SONY®

Controller
NU20-20/30

Read all the instructions in the manual carefully before use and strictly follow them.
Keep the manual for future references.

Instruction Manual
1st Edition (Revised 2)

[For the customers in U. S. A.]

WARNING

This equipment has been tested and found to comply with the limits for a Class A digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference when the equipment is operated in a commercial environment. This equipment generates, uses, and can radiate radio frequency energy and, if not installed and used in accordance with the instruction manual, may cause harmful interference to radio communications. Operation of this equipment in a residential area is likely to cause harmful interference in which case the user will be required to correct the interference at his own expense.
You are cautioned that any changes or modifications not expressly approved in this manual could void your authority to operate this equipment.

Trademarks

- Microsoft Windows®95 is a registered trademark of Microsoft Corporation.

1. Contents

2. Before Operation

Welcome to the world of the Mill Station!
We are very proud that the articles that you create using the Mill Station are used in applications all over the world.

The Product Designers

WARNING – This equipment has been tested and found to comply with the limits for a Class A digital device, pursuant to Part 15 of the FCC Rules. These limits are designed to provide reasonable protection against harmful interference when the equipment is operated in a commercial environment.

This equipment can generate and radiate radio frequency energy and, if not installed and used in accordance with the instruction manual, may cause harmful interference to radio communications. Operation of this equipment in a residential area is likely to cause harmful interference in which case the user will be required to correct the interference at his own expense. You are cautioned that any changes or modifications not expressly approved in this manual could void your authority to operate this equipment.

NOTICE
THIS NOTICE IS APPLICABLE FOR USA ONLY.
If shipped to USA, use the UL LISTED power cord specified below.

DO NOT USE THE OTHER POWER CORDS EXCEPT THE ONE IN ACCESSORIES.

2-1.   Precautions

This Sony Precision Technology product has been designed with full consideration given to safety. Improper handling of the device during operation or installation is dangerous and can lead to death, serious injury, or other accidents due to fire or electrical shock. This can also lead to impaired product performance.

Always observe the safety precautions to prevent these kinds of accidents from occurring. Be sure to read the precautions before performing any operations, installation, maintenance, inspection, repairs, or other work.

The following safety notices are displayed in this manual. Be sure that you fully understand the safety points before reading this manual.

⚠ Caution

Failure to observe a Caution notice can result in an injury or damage to the surrounding equipment due to electrical shock or other failure.

⚠ Warning

Failure to observe a Warning notice can result in death, serious injury, or other accidents due to fire, electrical shock, malfunction, or other failure.

Note

Points that should be noted for proper handling of the product.

⚠ Warning

1. When installing the NU20 controller, be sure to always ground the grounding terminal for the power cable. Also, if connecting to the NZ12 electrical box, be sure to always ground the power cable grounding wire.
2. Do not expose the NU20 (or the NZ12) to direct sunlight or heat emitted directly from a heater.
3. Do not use electrical welding equipment in the area around the NU20 or perform other work which can cause electrical interference. If this work must be performed, then take all necessary steps at the source of interference and at the NU20 to prevent the interference from occurring.
4. Be careful not to hit your head on the controller or the arm.
5. Be careful that hot scraps, casting particles, and other substances do not get in contact with the controller.
6. Do not put the plastic cover on the Mill Station system while it is operating.
7. Observe the following conditions for the Mill Station system.

Operating temperature	Ambient temperature: 32°F – 104°F (0 – 40°C)
Operating humidity	20% – 80% RH, no condensation
Storage temperature	Ambient temperature: –4°F – 140°F (–20 – 60°C)
Storage humidity	10% – 90% RH, no condensation
Supply voltage	AC 115 V ± 10% 60 Hz

Failure to observe these operating conditions can lead to operation failure or malfunctions.

8. Do not modify the Mill Station system or disassemble unless instructed to do so by Sony Precision Technology. Any modifications not expressly approved will void the warranty.

9. Do not cover the gap at the bottom of the controller. This gap serves as a ventilating hole for cooling the system. Do not place any objects within 10 inches (25 cm) of this ventilating hole.

10 inch

10 inch
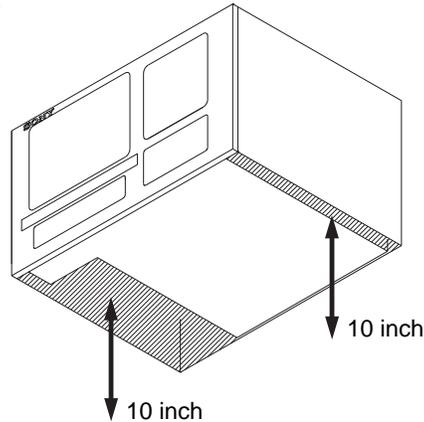
10. **Be sure to always press the POWER OFF key on the controller when turning the system off. Do not turn off the system by setting the main power switch on the electrical box to OFF. Doing so could result in a malfunction the next time that the power is turned on.**

11. The connectors for the keyboard and mouse are used for maintenance. Do not connect a keyboard or mouse. This could lead to a breakdown or malfunction.

12. Do not install any other applications on the Mill Station system. This could lead to a breakdown or malfunction.

13. Read the NZ12 instruction manual for the precautions regarding the NZ12 electrical box.

Machine oil

Cutting oil

Switchboard
(Locate more than 0.5 m/20" away.)

Chips

Welder
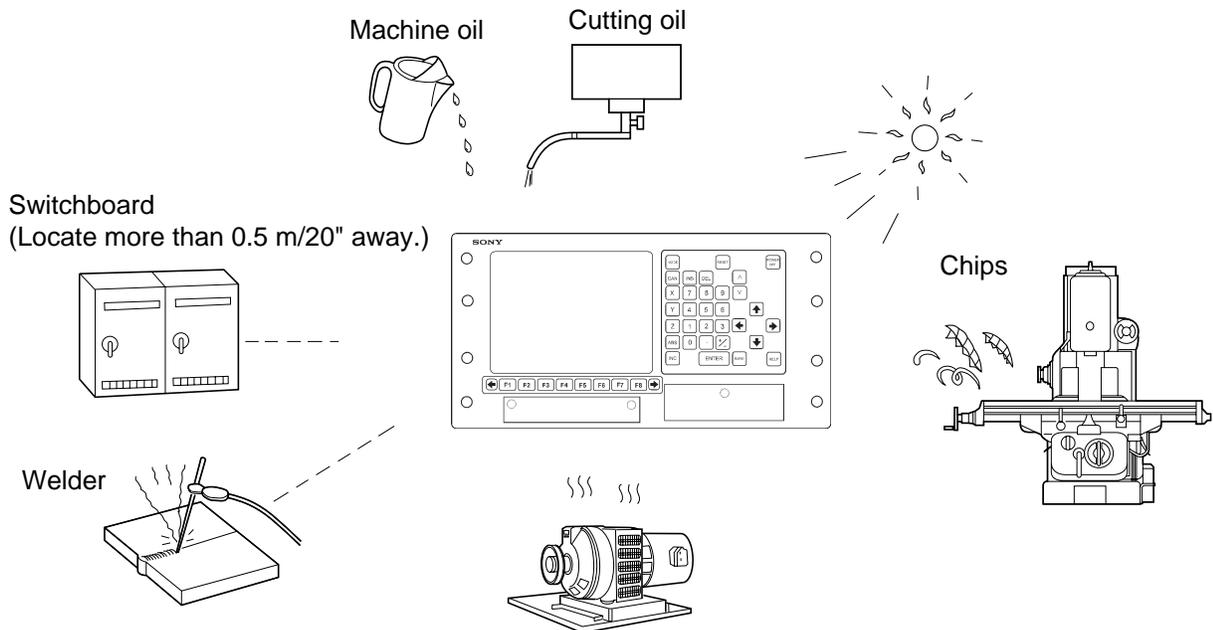
2-2.   Maintenance

⚠ **Caution**

(1) Supply oil to the sliding surface at the manual oiling pump on the milling cutter according to the milling cutter instruction manual. The oil supplied to the sliding surface should meet or exceed the designated rotations and intervals in the machine instruction manual. For continuous operation, oil should be supplied at least once an hour. The machine will not operate properly if you fail to provide oil properly. Improper operation of the milling cutter could also lead to malfunctions of the Mill Station system.

(2) When moving the milling cutter, be careful not to subject the Mill Station system to a shock of 0.6 G or more. (1 G is equal to the acceleration of gravity, an acceleration of 9.8 m/s^2.)

(3) When cleaning the Mill Station system, use a soft cloth which has been dipped in a diluted neutral detergent and wringed dry. Use of other solvents can result in discoloration and deformation of the casing.

(4) Air filter should be replaced every 3 months. There are two different covers as shown below. First, remove the cover if it does not have an opening. Remove the filter, wash it with a neutral detergent, dry it thoroughly, and then reattach the filter.

Filter
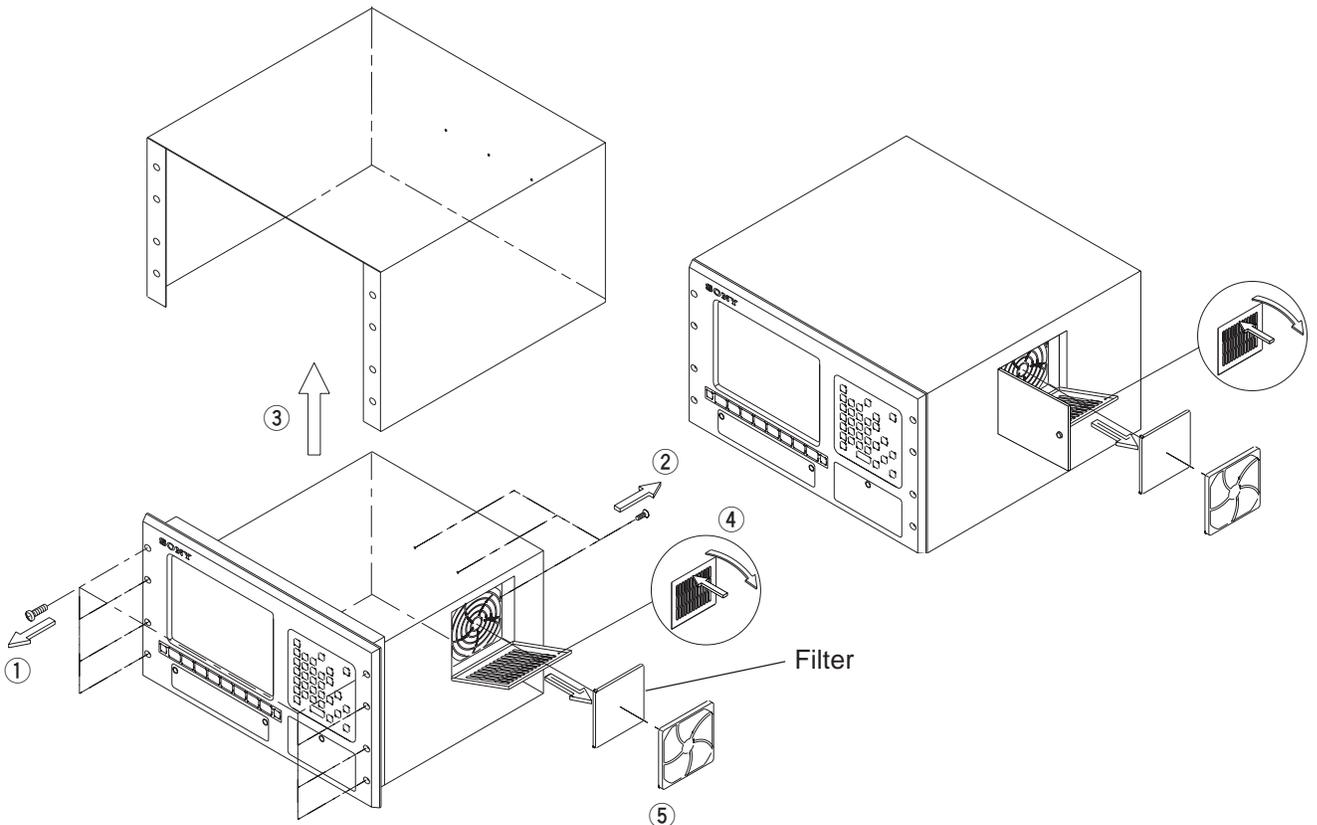
3.   General

3-1.   System Configuration

The system configuration when combined with the NZ12 electrical box is shown below.

