored in Memory

Yes

Factory Default

1800

Setting Range

0 to 1800

Steps

60

Description

This sets the upper limit for cutting speed. This setting ensures that the cutting speed of the PNC-3200 does not exceed a certain value. It is used to help prevent breakage of the tool during cutting.

For example, this setting can be used to restrict cutting speed and avoid tool breakage in cases such as when using a material or tool other than that assumed by the program. It can also be used to protect the tool when a faster cutting speed than intended has mistakenly been specified.

The setting is stored in memory and affects all programs. Because a cutting speed which should be specified by programming may not actually be achieved, it is advisable to return the value to the maximum speed after the desired cutting has been performed.

This setting does not affect the speed of positioning.

You can also make the setting while paused or during single-block execution.

The displayed speed is in units of mm/min.

SPINDLE RPM [RPM] <3000>

Stored in Memory

Yes

Factory Default

3000

Setting Range

3000 to 8000

Steps

100

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.

You can also make the setting while paused or during single-block execution. 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.

DEFAULT SCALE (G51) <1.00>

Stored in Memory

Yes

Factory Default

1.00

Setting Range

0.01 to 100.00

Steps

0.01

Description

The scaling factor specified by this setting is used when no scaling factor has been specified by programming (G51).

This setting can be used to set the scaling factor on the PNC-3200 without altering programming. It is used in order to set scaling on the PNC-3200 without making a scaling setting with the program.

The setting is stored in memory and affects all programs. For this reason, executing a subsequent program with the scaling setting remaining at its altered value may result in cutting at the wrong scale. It is advisable to return the scaling value to 1.00 when the desired cutting has finished.

G54 X [mm]
<0.00>

G59 Z [mm]
<0.00>

Stored in Memory

Yes

Factory Default

0.00

Setting Range

-300.00 to 300.00

Steps

0.01

Description

This sets workpiece coordinate systems for G54 to G59. The origin point for each axis is set as the distance from the EXOFS (external workpiece origin point amount of offset). (See 2 Setting a Workpiece Coordinate System—Setting a Workpiece Coordinate System Using G54 to G59.)

A workpiece coordinate system can also be set by pressing the [ENTER] key at the coordinate-value display (see Setting the Amount of Shift from the EXOFS As a Numerical Value).

This displayed length is in unit of mm.

EXOFS X [mm]
<0.00>

EXOFS Z [mm]
<0.00>

Stored in Memory

Yes

Factory Default

0.00

Setting Range

-300.00 to 300.00

Steps

0.01

Description

This sets the external workpiece origin point amount of offset (EXOFS). The amount of shift for each axis is set as the distance from the machine coordinate origin point. (See 2 Setting a Workpiece Coordinate System—Setting a Workpiece Coordinate System Using G54 to G59.)

The workpiece coordinate systems for G54 to G59 are each set as the distance from the EXOFS. This means that changing the EXOFS causes the workpiece coordinate systems for G54 to G59 to change.

This displayed length is in unit of mm.

CHORD TOLERANCE [°]
<1.0>

Stored in Memory

Yes

Factory Default

1.0

Setting Range

0.1 to 5.0

Steps

0.1

Description

This sets the resolution for circular interpolation.

The PNC-3200 cuts circles and arcs as a continuous series of small lines. The resolution sets the center angle for these small lines.

Making the resolution smaller results in smooth curves, but because the number of lines increases, considerable time is required for cutting. Using a larger resolution reduces the time needed to cut circles and arcs, but the results of cutting are visibly rough.

THICKNESS [mm] <*. **>

Stored in Memory
Yes

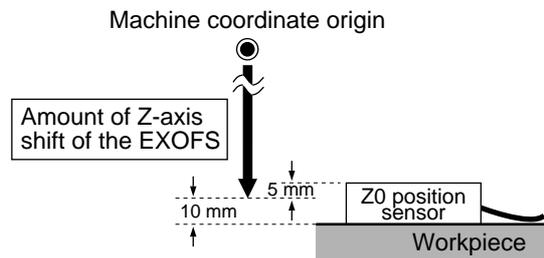
Factory Default
15.00

Setting Range
0.00 to 30.00

Steps
0.01

Description

This sets the thickness of the Z0 sensor. The actual measured value when shipped from the factory is entered for the thickness of the included Z0 sensor. 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 amount of Z-axis shift of the EXOFS (see 2 Setting a Workpiece Coordinate System—Setting a Workpiece Coordinate System Using G54 to G59). The amount of Z-axis shift of the EXOFS can be set to a location which is a fixed distance from the location where the Z0 sensor is placed. In this explanation, the thickness of the Z0 sensor is assumed to be 15.00 mm. 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 10 mm from the surface, the thickness should be set to a value of “5.00”.



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 displayed length is in units of mm.

DISPLAY <COORDINATE>PROGRAM

Stored in Memory
Yes

Factory Default
COORDINATE

Selection Choices
COORDINATE,
PROGRAM

Description

This selects whether the panel shows the coordinate-value display or program code when a program is executed. The panel display is selected immediately after program execution. The [DISPLAY] key can also be used to toggle between coordinate display and code display after program execution. It may be convenient to make the setting for code display when the single-block mode has been selected.

Coordinate-value display

*X	0	Y	0
Z	0	3000	RPM

Code display

O0001	O0001
G01X10000Y200F600.	

OFFSET TYPE
<TYPE-A> TYPE-B

Stored in Memory
Yes

Factory Default
TYPE-A

Selection Choices
TYPE-A, TYPE-B

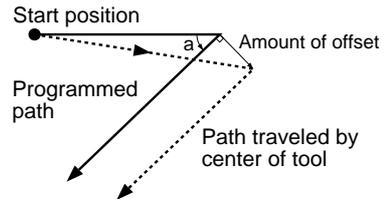
Description

This selects the program path for outer-side travel when starting or ending cutter compensation.

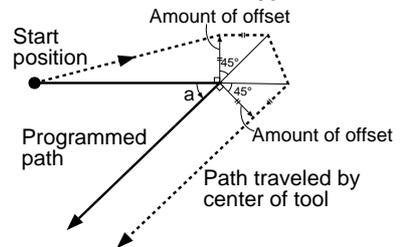
A case of shifting from a line to another line with travel on the outer side of an acute angle when cutter compensation starts is given as an example. For more information, refer to the NC Code Programmer's Manual.