NU20
(Controller)

NZ31
(Spindle control unit)

NZ12
(Electrical box)

GA-A, GB-A
SJ700 scale

NZ32 (Overtravel unit)

NZ30 (JOGMPG unit)

Servo motor

START/PAUSE SW

NZ33 (Zero point sensor)

TS-xxxA (Touch sensor)
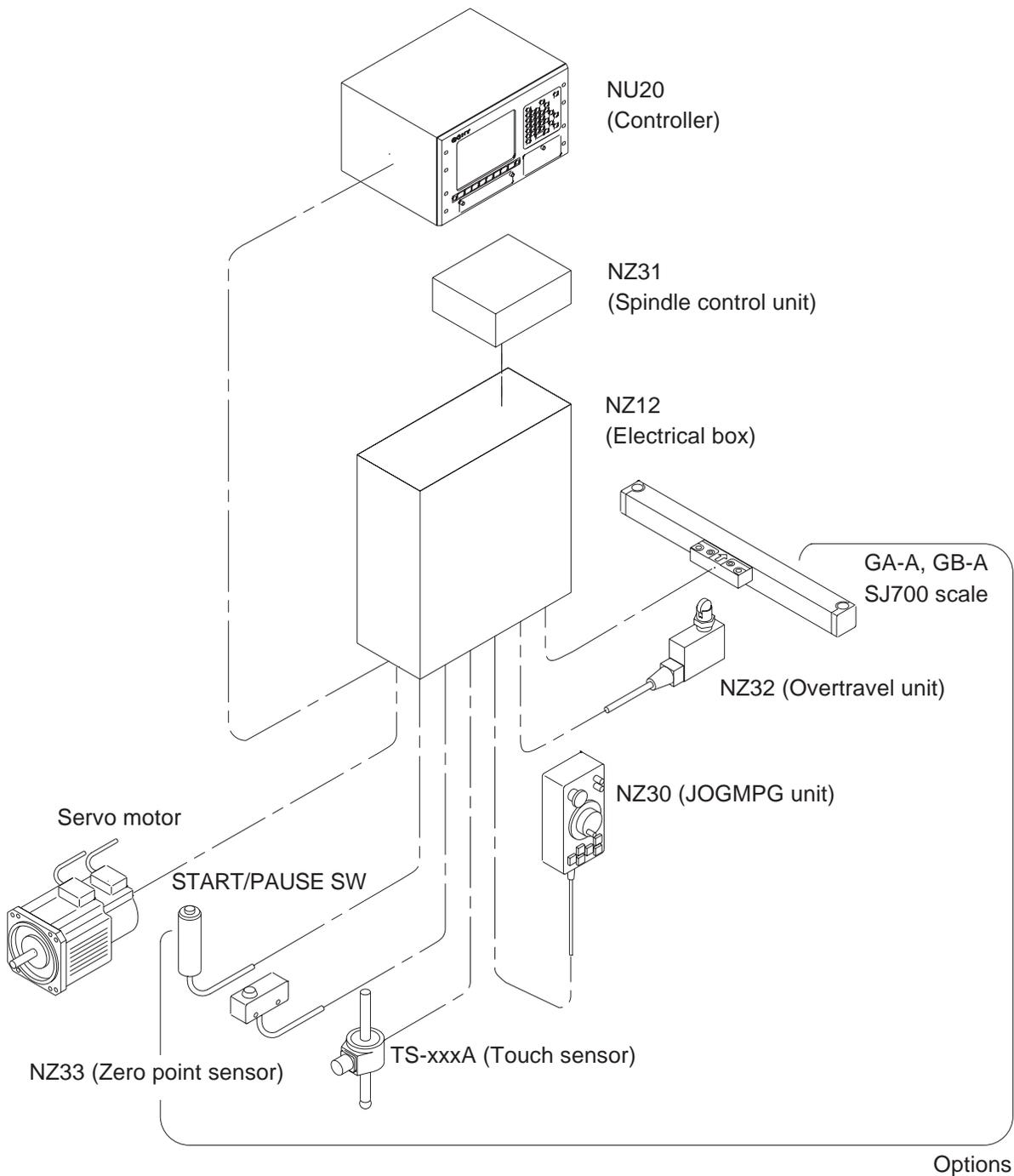
Options

NU20 Key Description
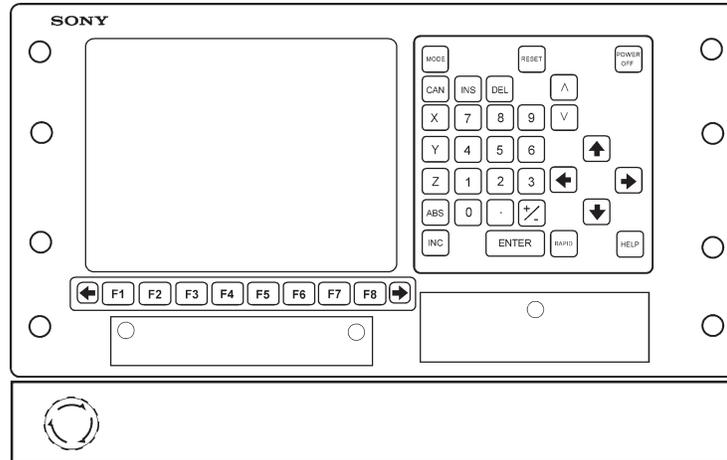
MODE MODE Key: Pressing this key from any screen returns you to the Mode Selection screen.

CAN **INS** **DEL** CAN, INS, DEL Keys: These keys are used for editing numbers and text. CAN=Cancel, INS=Insert, DEL=Delete

X **Y** **Z** **0** **–** **9** X, Y, Z, 0-9 Keys: These keys are used for entering various numerical data.

+/− +/− Key: This key is used for entering negative numbers. Pressing it again makes it positive.

ENTER ENTER Key: This key is used to input the data into the program.

↑ ↓ ← → ↑ ↓ ← → Keys (Cursor Keys): These keys are used to move the cursor during jog feed movement and entering numerical data for the X and Y axes.

RAPID RAPID Key: Press this key during jog feed to perform rapid traverse.

∧ ∨ ∧∨Keys (UP/DOWN Keys): This key is used to move the cursor during jog feed movement and numerical input for the Z axis.

HELP HELP Key: Pressing this key shows a description of the screen being displayed.

POWER OFF POWER OFF Key: Press this key to turn off the power.

Note
Do not turn off the power using any other method. This could cause the system to break down.

RESET RESET Key: When this key is pressed, the MDI operation or program operation being executed is stopped, and the program data is initialized. Also, if the machine is operating, it will stop when the RESET key is pressed. **To proceed with operation, start again from the beginning.**

7

- Spindle rotation does not stop when the RESET key is pressed.
- If the RESET key is used to prevent a dangerous situation, remove the cause of danger and then check for safety before starting operation again.

ABS ABS Key: This key is used to set the workpiece coordinates. Movement of the machine by program or MDI commands is performed based on the workpiece coordinates. The zero position on the workpiece coordinate system is called the datum point.

INC INC Key: This key is used during MANUAL mode. This key does not affect commands for program machining or MDI machining.

F1 — **F8** F1 to F8 Keys: These are the function keys.

STOP switch: This is the STOP switch operating with the highest priority for motor driving and software execution. If the STOP switch is pressed while the machine is operating, the switch locks, all axis movement is cancelled, and machine comes to a sudden stop. Turning the switch clockwise cancels the switch lock. When this switch is pressed, the program to be executed is reset. Therefore, **to proceed with operation, start again from the beginning.**

- Spindle rotation does not stop when the STOP switch is pressed. This is to prevent the machining tool from biting into the workpiece.
- If the STOP switch is used to prevent a dangerous situation, remove the cause of danger and then check for safety before starting operation again.

Mouse   Keyboard   Floppy Disk Drive
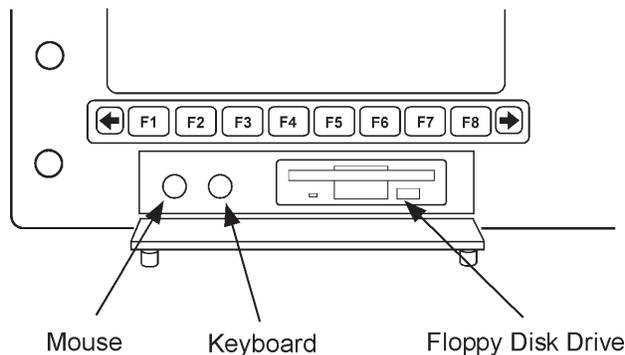
Mouse, keyboard:
The connectors for the keyboard and mouse are for maintenance use only. Do not connect a keyboard or mouse. This could lead to a breakdown or malfunction.
Floppy disk drive: Created programs can be saved or read to floppy disk.
Do not close the lid on the front panel when a floppy disk is inserted. When not using the floppy disk, remove it from the floppy disk drive, and close the lid.

SERIAL PORT

STOP SW INPUT

To Electrical Box

COOLING FAN   AC INPUT

COOLING FAN
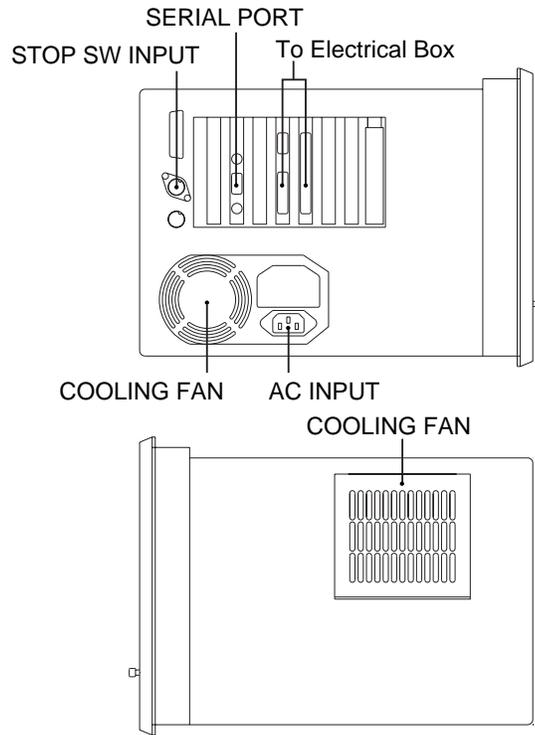
The cable connections to the main unit are shown below.

STOP Switch

20 pin

100 pin

AC Power Supply

(From Electrical Box)
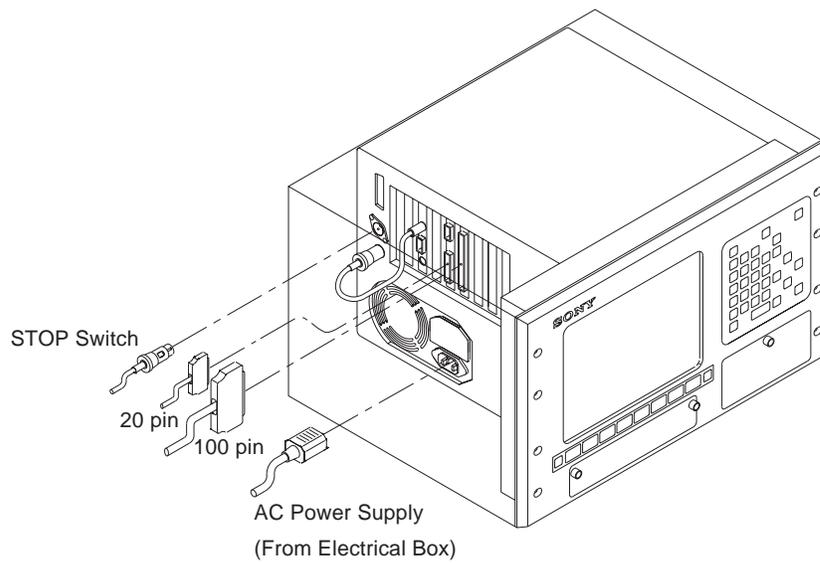
NZ30 (JOGMPG Unit) Switch Description
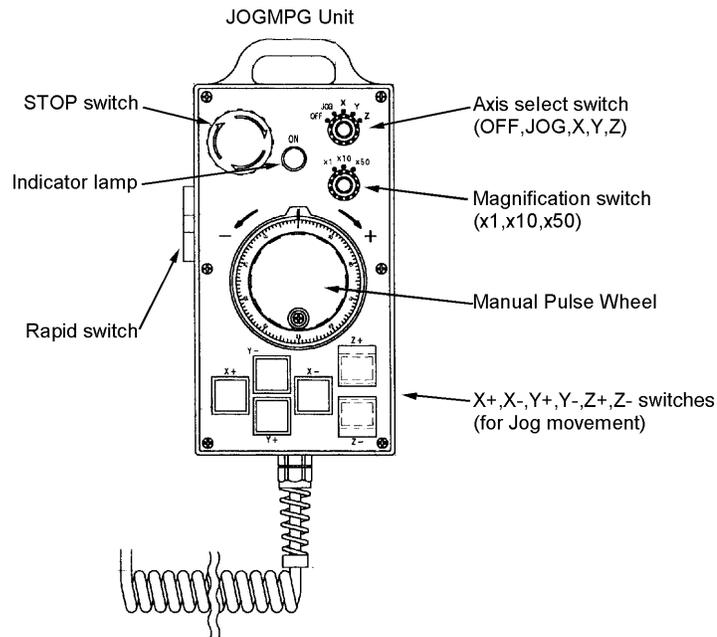
JOGMPG Unit

STOP switch

Indicator lamp

Rapid switch

Axis select switch (OFF,JOG,X,Y,Z)

Magnification switch (x1,x10,x50)

Manual Pulse Wheel

X+,X-,Y+,Y-,Z+,Z- switches (for Jog movement)

STOP switch: This is the STOP switch operating with the highest priority for motor driving and software execution. If the STOP switch is pressed while the machine is operating, all axis movement is cancelled, and machine comes to a sudden stop. When this switch is pressed, the program to be executed is reset. Therefore, to proceed with operation, start again from the beginning. The STOP switch locks when it is pressed, and it unlocks when turned clockwise.

Caution
* If the STOP switch is used to avoid a dangerous situation, check for safety before starting operation again.

Manual Pulse Wheel: Rotating this wheel allows the machine to be moved manually.

Axis Select Switch: Setting this switch to OFF disables the JOGMPG Unit function so that it does not operate. When the switch is set to JOG, only Jog Feed (X+, X–, Y+, Y–, Z+, Z–) is enabled. When the switch is set to X, Y, Z, the Manual Pulse Wheel can be used to move each axis. In this case, all of the JOG commands also become enabled at the same time. Also, when the switch is set to the X, Y, Z position, the servo lock for the servo motor is set to ON, and the table cannot be moved using the machine hand wheel. When the switch is set to the JOG position, the table can be moved manually using the machine hand wheel as long as the X+, X–, Y+, Y–, Z+, Z– switches are not pressed.

Magnification Switch: This switch multiplies the movement amount by a fixed value when moving the table using the Manual Pulse Wheel. The x1 setting moves the table by the preset minimum command increment. The x10 setting multiplies this increment by 10, and the x50 setting multiplies this increment by 50. Normally, x1 is set to a movement amount of 0.0001 inch (0.001 mm). The x50 will be enable after set the reference point return operation.

Rapid Switch: Press this key during Jog Feed allows rapid traverse to be performed.