Outer-side Acute Angle (a < 90°)

From a line to a line -- Type A



From a line to a line -- Type B



S CODE
<RPM> 2-FIGURES

Stored in Memory
Yes

Factory Default
RPM

Selection Choices
RPM,
2-FIGURES

Description

This selects whether values specified by S-codes are units of speed (rpm) or numeric codes assigned to speeds. Values are interpreted according to the setting made with this menu, regardless of the size of the specified value.

For more information, refer to the NC Code Programmer's Manual.

TOOL DIAMETER OFFSET
<D-CODE> H-CODE

Stored in Memory
Yes

Factory Default
D-CODE

Selection Choices
D-CODE,
H-CODE

Description

This sets the type of code for specifying the amount of offset of cutter compensation.

The amount of offset for cutter compensation is generally specified with D-codes.

Some applications may use H-codes to set the amount of offset for cutter compensation. There are also some cutting-data programs which use H-codes to set the amount of offset for the tool radius. In cases such as these, the setting should be changed to "H-CODE."

TOOL CHANGE
<VALID> IGNORE

Stored in Memory
Yes

Factory Default
VALID

Selection Choices
VALID, IGNORE

Description

This specifies the operation that occurs when M06 (tool-change command) is executed.

"VALID" Operations are paused when M06 is executed.

"IGNORE" When M06 is executed, operations are not paused in the M06 command is ignored.

DECIMAL POINT INPUT <NORMAL> CALCULATOR
--

Stored in Memory

Yes

Factory Default

NORMAL

Selection ChoicesNORMAL,
CALCULATOR**Description**

This selects how numeric values are interpreted during real-number entry or integer entry.

When “NORMAL” is selected, numeric values for real-number entry are interpreted as millimeters or inches, and numeric values for integer entry are interpreted as the minimum metric or inch units.

When “CALCULATOR” is selected, numeric values for both real-number entry and integer entry are interpreted as millimeters or inches.

When set to “CALCULATOR” the applicable words can be selected (see the following section).

CALCULATOR INPUT <ALL> F IJKRXYZ

Stored in Memory

Yes

Factory Default

ALL

Selection Choices

ALL, F, IJKRXYZ

Description

This is displayed only when “CALCULATOR” has been selected at the “DECIMAL POINT INPUT” menu.

This allows selection of the applicable words for “CALCULATOR.”

“ALL”	Applied to all F, I, J, K, R, X, Y, and Z words
“F”	Applied only to F functions
“IJKRXYZ”	Applied to all words other than F functions (i.e., I, J, K, R, X, Y, and Z words)

OFFSET No. 1 [mm] <0.00>

OFFSET No.10 [mm] <0.00>

Stored in Memory

Yes

Factory Default

0.00

Setting Range

-10.00 to 10.00

Steps

0.01

Description

This specifies the amount of offset set for cutter compensation.

When G41 or G42 (cutter compensation) specifies an offset number which has not been set with G10, the value set on the PNC-3200 is used.

The amount of offset is used by cutter compensation. For example, if an amount of offset of 2 mm is set for offset number 2, operation is as follows.

Cutter compensation: **G00G41D01X5.0**

Offset of 2 mm to the left-hand side relative to the direction of forward movement

If a negative value is specified for the amount of offset, the direction of offset is revised, as shown below. For example, if an amount of offset of -3 mm is set for offset number 1, operation is as follows.

G00G41D01X5.0

Offset of -3 mm to the left-hand side relative to the direction of forward movement = 3 mm offset to the right-hand side

The allowable range for the amount of offset used during cutter compensation is -10.00 to 10.00 mm.

The unit of the displayed length varies according to the setting for “UNIT **inch/metric**.”

F-function in
<PRIOR> POSTERIOR

S-function in
<PRIOR> POSTERIOR

M-function in
<PRIOR> POSTERIOR

Stored in Memory
Yes

Factory Default
PRIOR

Selection Choices
PRIOR,
POSTERIOR

Description

This selects whether F, S, and M functions are activated at the start of the block in which they appear, or after the block.

For example, if “**PRIOR**” is specified for F functions, a cutting movement command in the same block as an F function operates at the feed rate that has been set.

When “**POSTERIOR**” is specified, cutting movement commands in the subsequent block and after operate at the feed rate that has been set.

Operation is similar for S functions and M functions.

The M-function codes to which this setting applies are \ M03, M05, M06, M00 and M01 always operate after the block in which they appear.

MOVE TYPE
<G00>

PLANE
<G17>

UNIT inch/metric
<G21>

WORK COORDINATE
<G54>

Stored in Memory
Yes

Factory Default
G00

Selection Choices
G00, G01, G02, G03

Stored in Memory
Yes

Factory Default
G17

Selection Choices
G17, G18, G19

Stored in Memory
Yes

Factory Default
G21

Selection Choices
G21, G22

Description

This sets the interpolation method for dimension words (X, Y, Z, I, J, K, and R) when specification of an interpolation method is not made by programming (G00, G01, G02, or G03).

Description

This sets the interpolation plane for circular interpolation (G02 or G03) when specification of a plane is not made by programming (G17, G18, or G19).

Description

This specifies the unit for amounts of movement, feed rate, and amounts of offset when a unit setting is not made by programming (G20 or G21).

The units of setting values change on the display menus for length and amount of offset.

Example

G20 (inches)	G21 (metric)
OFFSET No. 1 [Inch] <0.1181>	OFFSET No. 1 [mm] <3.00>

Stored in Memory
Yes

Factory Default
G54

Selection Choices
G54, G55, G56,
G57, G58, G59

Description

This selects the workpiece coordinate system when no selection of workpiece coordinate system is made by programming (G54 to G59).

POS. COMMAND SYSTEM
<G90>

Stored in Memory
Yes

Description

This specifies the setting method for coordinate values when no specification for absolute or incremental is made by programming (G90 or G91).

Factory Default
G90

Selection Choices
G90, G91

RETURN POSITION
<G98>

Stored in Memory
Yes

Description

This sets the return-point level when no specification for fixed-cycle return point is made by programming (G98 or G99).

Factory Default
G98

Selection Choices
G98, G99

SINGLE BLOCK
<OFF> ON

Stored in Memory
Yes

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. You can also make the setting while paused or during single-block execution.

Factory Default
OFF

Selection Choices
ON, OFF

MACHINE LOCK
<OFF> XYZ Z

Stored in Memory
Yes

Description

This setting causes no operations to be performed along the specified axis. Even for a locked axis, however, the coordinate-value display continues to the same values as when actual operations are performed, and the same time is required as when not locked. Movement of the tool using the arrow keys or the [Z up] and [Z down] keys is not affected. This setting can be used to check operations before starting actual cutting. For example, the Z axis alone can be locked and the program executed to check the cutting time and movement of the X and Y axes. The coordinate-value display may not necessarily show the actual tool position immediately after a program has been run with the machine lock activated. Toggling the display causes the correct display to appear.

Factory Default
OFF

Selection Choices
OFF, XYZ, Z

OPTIONAL STOP
<OFF> ON

Stored in Memory
Yes

Description

This setting determines whether M01 is activated or ignored. M01 is activated when set to "ON" and ignored when set to "OFF."

Factory Default
OFF

Selection Choices
OFF, ON

WORK AREA OVER STOP?
<OFF> ON

Stored in Memory

Yes

Factory Default

OFF

Selection Choices

OFF, ON

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 "ON," blocks for which optional block skip is specified are skipped.

WORK AREA OVER STOP?
<OFF> ON

Stored in Memory

Yes

Factory Default

OFF

Selection Choices

OFF, ON

Description

Choose the operation when the tool returns from outside the cutting area to within the area (the tool cannot actually move outside the cutting area, but internal processing is carried out as if it had).

When set to "OFF," operation does not pause when the tool returns to inside the cutting area. Cutting continues without interruption.

When set to "ON," operation pauses when the tool returns to inside the cutting area.

SELECT USE CODE
ASCII ISO EIA<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.

If the character code cannot easily be determined when set to "AUTO," the setting should be made manually.

When a program is received with an incorrect code, an error message may be displayed or unexpected operation may occur.

SELECT I/O PORT
PARA SERIAL <AUTO>

Stored in Memory

Yes

Factory Default

AUTO

Selection Choices

PARA, SERIAL,
AUTO

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.

STOP BIT
<1> 2

Stored in Memory

Yes

Factory Default

1

Selection Choices

1, 2

Description

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

DATA BIT
7 <8>

Stored in Memory

Yes

Factory Default

8

Selection Choices

8, 9

Description

This sets the data bit length for the communication parameters.

PARITY		
<NONE>	ODD	EVEN

Stored in Memory

Yes

Factory Default

NONE

Selection ChoicesNONE, ODD,
EVEN**Description**

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

BAUDRATE	
4800	<9600>

Stored in Memory

Yes

Factory Default

9600

Selection Choices

4800, 9600

Description

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

HANDSHAKE	
<HARDWARE>	XON/XOFF

Stored in Memory

Yes

Factory Default

HARDWARE

Selection ChoicesHARDWARE,
XON/XOFF**Description**

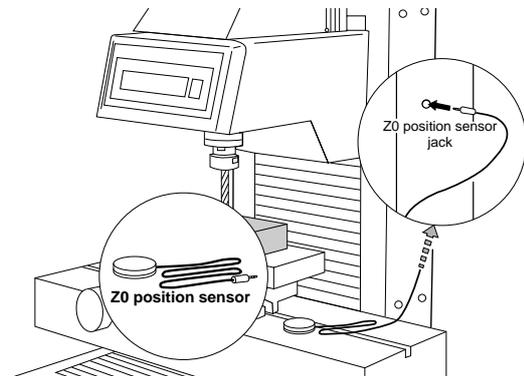
This sets the handshaking communication parameter.

*X	0	Y	0
Z	0	SENS_MODE	

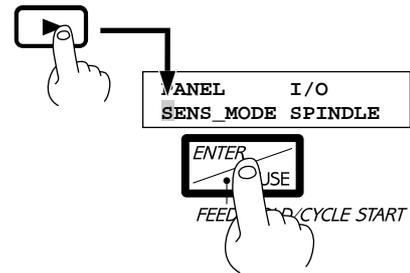
Description

This sets the amount of Z-axis shift for the EXOFS (external workpiece origin point amount of offset) using the Z0 sensor included with the PNC-3200. Changing the amount of shift causes the setting for "EXOFS Z" to change.

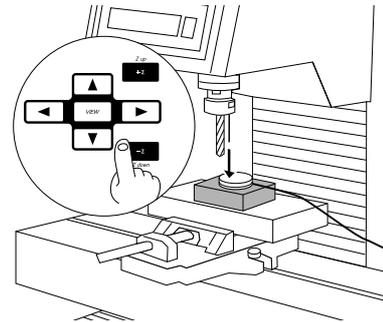
Connect the Z0 sensor to the PNC-3200 before entering "SENS_MODE." An error is generated if the cord is loose or the Z0 sensor is not connected.



Enter "SENS_MODE."



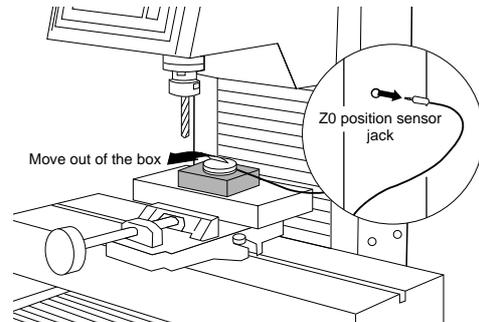
Position the Z0 sensor at the area where the amount of Z-axis shift for the EXOFS is to be set, and move the tool until the tip of the tool touches the Z0 sensor.



When the tool touches the sensor, the following message appears on the display and the amount of shift is set.

Z ORIGIN SET!,UP SP-
INDLE, PUSH "CANCEL"

Raise the tool and remove the Z0 sensor.



Press the [CANCEL] key to end "SENS_MODE."

REVOLUTION TIME
0 hours

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 (For more information on maintenance, see User's Manual 1 -- "Setup and Maintenance.")

What to Do If...

When the PNC-3200 does not work...

Is operation paused?	Cancel the paused state.
Is the power for the PNC-3200 switched on?	Make sure the PNC-3200 is powered up.
Has the machine lock function been activated?	No operation takes place when machine lock has been activated for the X, Y, and Z axes. At this time, only the displayed coordinate values change.
Is cutting data stored in the buffer?	No operation takes place if no cutting data is stored in the buffer. Switching off the power causes all cutting data stored in the buffer to be deleted.
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.

When the spindle does not rotate ...