3-3.   List of Functions

The NU20 is equipped with the functions shown below.

```
                                    ┌──────────────┐
                      POWER ON      │     DRO      │
                          │         ├──────────────┤
                          │         │     JOG      │
                          ▼         ├──────────────┤
                      ┌─────────┐   │  REF RETURN  │
                      │ MANUAL  │───┤              │
                      └─────────┘   │ LINEAR MILL  │
                                    ├──────────────┤
                                    │  CIRCLE MILL │
                                    ├──────────────┤
                                    │   SPINDLE    │
          Press                     ├──────────────┤
         ┌──────┐                   │  SAFE ZONE   │
         │ MODE │                   ├──────────────┤
         └──────┘                   │ TOUCH SENSOR │
           Key                      └──────────────┘
  ┌──────────┐                      ┌──────────────┐
  │  MODE    │◄──     ┌─────────┐   │    CREATE    │
  │Selection │        │  PROG   │───┤              │
  │ Screen   │        └─────────┘   │     EDIT     │
  └──────────┘                      ├──────────────┤
                                    │     RUN      │
                                    ├──────────────┤
                                    │  PROG SEND   │
                                    ├──────────────┤
                                    │  PROG RECEIV │
                                    ├──────────────┤
                                    │     DNC      │
                                    └──────────────┘
                      ┌─────────┐   ┌──────────────┐
                      │   MDI   │───│    DRILL     │
                      └─────────┘   ├──────────────┤
                                    │  FACE MILL   │
                      ┌─────────┐   ├──────────────┤
                      │TOOL DATA│   │ ISLAND MILL  │
                      └─────────┘   ├──────────────┤
                                    │ POCKET MILL  │
                      ┌─────────┐   ├──────────────┤
                      │ 2AX/3AX │   │  FRAME MILL  │
                      └─────────┘   ├──────────────┤
                                    │  RING MILL   │
                      ┌─────────┐   ├──────────────┤
                      │SET SYSTEM│  │   LINE ARC   │
                      └─────────┘   └──────────────┘

                      ┌─────────┐
                      │DIAGNOSTIC│
                      └─────────┘
```
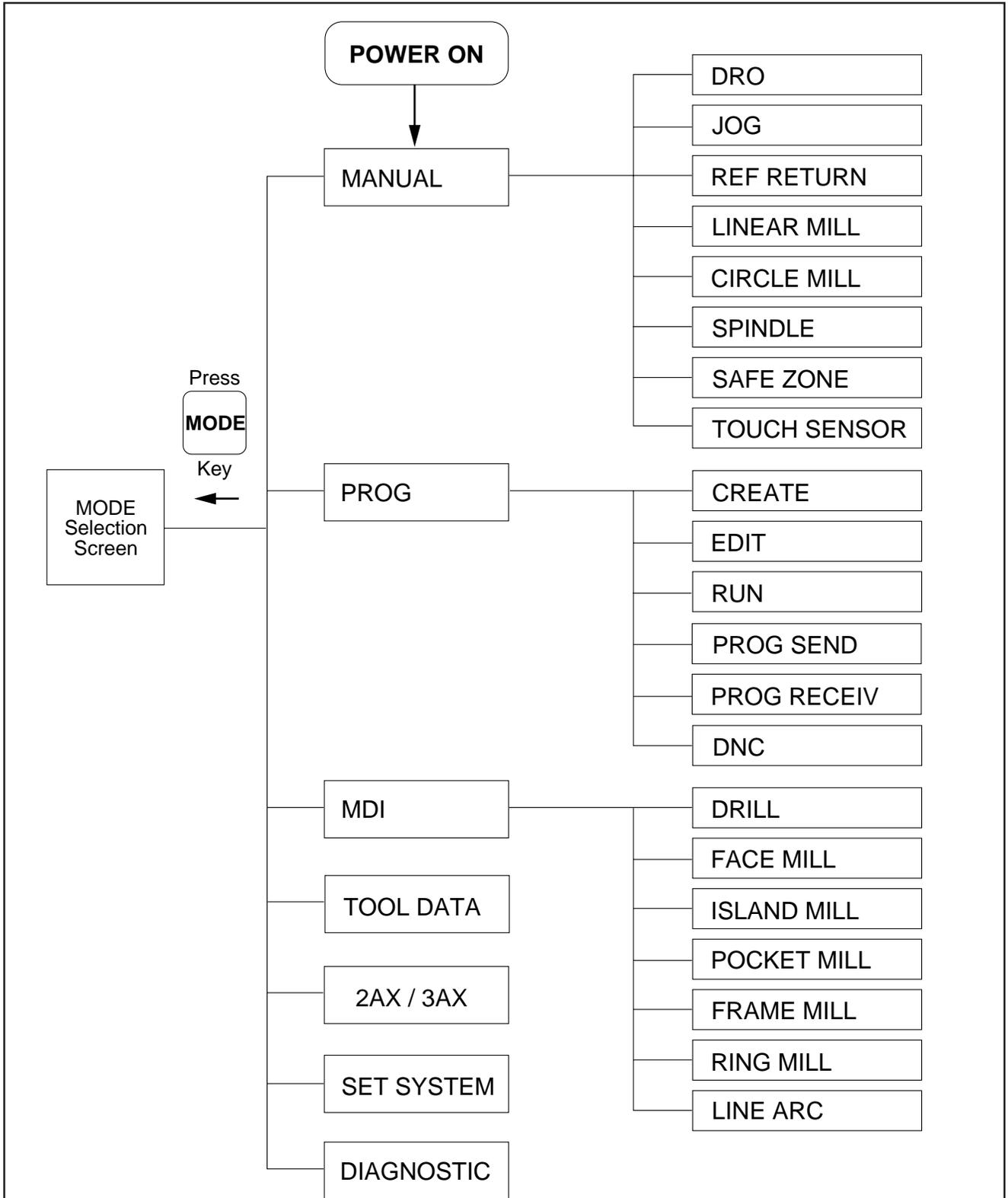
4.   Coordinate Definitions

4-1.   X, Y, Z

The NU20 recognizes and controls machine movement as the movement directions shown below.
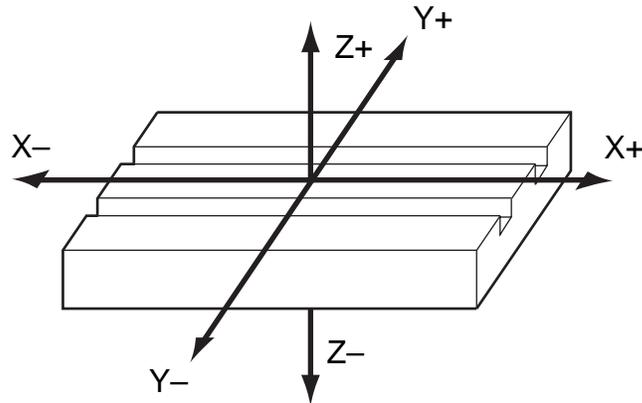
The tool moves to the right when the X+ key is pressed (when turned in the + direction). In other words, the table moves to the left.
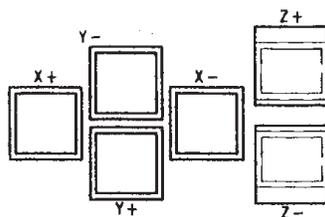The tool moves to the back when the Y+ key is pressed (when turned in the – direction). In other words, the table moves forward.
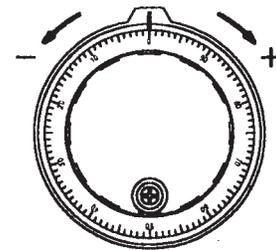The tool moves up when the Z+ key is pressed (when turned in the + direction). (For 2-axis models, move the tool up or down manually.)
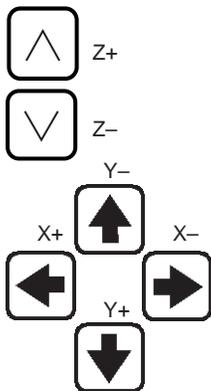
NU20 Key Panel   **JOGMPG JOG switches**   **JOGMPG Manual Pulse Wheel**

NOTE
Z+ and Z– are only used in the 3-axis model.

The NU20 is provided with the coordinate systems shown below.

### Machine Coordinate System: (MACHINE POSITION)

This coordinate system is based on using the zero position when the reference point return operation is performed as the reference zero for the machine coordinate system. All coordinates are then based on this zero point. Normally, the zero point is not displayed on the screen since the NU20 system is being used.

Workpiece Coordinate System: (WORK POSITION)

This coordinate system uses the commands issued during MDI machining and program machining. The zero position for the workpiece coordinate system is called the datum

point. The coordinate settings are made using the $\boxed{\text{ABS}}$ key in the numeric keypad.

Incremental Coordinate System: (INCREMENTAL POSITION)

This coordinate system is used when you want to temporarily change coordinates in Manual mode. It does not affect the coordinates during MDI operation and program

operation. The coordinate settings are made using the $\boxed{\text{INC}}$ key in the numeric keypad.
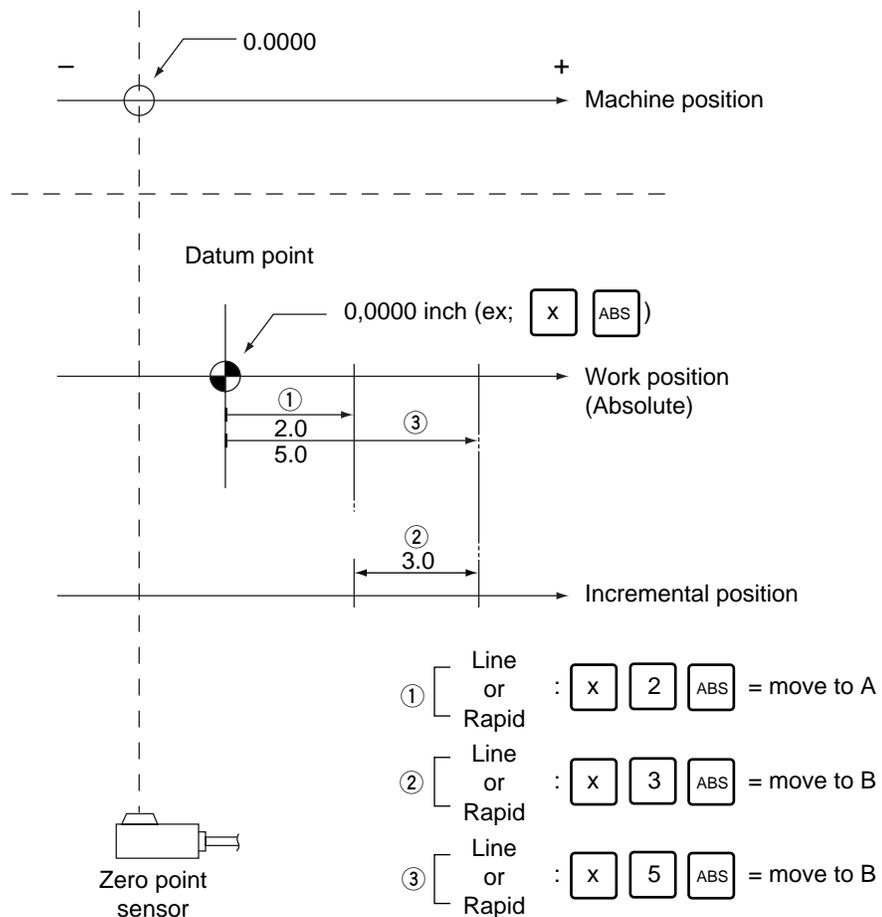
5. Power ON/OFF

5-1. Turning the Power On

(1) Before turning on the power, check that there is nothing unusual with the controller, electrical box, milling machine, and other equipment. It is recommended that you press the STOP switch as a safety measure if it is not already pressed.
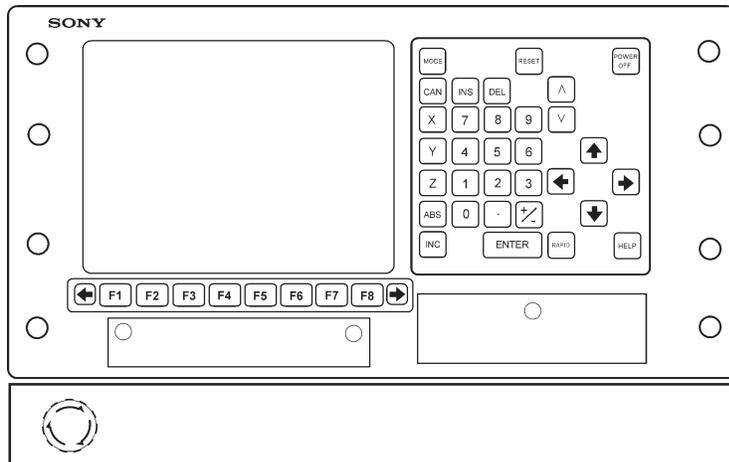
(2) Raise the switch on the left side of the electrical box at the back of the machine (when using the NZ12). If the configuration has another electrical system, turn on the AC inlet.
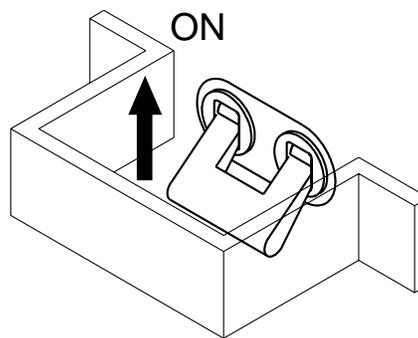
(3) Wait until the screen shown below is displayed. The power on procedure is complete when this screen appears. The displayed numerical settings are the settings when the power was last turned off normally.