Is the spindle switch for the PNC-3200 set to "Off?"	Turn on the spindle switch.
--	-----------------------------

Data cannot be sent

Do the PNC-3200's connection parameter settings match the settings for the computer?	In the separate User's Manual 1 -- "Setup and Maintenance," refer to "6 Setting the Connection Parameters" and make the correct settings for the connection parameters.
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.
Is the correct output device set for the application or driver software?	Refer to the manual for the application or driver software to set the output device correctly.

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.
--------------------------------	--

Error Message

An error message appears on the display when any type of error occurs on the PNC-3200.

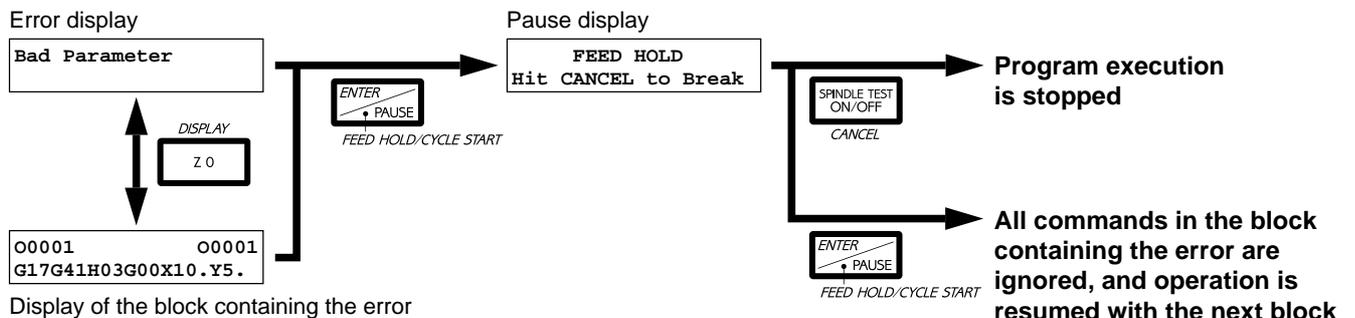
Errors Occurring During Program Execution

When an error occurs, operation pauses and an error message is displayed. When this happens, press the [DISPLAY] key to display the block which contains the error.

The cutting operation can be continued as-is after an error has been displayed, but this is not recommended because operation following the 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 compensation.
Program Number Not found	The program number specified by M98 or M99 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.

<Display Operations When an Error Occurs During Program Execution>

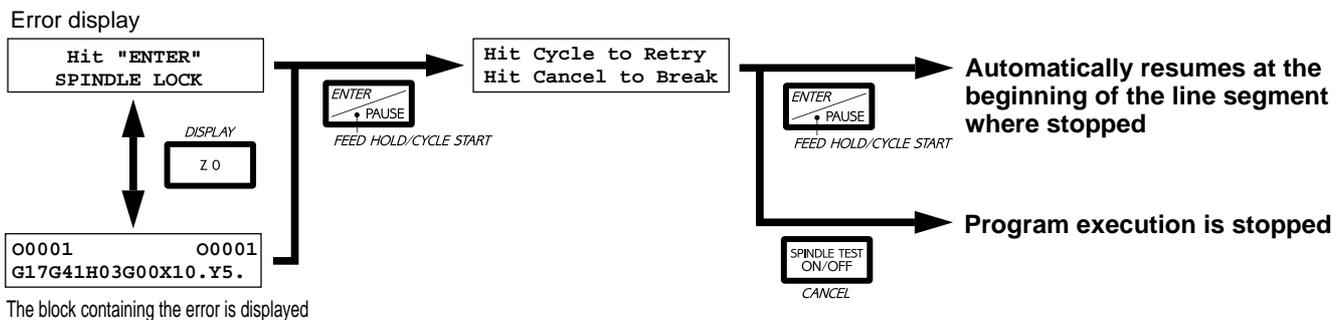


In the following case, the details of the error are displayed and operation stops. If this happens, you can choose to resume cutting or to stop cutting.

Although cutting can be resumed, resuming cutting is not recommended because cutting precision after resuming operation is not guaranteed. When cutting is resumed, the machine origin point on the X, Y, and Z axes is detected again and operation automatically resumes at the beginning of the line segment where it was stopped.

Error message	Description
<div style="border: 1px solid black; padding: 5px; width: fit-content;"> <p>Hit "ENTER" SPINDLE LOCK</p> </div>	<p>The PNC-3200 stops automatically if an excessive load is placed on the spindle motor and speed falls below 1,000 rpm during cutting. If this occurs, the message shown at left appears on the display. Possible causes of such overload may be a material that is too hard, an overly large bite, or a feed rate that is too fast. Check the PNC-3200 and programming and eliminate the cause of the overload.</p> <p>Pressing the [ENTER] key displays a message for choosing whether to resume or stop cutting. Follow the steps below to make your selection.</p>

<Display Operations When an Automatic Stop Occurs During Program Execution>



The block containing the error is displayed

Other Error Messages

Errors in Cutting-data Transmission

These include errors related to communication parameters, as well as errors which may occur when the amount of data exceeds the capacity of the destination for storage.

Error message	Description
<div style="border: 1px solid black; padding: 5px; width: fit-content;"> <p>Framing/Parity Error</p> </div>	<p>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).</p> <p>Stop sending data from the computer, and make the correct settings for the communication parameters.</p> <p>Any data received after this error occurs is ignored and not saved.</p>
<div style="border: 1px solid black; padding: 5px; width: fit-content;"> <p>Memory Full ! Stop Sending Data</p> </div>	<p>The amount of data sent has exceeded the buffer capacity allocated to the data storage area (data number). Data receiving before the error occurred is stored.</p> <p>Stop sending data from the computer, and press the [CANCEL] key. Then either increase the amount space allocated to the buffer or choose a data number with a larger space, and send the data from the computer again.</p>

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 PNC-3200 are received.
Make the setting for the character code again.

Error While Registering Cutting Data

A check of subprograms is carried out when a program is saved in the buffer. Depending on the results of the check, one of the following error messages may be displayed.

Error message	Description
Sub-Program Table Over	There are more than ten subprograms. The maximum number of subprograms that can be specified in one set of data is ten. Stop sending data, and press the [CANCEL] key. Correct the program and 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. Stop sending data, and press the [CANCEL] key. Correct the program and resend the data.