MANUAL				3AXIS	inch
11:20:00					

X   0.0000 ABS
Y   0.0000 ABS
Z   0.0000 ABS

JOGFEED	7.9	inch/min
OVERRIDE	100	%

| FEED ↑ | FEED SET | FEED ↓ | OVER RIDE↑ | 100% | OVER RIDE↓ | MOVE ABS 0 | REF RETURN |

(4) Turn the STOP switch clockwise to unlock it. The STOP switch is located at the bottom left of the controller. The machine can now operate. **If the JOGMPG Unit (option) is connected,** turn the STOP switch on the JOGMPG Unit clockwise to unlock it.

5-2.   Turning the Power Off
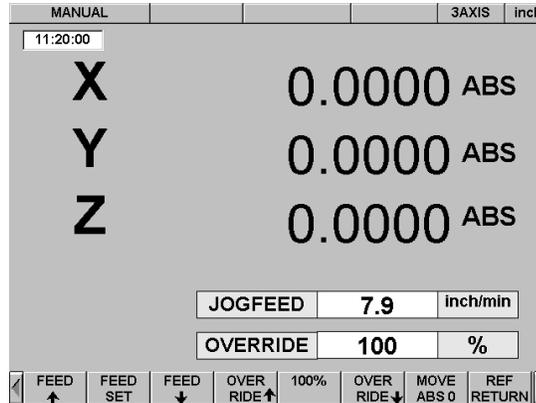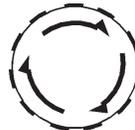
(1) Stop all operations, and stop all machine movement. If a program is being created, close the program creation or editing.
(2) Press the STOP switch on the controller side.

(3) Press the POWER OFF key on the controller and then press the ENTER key. The power will automatically shut off after a short time.

POWER OFF   and then   ENTER

⚠ Warning
Be sure to only use the POWER OFF key on the controller when turning the system off. Do not turn off the system by setting the main power switch on the electrical box to OFF. Doing so could result in a malfunction the next time that the power is turned on.

⚠ Caution
If you accidentally press the POWER OFF key and then ENTER key during operation and the system shuts down, wait at least 30 seconds before turning the power on again. After turning the power on again, be sure to always redo the reference point return.
Once the POWER OFF sequence has been started, do not under any circumstances attempt to move the machine or perform other operations until the power supply has been cut off. This could prevent the power from shutting off normally, and the machine may not operate properly when the power is turned on the next time.

6.   Limit Functions

The NU20 is equipped with three limit functions for ensuring safety. The features and usage procedures for each function are described below.

6-1.   Overtravel

When the NZ32 Overtravel Unit (option) is mounted on the machine, the motor can be stopped at the position where the Overtravel Unit is located. This is used to prevent machine damage.

Note

When the machine hand wheel is used to move the table, a message such as LIMIT X+ is shown on the screen at the position where the Overtravel Unit (option) is mounted. However, the table movement is not stopped.
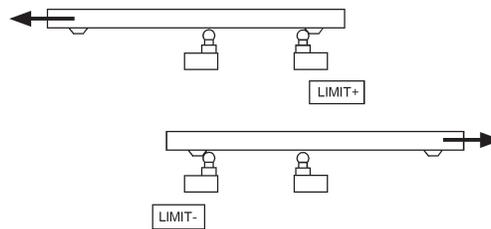
6-2.   Software Limit

The Software Limit is a function which limits the allowable machine movement range at the NU20 software side. The machine automatically stops when the Software Limit point is reached during jog feed, MPG feeding, MDI operation, or program operation.
This function is automatically set after the reference point return operation is performed. The allowable movement range is determined by the system parameters. Just before the machine reaches the software limit point, it begins to decelerate. The machine then decelerates further at the limit position, and then comes to a stop. The distance from the setting point to the stopping point depends on the movement rate.
When the machine starts to decelerate, SLOW is shown on the Status display on the screen. The feed rate in the deceleration area is determined by the system parameters. If the machine enters the deceleration area with a feed rate slower than the deceleration area feed rate, then it continues to move at the same rate. The operation and screen display is shown in the figure below.

Indication of the screen
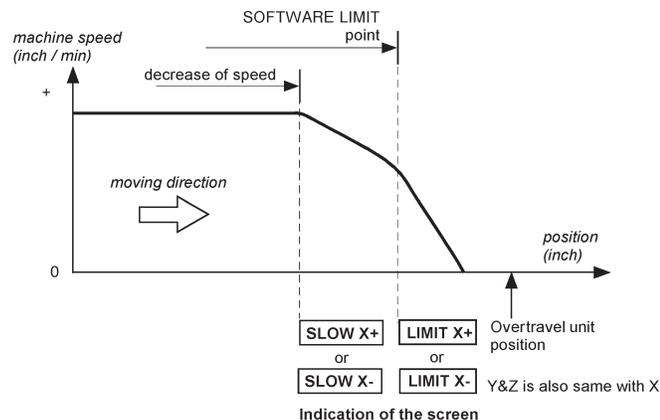
Software Limit function does not work unless the reference point return operation is performed. This means that rapid traverse cannot be performed at the JOG feed function (see the section on the Reference Point function for more information). Be sure to always perform the reference point return operation when the power is on immediately. To change the settings to allow rapid traverse once the power is turned on, change the system parameters. (Please inquire at Sony Precision Technology for the procedure of setting the system parameters.)

The settings for the Software Limit values are reset when the power is turned off. Perform reference point return again when turning on the power again. The Software Limit function works only when the servo motor performs the driving action.

Note
If the reference point setting is not made, the Software Limit function does not work. In this case, if the Overtravel Limit Switch (option) is also not installed, then issuing a command moving the machine beyond the machine stroke end when the servo motor performs the driving action should damage the machine.

Note
If the machine is stopped by the Software Limit during MDI operation or program operation, first, switch to Manual mode, and then use jog feed, Manual Pulse Wheel, and the machine hand wheel to move the machine to where the limit is cancelled.

6-3.　Safe Zone

The Safe Zone is a function for setting the movement forbidden area of the machine. This function is used in all motor driving modes when preventing the tool from entering an undesirable cutting area during workpiece machining. Use this function for rough shavings and other operations.

Two areas can be set at the same time. Setting the ABS coordinates for XY allows you to make settings which cannot enter the selected rectangular (or square) area and to make settings which cannot leave the area. The setting area is the rectangle determined by two points which form opposite angles.

Note
The ABS coordinates for setting the area are the tool center values. You must consider the diameter of the tool for setting these areas.

☐ :Movement forbidden area
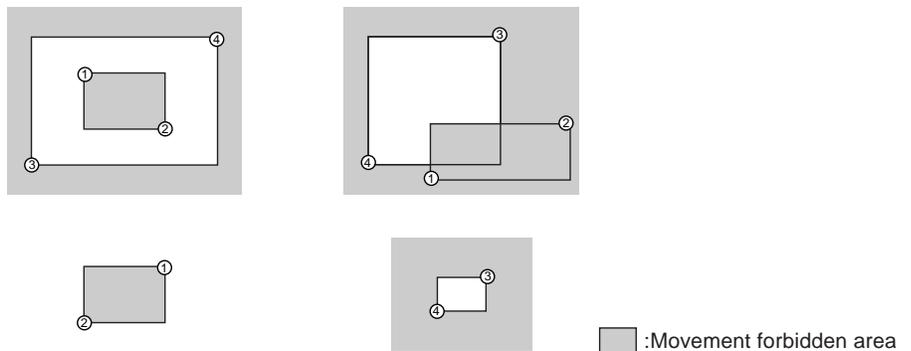
Note
The machine stops moving near the position set by the Safe Zone function. The stopping accuracy is affected by the machine moving rate. If the machine is moving at a slow feed rate, then it will stop just before the Safe Zone. If it is moving at a fast feed rate, then it will enter the Safe Zone. If the machine stops just before the Safe Zone, the screen will not show a message such as "LIMIT X+". When moving with the MPG, the machine will enter the Safe Zone and stop.

6-4.　Setting the Safe Zone

1) Key Description

The keys for the Safe Zone are described below.

1. Press the MODE key.

2. Press the F1 (MANUAL) key.

3. Press the ⬅ or ➡ function key.

4. Press the F4 (Safe Zone) key.

5. The system enters Safe Zone mode. The following function keys can be used.

<	POINT SELECT	TEACH		PREV DATA	VIEW CHANGE	INSIDE ON/OFF	OUTSIDE ON/OFF	PAGE BACK	>
<	FEED ↑	FEED SET	FEED ↓	OVER RIDE ↑	100%	OVER RIDE ↓		PAGE BACK	>

Key Name	Function
POINT SELECT:	
	Select the setting points for setting the Safe Zone. There are a total of four points: points ① and ② for settings which cannot enter inside the area and points ③ and ④ for settings which cannot leave the area. The points for selection change each time the key is pressed. The displayed numerical position is the ABS coordinates for the tool center.
TEACH:	
	Pressing this key sets the ABS coordinate value for the current tool center at the point selected by POINT SELECT.
PREV DATA:	
	This key returns the changed numerical settings for a zone back to the numerical values when the Safe Zone was first retrieved.
VIEW CHANGE:	
	This key enlarges or reduces the Safe Zone display shown on the screen.
INSIDE ON/OFF:	
	When set to ON, the tool center is forbidden from entering inside the Safe Zone. When set to ON, the inside of the rectangle set by points ① and ② is shaded, and the INSIDE ENABLE MODE display at the left-center of the screen changes to DISABLE MOVE and turns to green.
OUTSIDE ON/OFF:	
	When set to ON, the tool center is forbidden from going outside the Safe Zone. When set to ON, the outside of the rectangle set by points ③ and ④ is shaded, and the OUTSIDE ENABLE MODE display at the left-center of the screen changes to DISABLE MOVE and turns to green.

Setting the Safe Zone

1. Entering Safe Zone mode
2. Setting the movement forbidden area (movement inside is forbidden)
3. Setting the movement forbidden area (movement outside is forbidden)
4. Changing the display
5. Correcting the input values
6. Enabling/disabling the Safe Zone function settings (inside)
7. Enabling/disabling the Safe Zone function settings (outside)

1. Entering Safe Zone mode

1.1 Press the MODE key.

1.2 Press the F1 (MANUAL) key.

1.3 Press the ← or → function key.

1.4 Press the F4 (SAFE ZONE) key.

1.5 The system enters Safe Zone setting mode.

2. Setting the movement forbidden area (movement inside is forbidden)

2.1 First, set the area where movement inside is forbidden.

2.2 Press the F1 (POINT SELECT) key and then select P1.

2.3 Press X 0 – 9 ENTER to enter the desired numerical setting for the X axis.

2.4 Press Y 0 – 9 ENTER to enter the desired numerical setting for the Y axis.

2.5 The point ① display in the screen right side area begins to blink.

2.6 Press the F1 (POINT SELECT) key and then select P2.

2.7 Press X 0 – 9 ENTER to enter the desired numerical setting for the X axis.

2.8 Press Y 0 – 9 ENTER to enter the desired numerical setting for the Y axis.

2.9 The point ② display in the screen right side area begins to blink.

2.10 As soon as the setting position for the P2 X-axis and Y-axis is made, the inside forbidden area with opposite angles formed by points ① and ② is displayed on the screen as a rectangle with green lines. This completes the area settings.

Note
• P1 and P2 should always be selected so that they form the opposite angles of the desired area.
• The width of the INSIDE area should be set to at least 0.04 inches (approx. 1 mm). The machine may not operate properly if the width is less than 0.04 inches.

3. Setting the movement forbidden area (movement outside is forbidden)

3.1　Use the Teach function to set the area where movement is forbidden outside.

3.2　Press the [F1] (POINT SELECT) key and then select P3.

3.3　Use the JOG key or the JOGMPG Unit to move the table to the desired position.

3.4　Press the [F2] (TEACH) key.

3.5　The current position is input for the P3 X-axis and Y-axis, and the point ③ display in the screen right side area starts to blink.

3.6　Press the [F1] (POINT SELECT) key and then select P4.

3.7　Use the JOG key or the JOGMPG Unit to move the table to the desired position. The position display is shown at WORK POSITION at the top left of the screen.

3.8　Press the [F2] (TEACH) key.

3.9　As soon as the current position for the P4 X-axis and Y-axis is entered, the outside forbidden area with opposite angles formed by points ③ and ④ is displayed on the screen as a rectangle with blue lines. This completes the area settings.

4. Changing the display

If the display area shown at the right side of the screen is not clearly visible after setting the forbidden area, then you can switch the display area, as below.

4.1　Each time that the [F5] (VIEW CHANGE) key is pressed, the screen switches to entire area, INSIDE Zoom, OUTSIDE Zoom. Switch to the display with maximum visibility.

5. Correcting the input values

5.1　To change the numerical settings back to their original values while entering from the numeric keys, press the [CAN] key. This returns the values to their settings before input.

5.2　To change the numerical settings back to their original values after the numeric values have been input to P1 to P4, press the [F4] (PREV DATA) key. This returns the Safe Zone settings to the original values when retrieved.

6. Enabling/disabling the Safe Zone function settings (inside)

6.1　Press the [F6] (INSIDE ON/OFF) key.

6.2　The inside of the rectangle set by points ① and ② is shaded, and the INSIDE ENABLE MODE display at the left-center of the screen changes to DISABLE MOVE and switches to ON. The tool center is forbidden from entering inside the area.

6.3　Pressing the [F6] (INSIDE ON/OFF) key removes the shading and disables this setting.

7. Enabling/disabling the Safe Zone function settings (outside)

7.1   Press the ⬚F7⬚ (OUTSIDE ON/OFF) key.

7.2   The inside of the rectangle set by points ③ and ④ is shaded, and the OUTSIDE ENABLE MODE display at the left-center of the screen changes to DISABLE MOVE and switches to ON. The tool center is forbidden from going outside the area.

7.3   Pressing the ⬚F7⬚ (OUTSIDE ON/OFF) key removes the shading and disables this setting.

7.4   You can check whether the Safe Zone function is enabled by pressing the MODE key, and then checking that the SAFE ZONE display is highlighted in blue on the Mode Selection screen.