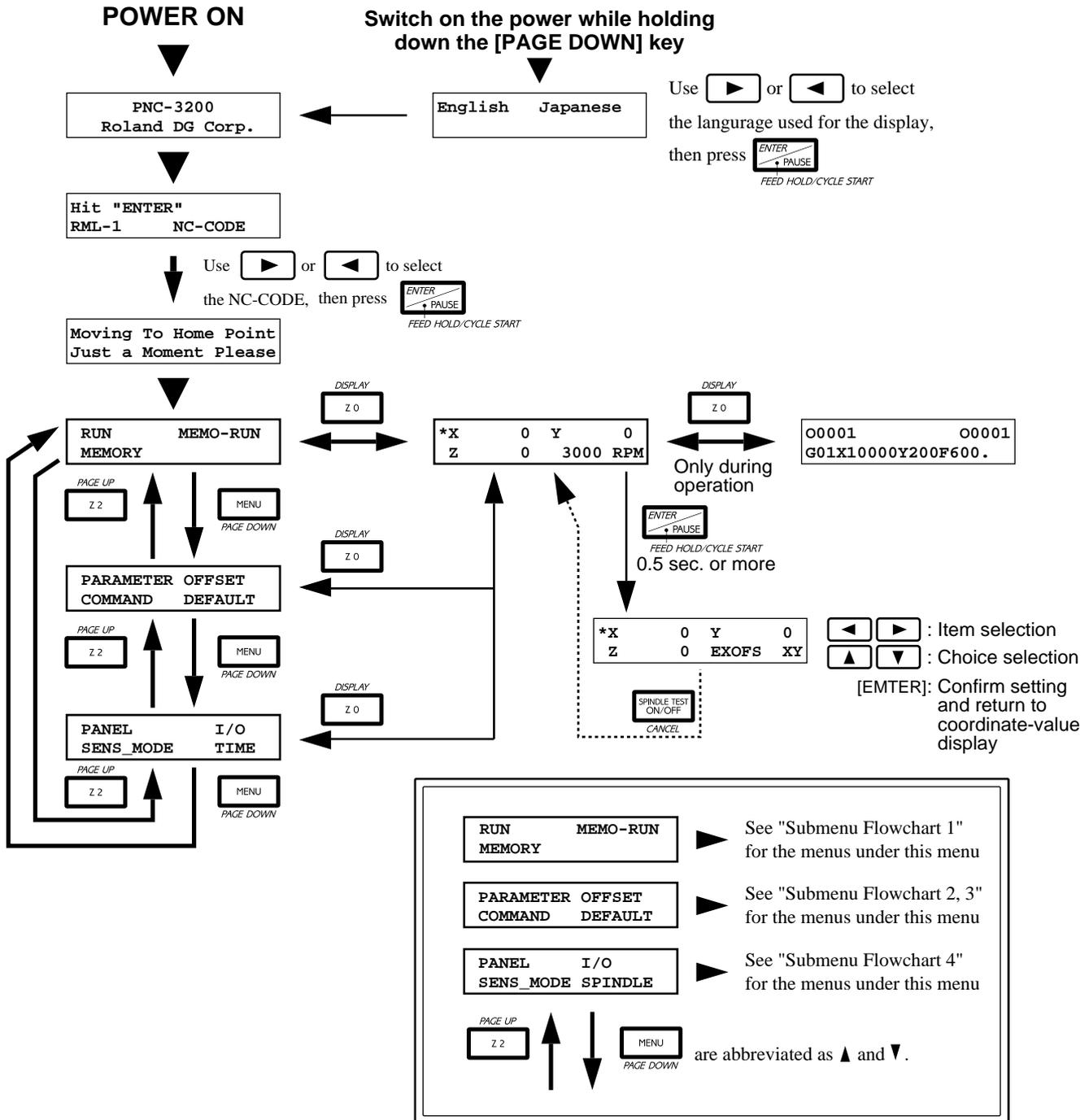
Buffer Space Allocation Error

Error message	Description
Not enough data area Delete current data	This error appears when an attempt is made to set all buffer space allocations to "0" (zero). The error message is displayed for approximately two seconds, then disappears.

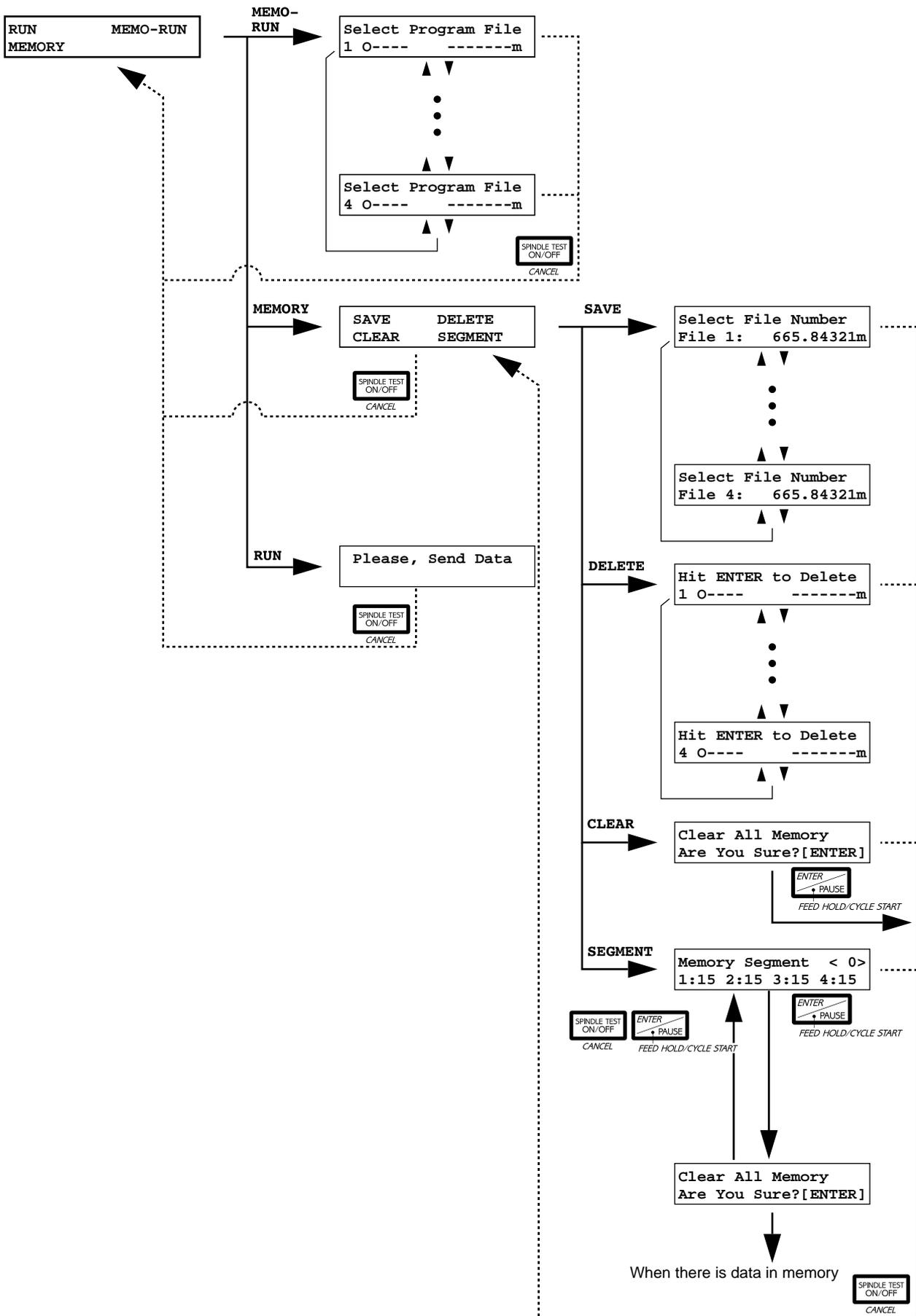
Z0 Sensor Connection Errors

Error message	Description
CAN'T TURN ON Z0SENSOR NOTHING	This error appears if the Z0 sensor is not connected when entering the sensor mode. The error message is displayed for approximately two seconds, then disappears. Connection the Z0 sensor before entering the sense mode.
CAN'T SET ZORIGIN Z0SENSOR ERROR	This appears when the Z0 sensor is displaced while moving the tool in the sensor mode. Switch the power off and back on again to cancel the error. Do not remove the sensor while in the sensor mode.
CAN'T SET Z ORIGIN POSITION RANGE UNDER	This appears when EXOFS Z is set at a location outside the operating area (below -150 mm). If this message appears, the EXOFS Z location is not changed.