The relationship between Overtravel Limit, Software Limit, and Safe Zone is shown in the figure below.

OVERTRAVEL LIMIT AREA
SOFTWARE LIMIT AREA
SAFE ZONE OUTSIDE
SAFE ZONE INSIDE
TABLE MOVE AREA
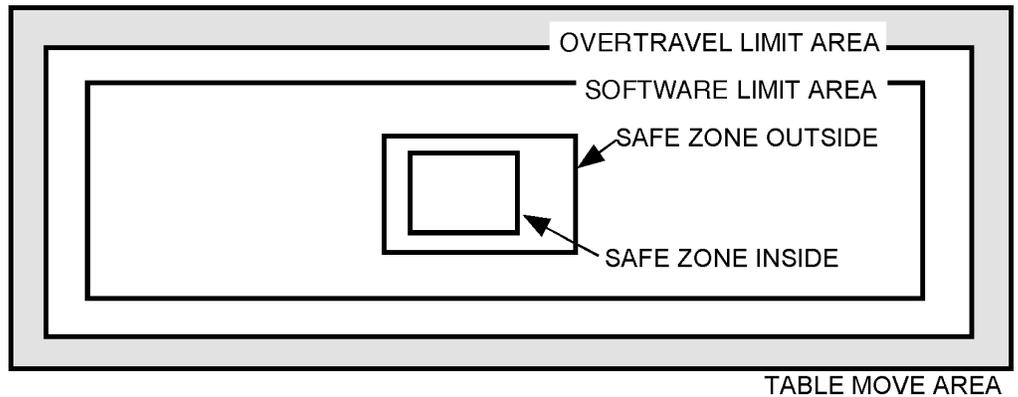
Note
When Overtravel Limit, Software Limit, and Safe Zone each reach their limit value, the following messages are shown at the top of the screen.

Message	Meaning
LIMIT X+	The X-axis cannot move any farther in the plus direction.
LIMIT X–	The X-axis cannot move any farther in the minus direction.
LIMIT Y+	The Y-axis cannot move any farther in the plus direction.
LIMIT Y–	The Y-axis cannot move any farther in the minus direction.
LIMIT Z+	The Z-axis cannot move any farther in the plus direction. (3-axis model only)
LIMIT Z–	The Z-axis cannot move any farther in the minus direction. (3-axis model only)

If one of these messages appears while the table is moving, then the table cannot advance any farther. In this case, move the table in the opposite direction that it had been moving.

Note

The stopping position is affecting by the movement rate of the machine. For slow feed rates, the machine stops just before the Software Limit or Safe Zone. For fast feed rates, the machine will enter the Software Limit or Safe Zone. If the machine stops just before the Software Limit or Safe Zone, the screen will not show a message such as "LIMIT X+". When moving with the Manual Pulse Wheel, the machine stops as soon as enter the Software Limit or Safe Zone.

7. Basic Operation

When the power is turned on, the NU20 first enters Manual mode. The basic operations for Manual mode are described here. Manual mode can be mainly divided into three functions: DRO function, Jog Feed function, and Simple Machining function (LINEAR MILL, CIRCLE MILL).

7-1. DRO Function

In Manual mode, the same operations as a counter can be performed. The major functions are given below.

ABS Coordinate Settings:

ABS coordinates are absolute position coordinates from the datum point. These coordinates are also called workpiece coordinates. (In ABS, the zero position is called the datum point.) Pressing the ABS key allows you to enter numbers for the ABS coordinates. Movement of the machine by program or MDI commands is performed according to the workpiece coordinates.

INC Coordinate Settings:

INC coordinates are coordinates expressed by the distance to a coordinate point from a given coordinate point. Pressing the INC key allows you to enter numbers in increments (INC) when setting. These coordinates cannot be used for program operation or MDI commands.

ABS/INC Display Selector:

This function allows you to switch the display for all axes or individual axes between ABS or INC.

Inch/mm Selector:

This function allows you to switch the unit display between either inches or mm. Once the units are changed, these changes also affect the program and MDI commands until the units are changed again.

1) Key Description

The key function for the DRO operation, refer to "3-2. Name and Function of Each Part".

1. Check at the top left of the screen that the system is in Manual mode. If not,
2. Press the MODE key.
3. Press the F1 (MANUAL) key.
4. The system enters Manual mode.
- The function keys shown below can be used.

<	FEED ↑	FEED SET	FEED ↓	OVER RIDE ↑	100%	OVER RIDE ↓	MOVE ABS 0	REF RETURN	>
<	LINER MILL	CIRCLE MILL	SPINDLE	SAFE ZONE	TOUCH SENSOR		INCH/MM		>

2)   Entering numeric values

1. Entering Manual mode

2. Using ABS to enter the numeric values

3. Using INC to enter the numeric values

4. Selecting ABS or INC display

5. Selecting Inch or mm display

---

1. Entering Manual mode

Check at the top left of the screen that the system is in Manual mode. If not,

1.1   Press the [MODE] key.

1.2   Press the [F1] (MANUAL) key.

2. Entering the numeric values in ABS

Reset the absolute value of X-axis to zero.

2.1   Press the [X] key. The X-axis display then waits for input.

2.2   Press the [ABS] key.

2.3   The X-axis display on the screen becomes **X 0.0000 ABS**, and the zero point for the ABS coordinates is made.

2.4   Entering the desired value for the Y-axis

2.5   Press the [Y] key. The Y-axis display then waits for input.

2.6   Use the [0]-[9] numeric keys to enter the desired value.

2.7   Press the [ABS] key.

2.8   The Y-axis display on the screen becomes the entered value, and the ABS coordinate value is displayed.

3. Entering the numeric values in INC

Reset the incremental value of X-axis to zero.

3.1   Press the [X] key. The X-axis display then waits for input.

3.2   Press the [INC] key.

3.3   The X-axis display on the screen becomes **X 0.0000 INC**, and the zero point for the INC coordinates is made.

3.4   Entering the desired value for the Y-axis

3.5   Press the [Y] key. The Y-axis display then waits for input.

3.6   Use the [0]–[9] numeric keys to enter the desired value.

3.7   Press the [INC] key.

3.8   The Y-axis display on the screen becomes the entered value, and the incremental coordinate value is displayed.

4. Selecting ABS or INC display

Switching the all axes display to either ABS or INC

4.1   Press the [ABS] or [INC] key.

4.2   Each time when the key is pressed, all of the X, Y and Z axes will be changed to ABS or INC mode.

5. Selecting Inch or mm unit display

Select the unit between inch or mm for the display

5.1   Press the [←] or [→] function key.

5.2   Press the [F7] (inch/mm) key.

5.3   Each time when the key is pressed, the display units change to either inch or mm. These units are displayed at the top right of the screen.

Note
The machine is always set to inch mode when the power is turned on.

7-2.   Jog Feed Function

The Jog Feed function is used during Manual mode.
This function uses key operation to move the machine table by driving the servo motor. This function can also be used in the JOGMPG unit. The feed rate is changed by the function keys.

1)   NU20 Key Description and Examples of Operation

Jog Feed

Table movement direction

⬆ : Move backward (Y- direction)

Move left (X+ direction): ⬅   ➡ : Move right (X- direction)

⬇ : Move forward (Y+ direction)

∧ : The Z-axis quill rises (3-axis model only)

∨ : Z-axis quill descends (3-axis model only)

RAPID : Pressing this key while one of the above keys are held down will change the feed

rate to rapid traverse.

FEED ↑ (F1) key: The feed rate increases in 0.1 inch/min increments.

FEED ↓ (F3) key: The feed rate decreases in 0.1 inch/min increments.

FEED SET key (F2) and 0 - 9 and ENTER : These keys can be used to enter

the feed rate directly.

OVERRIDE

This function allows fine adjustment of the movement rate set by FEED SET. The movement rate can be adjusted by every 10% units from 0 to 200%. At the 100% setting, the machine moves at the movement rate shown in JOG FEED. The machine stops at 0%. In rapid traverse, the rate will not increase when set to 100% or more.

OVERRIDE ↑ key (F4): The feed rate increases in 10% increments.

OVERRIDE ↓ key (F6): The feed rate decreases in 10% increments.

100% key (F5): The feed rate is set to 100%.

MOVE ABS 0

This function automatically moves the machine table X-axis and Y-axis to the workpiece coordinate zero point (zero position in ABS).

MOVE ABS 0 key (F7): The machine table X-axis and Y-axis move simultaneously

to the workpiece coordinate zero point by rapid traverse.

2) JOGMPG Unit Operation Examples

1. Press the |MODE| key to display the Mode Selection screen.

2. Press the | F1 | (MANUAL) key. The system enters Manual mode.
   JOGMPG Unit can be performed in Manual mode.

3. Set the Axis Select switch to JOG.
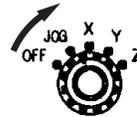
4. The indicator lamp lights up to show that the JOGMPG Unit can be used. In this setting, the same operations as NU20 Jog Feed can be performed.

Indicator lamp
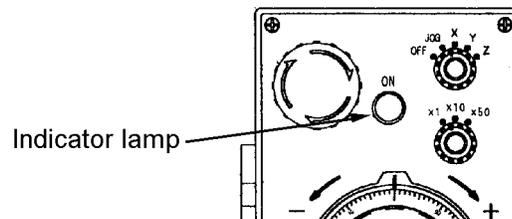
5. Setting the Axis Select switch to one of X, Y, or Z allows you to move the table (or Z-axis) for the corresponding axis by turning the Manual Pulse Wheel by hands. Since Jog Feed is enabled in this state, this function should be used too. For 2-axis models, the Z-axis selection is disabled. No axes can be moved at this setting.

6. Align the Magnification switch to x1, x10, or x50. For each graduation of the Manual Pulse Wheel, the table moves 0.0001 inch (0.001 mm) for x1, 0.001 inch (0.01 mm) for x10, and 0.005 inch (0.05 mm) for x50.

Note
For some machines, the inch mode may be set to move 0.0025 inches for x50.
The x50 will be enable after set the reference point return operation.

7. The table moves clockwise in the plus direction and counterclockwise in the minus direction.

• Machine Table Movement
If reference point return has been performed, machine movement is set by Software Limit point. The table starts to decelerate just before Software Limit point, and further decelerates at the Software Limit position to finally come to a stop.
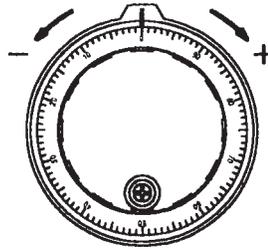When deceleration starts, SLOW X+ is displayed in the Status display at the top of the screen. Also, if the Safe Zone is set, the table stops suddenly when the Safe Zone area is entered. In either case, the messages shown below are displayed in the Status display at the top of the screen.

LIMIT X+   The X-axis cannot move any farther in the plus direction.
LIMIT X–   The X-axis cannot move any farther in the minus direction.
LIMIT Y+   The Y-axis cannot move any farther in the plus direction.
LIMIT Y–   The Y-axis cannot move any farther in the minus direction.
LIMIT Z+   The Z-axis cannot move any farther in the plus direction.
            (3-axis model only)
LIMIT Z–   The Z-axis cannot move any farther in the minus direction.
            (3-axis model only)

Note
In case that the feed rate is too slow, the machine may stop just before the limit setting. In this case, the above LIMIT message is not displayed.

⚠ **Caution**
When the Overtravel Unit is not connected or when the Software Limit has not set before reference point return, the machine table can be moved to the end of machine stroke. As a result, the table can be moved to the machine stroke end. However, this can damage the machine some cases, and so be sure to perform reference point return and set the Software Limit as a safety measure. For machines with no Zero Point Sensor (option), use the machine hand wheel when moving the machine for setting the Software Limit settings to minimize the chances of an accident.
Note
If the reference point return operation is not performed, the Software Limit settings are not made, and rapid traverse at JOG cannot be performed (see the section on zero point settings). Once the power is turned on, be sure to always perform the reference point return operation.
Note
If the machine passes over the Safe Zone when performing rapid traverse at JOG, the machine stops suddenly. In some cases, such operations can damage the machine.

7-3.   Linear Mill Function

This function allows linear machining to be performed easily on the XY, YZ, or ZX plane using the JOG key or the Manual Pulse Wheel. This function is especially useful when performing chamfering operations manually. This function can perform the following two types of machining operations.
*   Linear machining joining the current position and a given end point.
*   Linear machining at a given angle from the current position.

Note
The YZ and ZX planes can be used only in 3-axis models.

1)   Key Description

The keys for Linear Mill are described below.

1.   Press the MODE key.

2.   Press the F1 (MANUAL) key.

3.   Press the ← or → function key.

4.   Press the F1 (LINEAR MILL) key.

5.   The system enters Linear Mill mode. The following function keys can be used.

LINE ANGLE	END POINT		INFEED STEP		PLANE SELECT		PAGE BACK	

Key Description	Function
LINE ANGLE:	
	The angle for the direction that you want to move can be designed from the current position when the X+ direction is taken as 0 degree. The angle can be designated by every 0.001-degree.
END POINT:	
	The end point position that you want to move to can be designated from the current point. The machine stops automatically at the designated position.
INFEED STEP:	
	The machine can be moved plus 90 degrees or minus 90 degrees with respect to the line at the angle designated in LINE ANGLE. The movement starts from the current position when the key is pressed.
PLANE SELECT:	
	Specific plane, on which 2 axes will be used, can be selected from X – Y, Z – X and Y – Z planes.

2) LINEAR MILL operation procedure

    1. Entering Linear Mill mode
    2. Designating the end point
    3. Moving
    4. Designating the angle
    5. Moving
    6. Moving at a right angle

1. Entering Linear Mill mode

1.1   Press the MODE key.

1.2   Press the F1 (MANUAL) key.

1.3   Press the ← or → function key.

1.4   Press the F1 (LINEAR MILL) key.

1.5   The system enters Linear Mill mode.

2. Designating the end point

2.1   Make sure that the current position is adequate as the movement (or machining) starting point. If not, use Jog Feed or the Manual Pulse Wheel to move to the desired starting point.