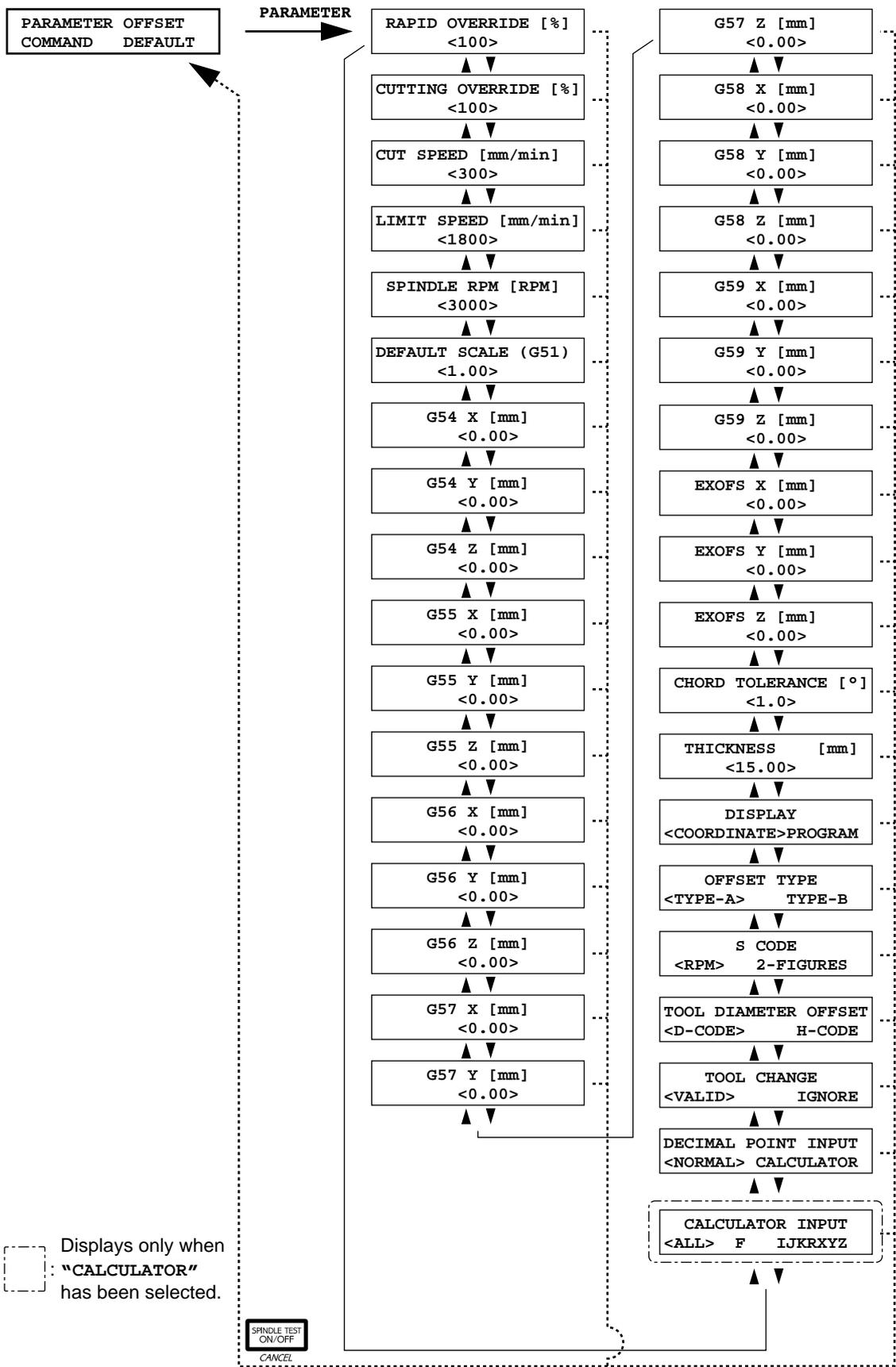
Display Menu Flowchart



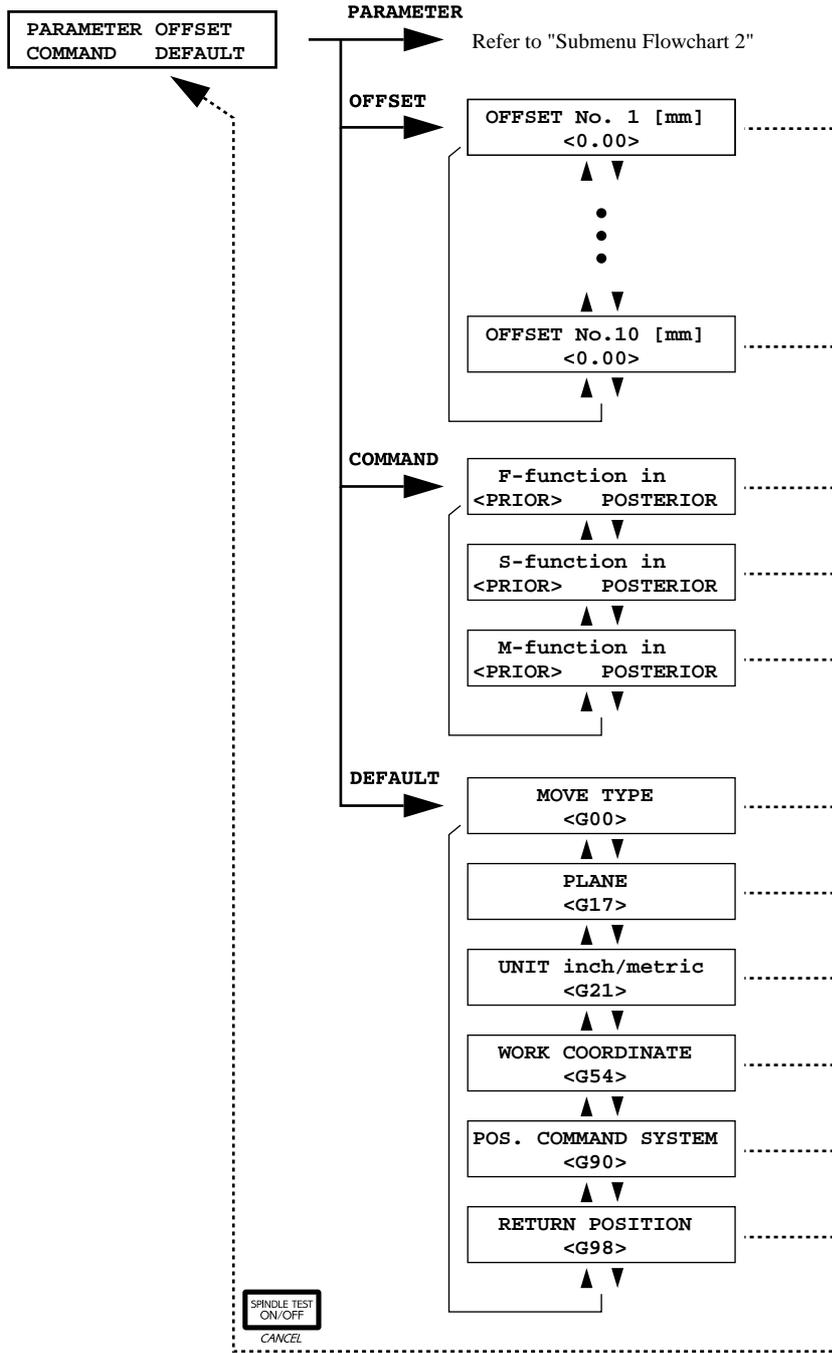
Submenu Flowchart 1



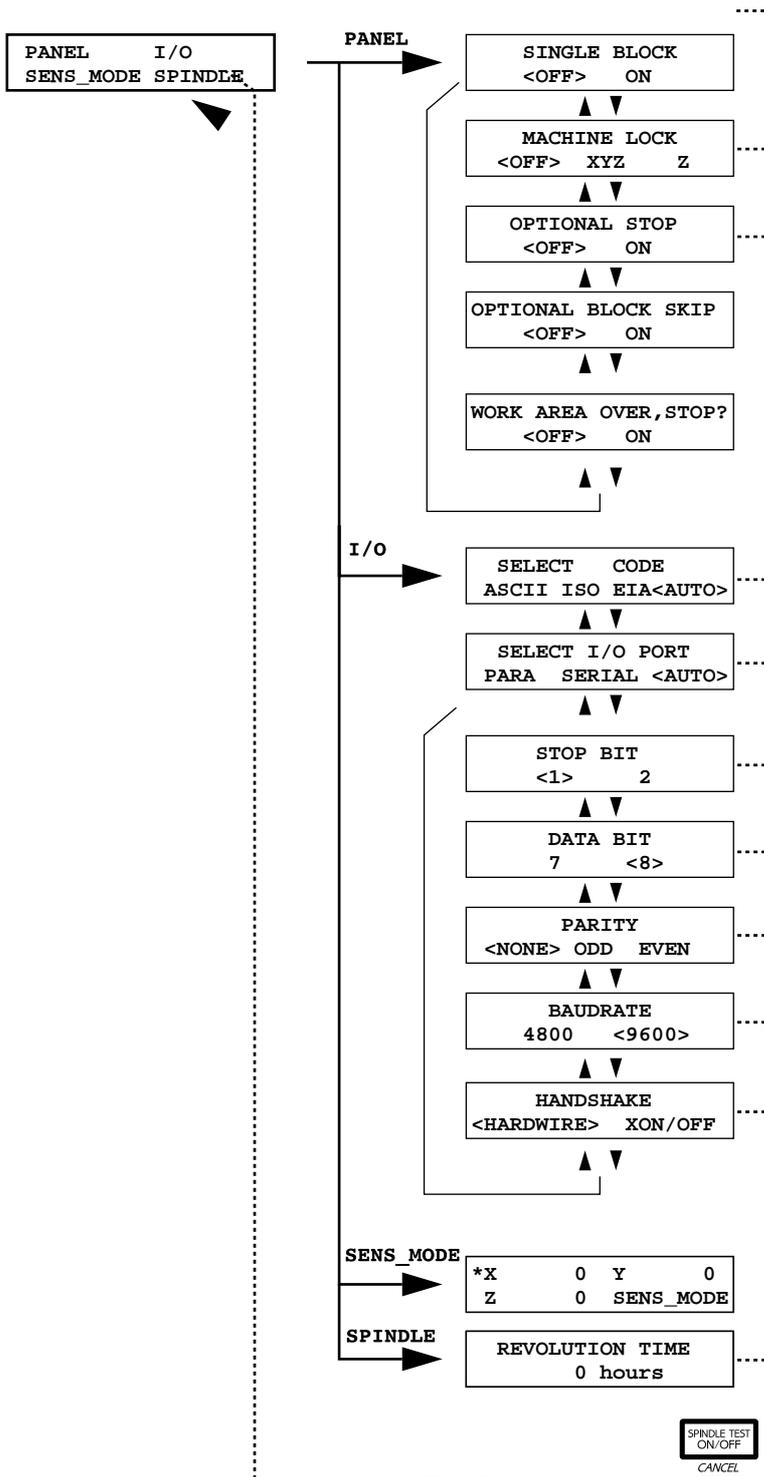
Submenu Flowchart 2



Submenu Flowchart 3

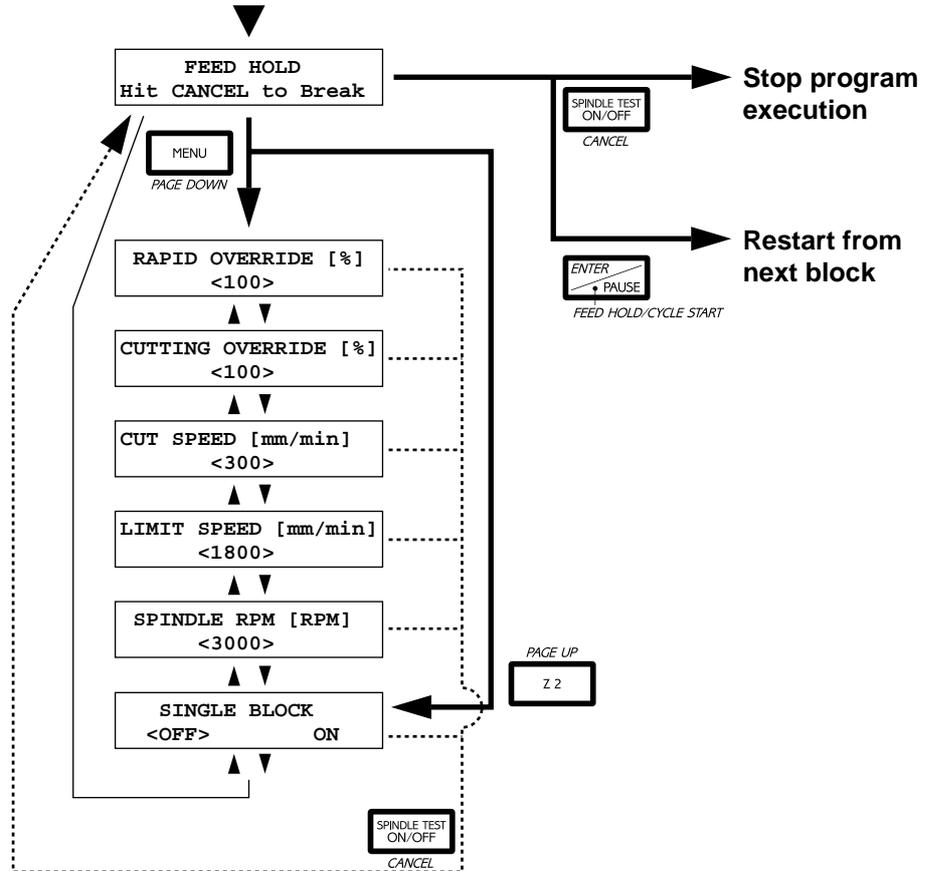