2.2   Press the F2 (END POINT) key.

2.3   The End Point Input screen appears.

2.4   Press the X key.

2.5   Use the 0 - 9 keys to enter the numerical value for the X-axis.

2.6   Press the ENTER key to fix the numerical value into the program.

2.7   Press the Y key.

2.8   Use the 0 - 9 keys to enter the numerical value for the Y-axis.

2.9   Press the ENTER key to fix the numerical value into the program.

2.10 A line is drawn on the screen from the current position to the designated direction. At the same time, the angle seen from the X+ direction is displayed as A on the screen.
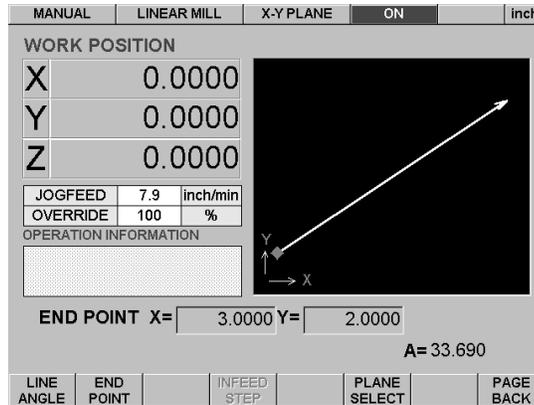
3. Moving

3.1 Use the ← key in the numeric keypad to move the tool (red circle on the screen) in the direction of the arrow on the screen. Use the → key to move in the direction opposite the arrow.

3.2 If the tool is moved in the direction of the arrow (designated position), then the tool stops automatically at the designated point.

3.3 If the tool is moved in the opposite direction of the arrow (designated position), then the continues to move until the key operations are performed instructing it to stop.

Note
The same operations can be performed using the Manual Pulse Wheel on the JOGMPG Unit.

3.4 Set the Axis Select switch to X.

3.5 When the Manual Pulse Wheel is turned in the positive direction, the tool moves in the direction of the arrow on the screen. When it is turned in the negative direction, the tool moves in the opposite direction of the arrow on the screen.

4. Designating the angle

4.1 By designating the angle, the tool can be moved from the current point in the direction of the desired angle.

4.2 Press the F1 (LINE ANGLE) key.

4.3 The system is in Numeric Input mode. Use the numeric keys to enter the desired numerical values.

4.4 Press the ENTER key.

4.5   The entries appear on the screen once they are made.

MANUAL	LINEAR MILL	X-Y PLANE	ON	inch

WORK POSITION

X   0.0000
Y   0.0000
Z   0.0000

JOGFEED	7.9	inch/min
OVERRIDE	100	%

OPERATION INFORMATION

90
180
0
270
Y
X

ANGLE(DEGREES)=   60.000   **L=** 0.0000

                                            D= 0.0000

LINE ANGLE	END POINT		INFEED STEP		PLANE SELECT		PAGE BACK

5. Moving

5.1   Use the ⬅ key in the numeric keypad to move the tool (red circle on the screen) in the direction of the arrow on the screen. Use the ➡ key to move in the direction opposite the arrow.

        Note

        The same operations can be performed using the Manual Pulse Wheel on the JOGMPG Unit.

5.2   Set the Axis Select switch to either X.

5.3   When the Manual Pulse Wheel is turned in the positive direction, the tool moves in the direction of the arrow on the screen. When it is turned in the negative direction, the tool moves in the opposite direction of the arrow on the screen.

5.4   The distance moved on the line is displayed as L on the screen.

6. Moving at a right angle

6.1   The tool can be moved plus 90 degrees or minus 90 degrees with respect to the line at the angle designated in LINE ANGLE. The movement start point is the current position of pressing the INFEED STEP key.

6.2   Stop the machine movement at the point where you want to move at a right angle.

6.3   Press the F4 key.

6.4   The right angle direction appears as blue arrow on the screen.

6.5   Use the ⬅ key in the numeric keypad to move the tool (red circle on the screen) in the direction of the arrow on the screen. Use the ➡ key to move in the direction opposite the arrow.

        Note
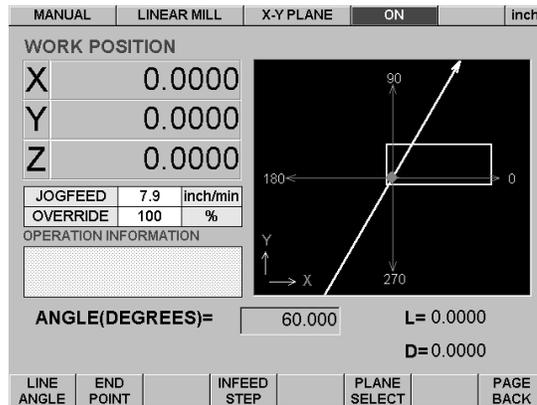
        The same operations can be performed using the Manual Pulse Wheel on the JOGMPG Unit.

6.6   Set the Axis Select switch to X.

6.7   When the Manual Pulse Wheel is turned in the positive direction, the tool moves in the direction of the arrow on the screen. When it is turned in the negative direction, the tool moves in the opposite direction of the arrow on the screen.

6.8   The distance moved on the right angle line is displayed as D on the screen.
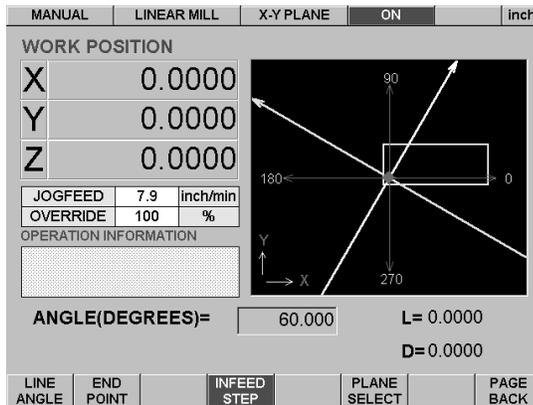
| MANUAL | LINEAR MILL | X-Y PLANE | ON | | inch |

WORK POSITION

X   0.0000
Y   0.0000
Z   0.0000

| JOGFEED | 7.9 | inch/min |
| OVERRIDE | 100 | % |

OPERATION INFORMATION

ANGLE(DEGREES)=   60.000   **L=** 0.0000

D= 0.0000

| LINE ANGLE | END POINT | | INFEED STEP | | PLANE SELECT | | PAGE BACK |

Note
If you switch to Line Angle after setting XY for End Point, the system automatically calculates the line angle at End Point and makes the settings for Line Angle. This function is useful when the chamfering position is shown without the chamfering angle on the drawing. The current position is displayed on the screen as a red circle.

Note
Movement in Linear Mill does not accelerate or decelerate when movement starts and stops.

⚠ **Caution**
The machine may be damaged if the Jog Feed setting (feed rate) is too fast.

7-4. Circle Mill Function

This function allows circular machining by the servo motor to be performed easily on the XY plane using the JOG key or the Manual Pulse Wheel. This function is useful when performing corner R machining. This function can perform the following two types of machining operations.
* Circular machining joining the current position and a given end point.
* Circular Machining for drawing a circle from the current position and a point designated as the circle center.

Note
The YZ and ZX planes can be used only in 3-axis models.

1) Key Description

The keys for Circle Mill are described below.

1. Press the MODE key.

2. Press the F1 (MANUAL) key.

3. Press the ← or → function key.

4. Press the F2 (CIRCLE MILL) key.

5. The system enters Circle Mill mode. The following function keys can be used.

CENTER POINT	END POINT	1/4 CIRCLE	ARC SELECT	INFEED STEP		PLANE SELECT	PAGE BACK	

Key Description	Function

CENTER POINT:

This function is used to move in a circular movement. The center point is designated by CENTER POINT. The radius length of circular is determined from the current point to CENTER POINT value.

END POINT:

This function allows you to designate the circle to be machined using the radius and end point from the current position. The tool automatically stops at the designated position.

1/4 CIRCLE:

This function allows you to select a quarter circle using the designated radius R from the current position. You can select from four choices.

ARC SELECT:

This function selects the center point of the arc drawn when an arc is designated with END POINT.

INFEED STEP:

This function is used to change the radius of the arc.
The tool can be moved plus 90 degrees or minus 90 degrees from the tangent of the arc designated at CENTER POINT. The movement start point is the current position of pressing the INFEED STEP key.

PLANE SELECT:

Specific plane, on which 2 axes will be used, can be selected from X – Y, Z – X and Y – Z planes.

2) CIRCLE MILL operation procedure

1. Entering Circle Mill mode
2. Designating the center point of the circle passing through the current position
3. Moving
4. Moving along with radius direction and setting concentric circle
5. Designating the arc reaching to the target position from the current point at a radius R
6. Selecting either clockwise (CW) or counterclockwise (CCW) as the advancement direction
7. Moving
8. Selecting the quarter circle perpendicular to the X, Y coordinate with the current position as the starting point

---

1. Entering Circle Mill mode

1. 1   Press the [MODE] key.

1. 2   Press the [F1] (MANUAL) key.

1. 3   Press the [◀] or [▶] function key.

1. 4   Press the [F2] (CIRCLE MILL) key.

1. 5   The system enters Circle Mill mode.

2. Designating the center point of the circle passing through the current position

2.1   Make sure that the current position is adequate as the movement (or machining) starting point. If not, use Jog Feed or the Manual Pulse Wheel to move to the desired starting point.

2.2   Press the [F1] (CENTER POINT) key.

2.3   The Circle Center Point (X, Y) Input screen appears.

2.4   Press the [X] key.

2.5   Use the [0]–[9] keys to enter the numerical value of center point for the X-axis.

2.6   Press the [ENTER] key to fix the numerical value into the program.

2.7   Press the [Y] key.

2.8   Use the [0]–[9] keys to enter the numerical value of center point for the Y-axis.

2.9   Press the [ENTER] key to fix the numerical value into the program.

2.10 A circle is drawn on the screen passing through the current position (red point). At the same time, the center value (R) and the current position angle seen from the center are displayed. This angle is measured with the X+ direction taken as 0 degrees.
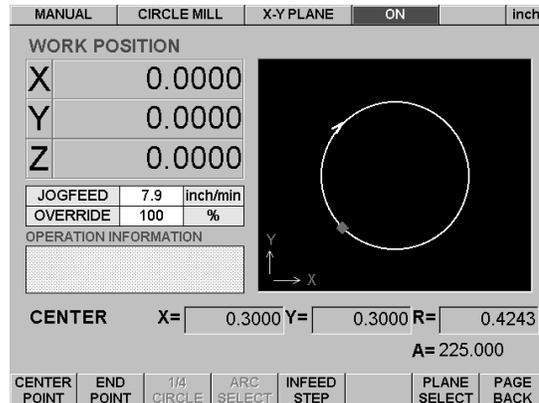
3. Moving

3.1 Use the ◄ key in the numeric keypad to move the tool (red circle on the screen) in the clockwise direction, and use the ► key to move in the counterclockwise direction.

The movement direction is the direction that the red circle moves on the screen. For the JOG key and Manual Pulse Wheel, the positive direction moves the tool clockwise, and the negative direction moves the tool counterclockwise.

4. Moving along with radius direction and setting concentric circle

The tool can be moved plus along with radius direction. By this operation you can set concentric circle. The movement starts from the current position when the key is pressed.

4.1 Stop at the point where you want to move at a right angle.

4.2 Press the F5 (INFEED STEP) key.

4.3 The right angle direction appears as blue on the screen.

4.4 Use the ◄ key in the numeric keypad to move the tool (red circle on the screen) in the direction of the arrow on the screen (the direction where the radius gets larger), and use the ► key to move in the direction opposite the arrow (the direction where the radius gets smaller).

4.5 The same operations can be performed using the Manual Pulse Wheel on the JOGMPG Unit.

4.6 Change the setting for the Axis Select switch to X.

4.7 When the Manual Pulse Wheel is turned in the positive direction, the tool moves in the direction of the arrow on the screen (the direction where the radius gets larger). When it is turned in the negative direction, the tool moves in the opposite direction of the arrow on the screen (the direction where the radius gets smaller).

4.8 The current radius is displayed as R on the screen.
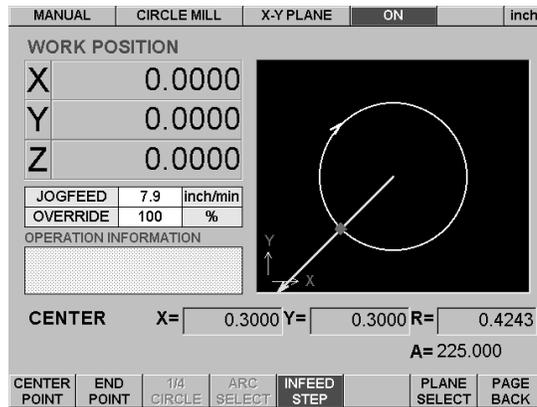
5. Designating the arc reaching to the target position from the current point at a radius R

First, designate the target position.

5.1   Press the ☐ F2 ☐ (END POINT) key.

5.2   The system is in Numeric Input mode. Use the numeric keys to enter the desired numerical values.

5.3   Enter the numeric value for X.

5.4   Press the ☐ ENTER ☐ key.

5.5   Enter the numeric value for Y.

5.6   Press the ☐ ENTER ☐ key.

5.7   Enter the numeric value for R.

5.8   Press the ☐ ENTER ☐ key.

5.9   The results of center point and drawings appear on the screen.
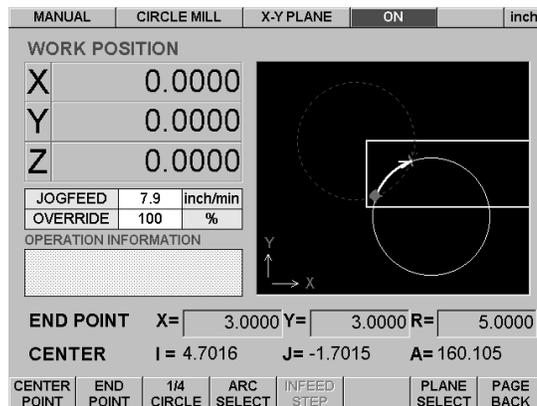
6. Selecting either clockwise (CW) or counterclockwise (CCW) as the advancement direction

    6.1   Press the ⎣ F4 ⎦ (ARC SELECT) key.

    6.2   The arc center position to the target point is switched each time that this key is pressed.

7. Moving

    7.1   Once the center position is determined, move the tool.

    7.2   Use the ⬅ key in the numeric keypad to move the tool (red circle on the screen) in the clockwise direction. Use the ➡ key to move in the counterclockwise direction. For the JOG key and Manual Pulse Wheel, the positive direction moves the tool clockwise, and the negative direction moves the tool counter-clockwise.

            Note
The same operations can be performed using the Manual Pulse Wheel of the JOGMPG Unit.

    7.3   Change the setting for the Axis Select switch to X.

    7.4   When the Manual Pulse Wheel is turned in the positive direction, the tool moves clockwise. When it is turned in the negative direction, the tool moves counterclockwise.

    7.5   The current position angle is displayed as A on the screen.

8. Selecting the quarter circle perpendicular to the X, Y coordinate with the current position as the starting point.

8.1 Press the ☐ F2 ☐ (END POINT) key.

8.2 Press ☐ ENTER ☐☐ ENTER ☐ to prepare for the input of R.

8.3 Enter the desired radius for the circle.

8.4 Press the ☐ ENTER ☐ key.

8.5 Enter the appropriate values for X and Y at this time.

8.6 Press the ☐ F3 ☐ (1/4 CIRCLE) key.

8.7 A quarter circle with radius R is drawn through the current position. The following information is displayed at this time:
X, Y = Target position; R = Radius; I, J = Center of circle; A = Angle at current position (with the X+ direction taken as 0 degrees).

8.8 Next, press the ☐ F3 ☐ (1/4 CIRCLE) key. Four types of circles are shown in order. Select the desired one.

8.9 Press the ☐ F4 ☐ (ARC SELECT) key to select the three quarters.

Note
The value for A is calculated based on the ABS position and R. Therefore, it may not be able to be displayed in 0.001 degree units in some cases. For example, 0.003 may be followed by 0.001.

Note
I, J indicates the distance to the center from the current position. I, J is calculated based on the ABS position and R. Therefore, there may be a slight error in some cases.

Note
In some cases, there is a slight error when CENTER POINT is displayed automatically. CENTER POINT is calculated based on the ABS position and R for XY.

Note
In some cases, there is a slight error when R is displayed automatically. R is calculated based on the ABS position for XY.

7-5.   Spindle Function

This function is available with optional NZ31 (Spindle control unit) and by supplying electric power for spindle motor through NZ31.
With this function SPINDLE FWD or SPINDLE REV of internal contacts of NZ31 will be driven.

Note

For spindle control with NZ31 main spindle switch on the machine must be set to ON.

1)   Key Description

The keys for Spindle are described below.

1. Press the $\boxed{\text{MODE}}$ key.

2. Press the $\boxed{\text{F1}}$ (MANUAL) key.

3. Press the $\boxed{\Leftarrow}$ or $\boxed{\Rightarrow}$ function key.

4. Press the $\boxed{\text{F3}}$ (SPINDLE) key.

5. The system enters Spindle mode. The following function keys can be used.

	SPINDLE FWD	SPINDLE REV	SPINDLE STOP					PAGE BACK	

Key Description	Function
SPINDLE FWD:	
	Spindle motor rotates in foward direction.
SPINDLE REV:	
	Spindle motor rotates in reverse direction.
SPINDLE STOP:	
	Spindle motor stops.

7-6. Selection Function between 2 axis and 3 axis

This function is valid only for 3 axis model (NU20-30) and allows the selection between 2 axis and 3 axis mode with key operation. Change to 2 axis mode is needed when it is necessary on 3 axis model to operate Z axis manually.

1) Key Description

Procedure to select between 2 axis and 3 axis as follows:

1. Press the MODE key.

2. Press the F5 (2AX / 3AX) key

   Pressing F5 key selects 2 axis mode or 3 axis mode alternately

8. Reference point functions

This section describes the functions and operating procedure for the reference point. When the zero point sensor is installed, the NU20 automatically performs reference point return. If the Reference point sensor is not installed, a manual Reference Point Return mode can be used. Even when there is no zero point sensor, the reference point return operation must still be performed in order to set the Software Limit.

1) Key Description

The keys for the reference point function are described below.

1. Press the MODE key.

2. Press the F1 (MANUAL) key.

3. Press the F8 (REF RETURN) key.

4. The system enters Reference point return mode. The function keys shown below can be used.

	ALL AXES	X AXIS	Y AXIS	Z AXIS	POS RECOVE	POS SELECT	ABORT	PAGE BACK	

Key Name	Function
ALL AXES:	
	This function is used when setting the reference point. Pressing this key automatically performs reference point return in the order Z → X → Y. If the zero point sensor is not installed, instruction messages appear on the screen and wait for your input.
X AXIS:	
	This function is used for making the X-axis reference setting. Reference point return can only be performed for the X-axis.
Y AXIS:	
	This function is used for making the Y-axis reference setting. Reference point return can only be performed for the Y-axis.
Z AXIS:	
	This function is used for making the Z-axis reference setting. Reference point return can only be performed for the Z-axis. This function is not included in 2-axis models.
POS RECOVE:	
	This function recovers all the coordinate values from when the power was last turned off. In 2-axis models, only the XY coordinates are covered.
POS SELECT:	
	The display coordinates can be switched between WORK POSITION and MACHINE POSITION. MACHINE POSITION is zero where the reference point return is performed.
ABORT:	
	This function is used to abort the reference point return operation while it is in progress.

8-1.   When the Zero Point Sensor is Provided

Once reference point return is performed, the workpiece coordinate system at power off and the machine coordinate system are automatically recovered.

1)   Reference point return Operation Procedures

1.   Performing reference point return for all axes

2.   Performing reference point return only for a designated axis

---

1. Performing reference point return for all axes

1.1   Press the $\boxed{\text{F1}}$ (ALL AXES) key.

1.2   Raise the Z-axis (quill) to a safe position by hand (for 2-axis models only).

1.3   Press the $\boxed{\text{F4}}$ (Z AXIS) key or the $\boxed{\text{Z}}$ key. The position where the key is pressed is recorded as the reference point for the Z-axis.

1.4   Reference point return is automatically performed for the X-axis and Y-axis, in that order.

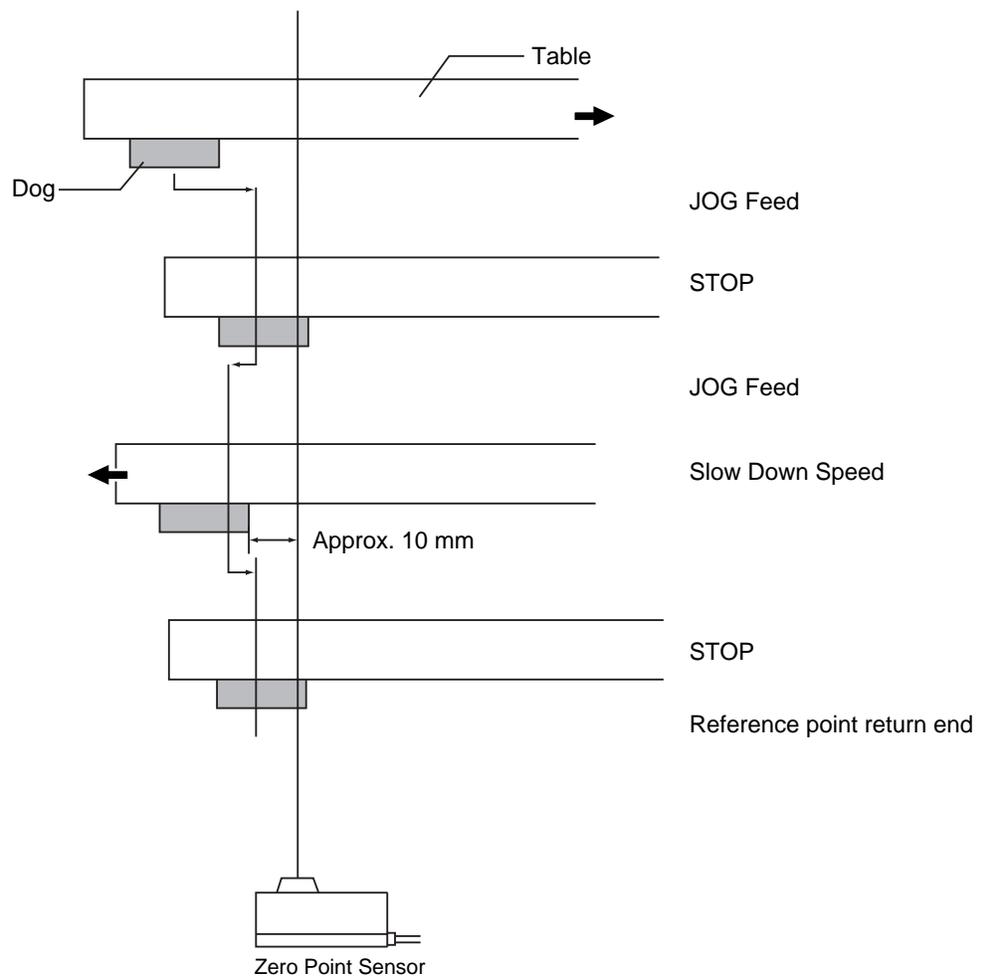The machine moving is as follows.

Machine Operation

1.   Press the $\boxed{\text{F4}}$ (Z AXIS) key. The X-axis moves in the X- direction (the table moves right) at the rate set by JOG FEED.

2.   When the reference point dog at the left edge of the table is reached, the movement direction reverses and the machine starts moving in the X+ direction.

3.   When the machine has advanced about 10 mm from the reference point dog, it reverses direction again and starts moving in the X– direction. This time, the feed rate proceeds at a rate suitable for reference point return.

4.   The machine stops automatically once the reference point dog is reached, and this position is set as the zero in machine coordinates. The ABS coordinates (workpiece coordinates) are set to zero at this time.

5.   Next, the table Y-axis moves in the Y– direction (the table moves back) at the rate set in JOG FEED.

6.   When the reference point dog at the back of the table is reached, the movement direction reverses and the machine starts moving in the Y+ direction.

7.   When the machine has advanced about 10 mm from the reference point dog, it reverses direction again and starts moving in the Y– direction. This time, the feed rate proceeds at a rate suitable for reference point return.

8.   The machine stops automatically once the zero point dog is reached, and this position is set as the zero in machine coordinates. The ABS coordinates (workpiece coordinates) are set to zero at this time.

Note
When the X AXIS or Y AXIS key is pressed, the machine performs reference point return only for the respective axis and then stops.

2. Performing reference point return only for a designated axis

   2.1   Raise the Z-axis (quill) to a safe position by hand (for 2-axis models only).

   2.2   Press the [F4] (Z AXIS) key, and then either press the [F4] (Z AXIS) key again

         or the [Z] key.

   2.3   Press either the [F2] (X AXIS) key or the [F3] (Y AXIS) key.

   2.4   Reference point return is automatically performed only for a designated axis.

Table

Dog

JOG Feed

STOP

JOG Feed

Slow Down Speed

Approx. 10 mm

STOP

Reference point return end

Zero Point Sensor

2) Operation procedure for recovering the ABS coordinate system at last power off

1. First, press the $\boxed{\text{F5}}$ (POS RECOVE) key. The POS RECOVE display is highlighted in blue.

   Note

   If the display does not turn blue, then the coordinate data at the last power off is no longer stored for some reason. In this case, the coordinate system cannot be recovered.

2. While the POS RECOVE display is blue, perform the Reference point return Operation Procedure in 1). The ABS coordinates at the last power off are then recovered.

## 8-2. When the Zero Point Sensor is Not Provided

The reference return operation is performed manually.

1) Reference point return Operation Procedure

1. Press the $\boxed{\text{F1}}$ (ALL AXIS) key.
2. Raise the Z-axis (quill) to a safe position by hand (for 2-axis models only).
3. Press the $\boxed{\text{F4}}$ (Z AXIS) key or the $\boxed{\text{Z}}$ key. The position where the key is pressed is recorded as the zero point for the Z-axis.
4. Move the machine table to the right edge (X–) using JOG or the machine hand wheel. As the tool approaches the table edge, feed it slowly using Manual mode to prevent damage to the table. When the machine stroke end is reached, press the

   $\boxed{\text{F2}}$ (X AXIS) key or the $\boxed{\text{X}}$ key. The position where the key is pressed is recorded as the zero point for the X-axis.
5. Move the machine table back (Y–) using JOG or the machine hand wheel. As the tool approaches the table back, feed it slowly using Manual mode to prevent damage to the table. When the machine stroke end is reached, press the $\boxed{\text{F3}}$ (Y

   AXIS) key or the $\boxed{\text{Y}}$ key. The position where the key is pressed is recorded as the zero point for the Y-axis.

   Note

   The zero position for the machine coordinate system is different depending on whether a zero point sensor is used or not. However, the range for the Software Limit which is set at the same time is the same for both cases. This is because when the NU20 is installed on the machine, the system parameters were made based on whether the zero point sensor was used or not.