Submenu Flowchart 4



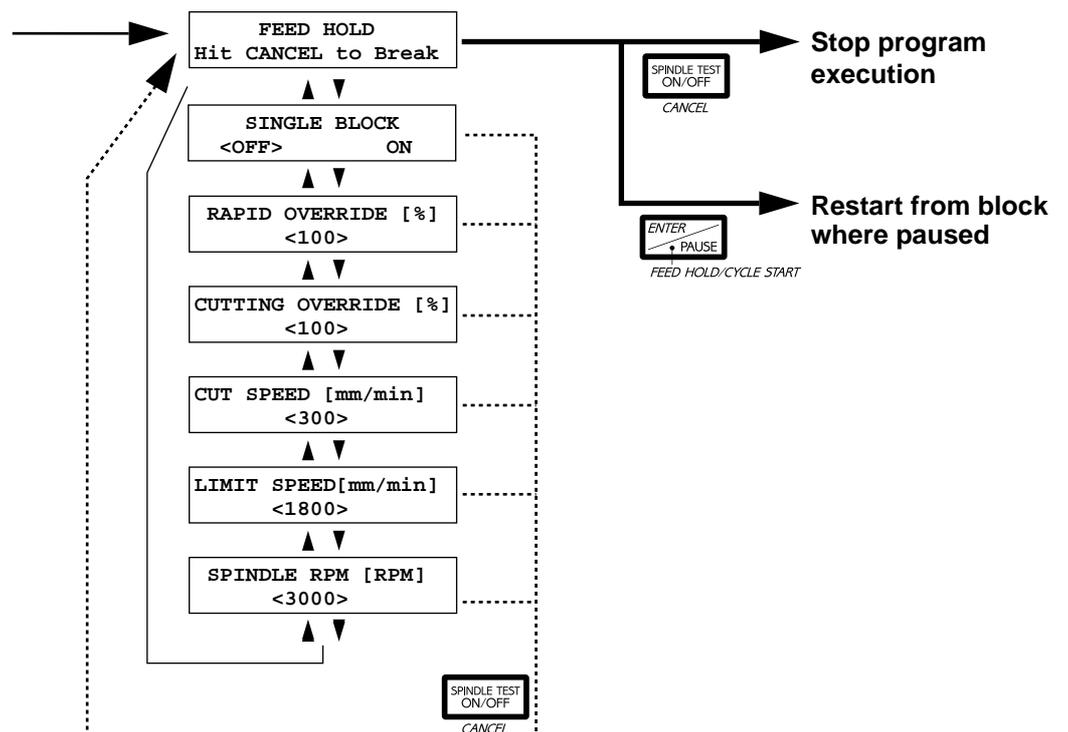
Menu Flowchart When Paused

Press the [ENTER (FEED HOLD/CYCLE START)] key while cutting in progress.



Menu flowchart during single-block operation

*X	0	Y	0
Z	0	3000 RPM	



- MEMO -