45
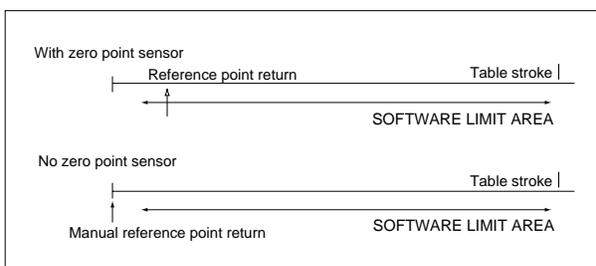
9. MDI Mode

In MDI mode, cutting work can be performed simply by selecting from any drilling pattern and preset machining shapes to desired patterns and shapes. The shapes which can be machined can be selected from seven groups: Drill, Face Mill, Island Mill, Pocket Mill, Frame Mill, Ring Mill and Line Arc.

This section describes the basic operating procedures using typical examples. For descriptions on the specific input data, press the HELP key on the NU20 keyboard to display more detailed information.

1) Key Description

The keys for MDI mode are described below.

1. Press the MODE key.
2. Press the F3 (MDI) key.
3. The system enters MDI mode.

DRILL	FACE MILL	ISLAND MILL	POCKET MILL	FRAME MILL	RING MILL	LINE ARC	PAGE BACK

Key Name	Function

DRILL:

Seven types of drilling patterns can be selected.

FACE MILL:

This function can easily work the workpiece surface into a flat surface.

ISLAND MILL:

This function can leave five types of shapes in the workpiece.

POCKET MILL:

This function can cut four types of impression shapes into the workpiece.

FRAME MILL:

This function can open holes with four types of shapes into the workpiece.

RING MILL:

This function can cut grooves with two types of shapes into the workpiece.

LINE ARC:

This function commands the machining of lines and arcs, and positioning.

4. Select the desired event from the [F1] (DRILL) to [F7] (LINE ARC) keys.
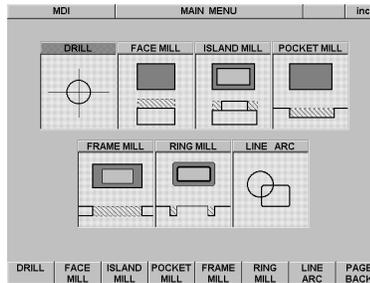
5. Next, a detailed menu appears. Select the desired machining process from each menu.
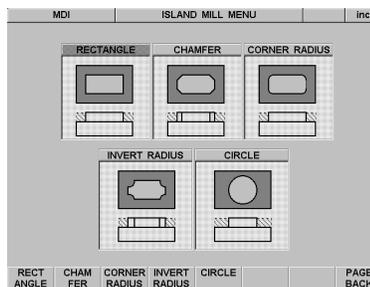
6. The system enters Input mode. Each key in input mode is described below.
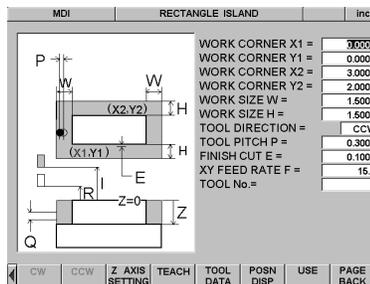
Z AXIS SETTING:

Enter the data for the Z axis. For 3-axis models, the Z-axis (quill) actually moves according to the commands. However, for 2-axis models, a message appears instructing you to move the Z-axis when it is supposed to be moved.

TOOL DATA:

TOOL DATA mode is for storing the necessary tool information during machining. The values entered here are used for the tool diameter compensation value during MDI machining.

POSN DISP:

This mode shows the WORK POSITION. At the same time, the tool names and tool diameters entered for the currently selected TOOL DATA are selected.

USE:

Press this key, to fix the input data for the MDI operation and starts machining preparation (Program Run mode).

7. By pressing ⎣F7⎦ (use) key after entering values the system enters the Program Run mode. Please refer to Section 10-3, Item 2 for the details of this mode.

    By pressing ⎣F6⎦ (START) key machining starts.
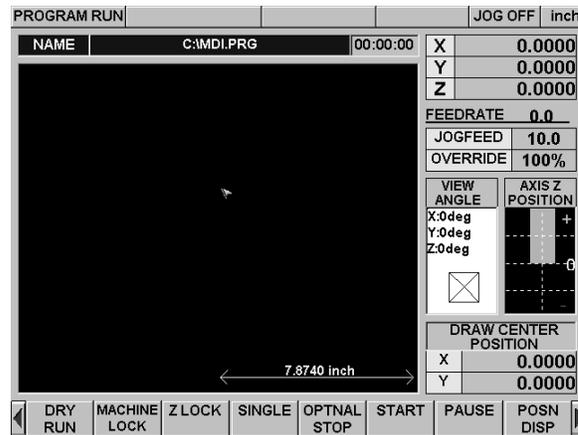
<	DRY RUN	MACHINE LOCK	Z LOCK	SINGLE	OPTNAL STOP	START	PAUSE	POSN DISP	>
<	SPINDLE FWD	SPINDLE REV	SPINDLE STOP						>
<	FEED ↑	FEED SET	FEED ↓	OVER RIDE↑	100%	OVER RIDE↓	GRAPH OPTION	PAGE BACK	>

8. When optional NZ31 is in use spindle function in forward and reverse directions is available with key operation. Press the ⬅ function key and then select the SPINDLE command.
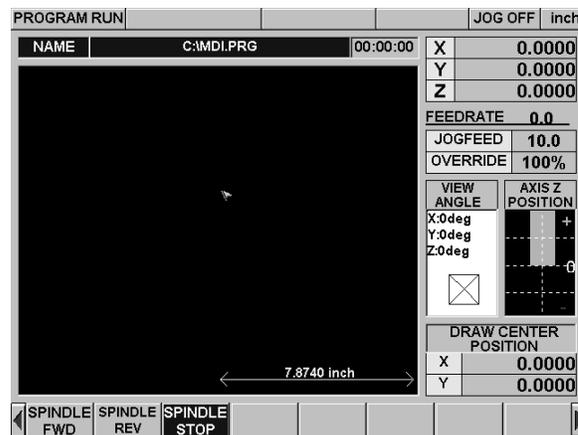
9. After machining is finished the system return to Input mode by pressing PAGE BACK function key.

9-1.   DRILL
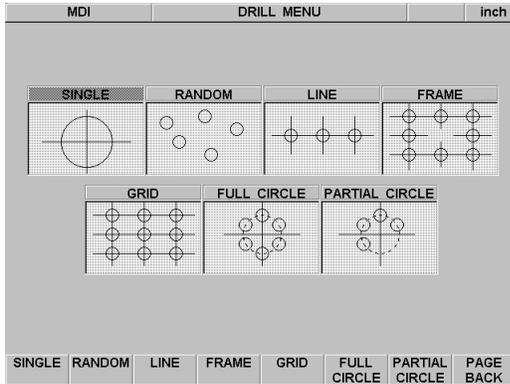
The desired drilling type can be selected using the F1 to F7 keys.
Press the PAGE BACK (F8 key) to return to the previous screen.
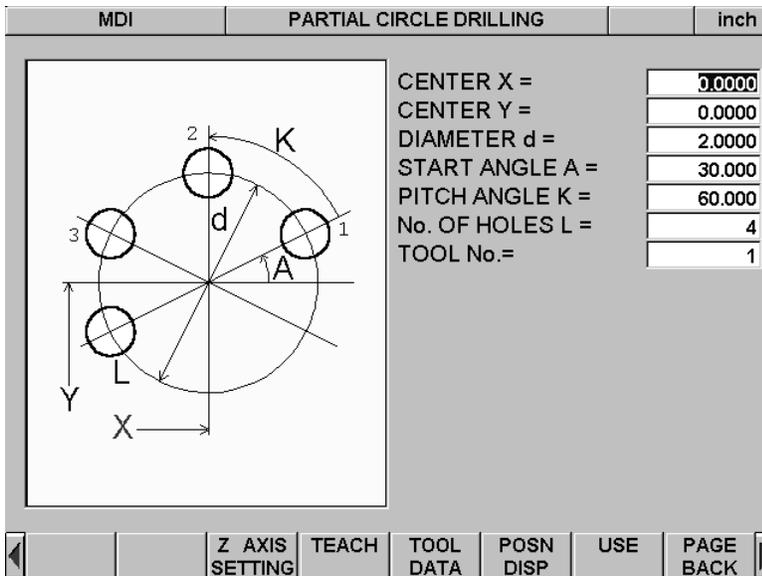
Typical Example

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

X, Y: Circle center
D: Circle diameter
A: By entering the angle (degrees), you can designate the hole position angle with respect to the X+.
L: Number of holes to be drilled
K: Even pitch angle of a hole

• Z AXIS SETTING

MDI	PARTIAL CIRCLE DRILLING		inch

CENTER X = 0.0000
CENTER Y = 0.0000
DIAMETER d = 2.0000
START ANGLE A = 30.000
PITCH ANGLE K = 60.000
No. OF HOLES L = 4

Z AXIS SETTING

DEPTH Z = -1.0000
INITIAL POINT I = 0.2000
SAFETY POINT R = 0.1000
RETURN POINT RP = R
STEP Q = 0.2500
Z FEED RATE FZ = 12.0
DWELL TIME P = 1.00
DEAD SPIND. RET = DISABLE
PECK RETURN RR = CHIP BRK

| Z AXIS SETTING | TOOL DATA | POSN DISP | USE | PAGE BACK |

The Z-axis can be controlled by using the Z AXIS SETTING. This is only enabled for the 3-axis control models. For 2-axis control models, instruction messages for Z-axis operation are displayed.

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.

Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.

Press the PAGE BACK (F8 key) to return to the previous screen.

Z: Depth to be cut
I: Tool initial point (if not entered, this is taken as the current position)
R: Cutting starting point (if not entered, this is taken as the same position as I)
RP: Return point after drilling the hole
Q: Depth to be cut in one step
FZ: Z-axis feed rate
P: Time for the tool to start the rising operation when the Z point is reached
DEAD SPIND. RET: When the spindle control unit (option) is installed, the spindle rotation can be instructed to return to the return point: RP after stopping. The ENABLE setting returns the Z-axis to the return point: RP after stopping. The DISABLE setting returns the Z-axis to the return point: RP with the spindle still spinning.
RR: Return point after each cutting feed step

Note
DEAD SPIND. RET command is available with optional NZ31 and by changing SYSTEM parameters.
At present, the system parameters have been set so that they cannot be changed by the user. To change them, please contact Sony precision Technology or your local dealer.
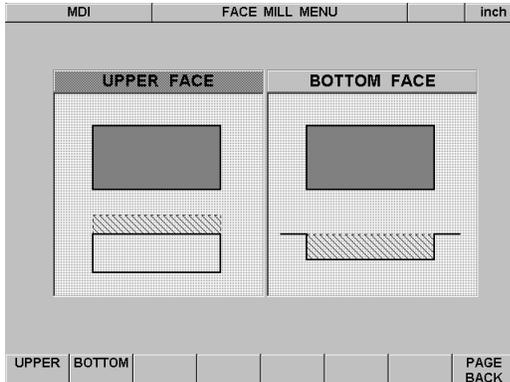
This function can easily work the workpiece surface into a flat surface.
Select the UPPER or BOTTOM setting.
Press the PAGE BACK (F8 key) to return to the previous screen.
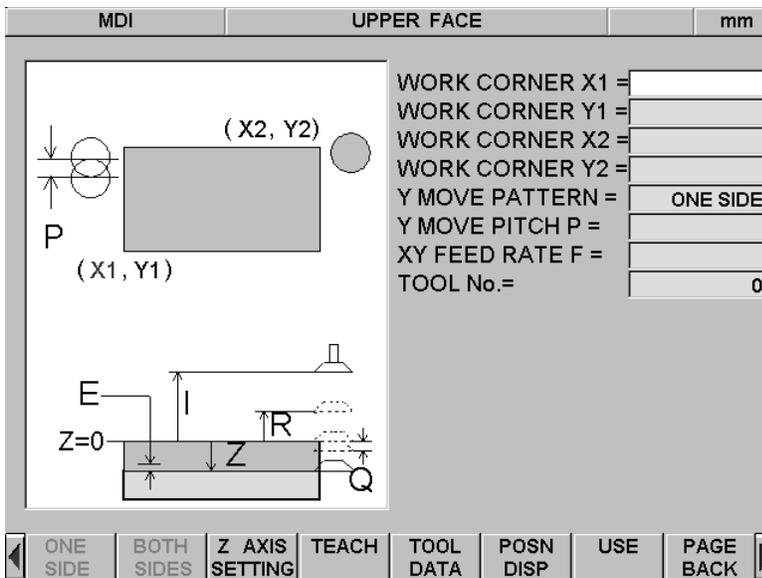
Typical Example

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
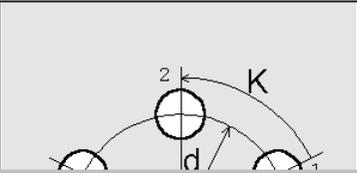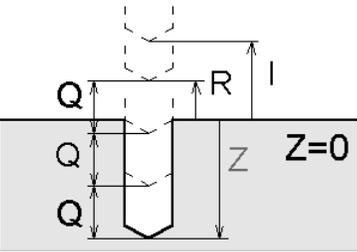Press the PAGE BACK (F8 key) to return to the previous screen.

X1, Y1, X2, Y2: Enter the coordinates for the opposite angles of the workpiece.
Y MOVE PATTERN: This is the tool movement directing during cutting. Select the ONE SIDE or BOTH SIDES setting. The tool movement direction is indicated by a blue line on the screen.
P: The STEP setting for the tool Y-axis is variable.
F: Feed rate for the X and Y axes during cutting

• Z AXIS SETTING

The Z-axis can be controlled by using the Z AXIS SETTING. This is only enabled for the 3-axis control models. For 2-axis control models, instruction messages for Z-axis operation are displayed.

Key entry method:

The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys. Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

Z: Depth to be cut
I: Tool initial point (if not entered, this is taken as the current position)
R: Cutting starting point (if not entered, this is taken as the same position as I)
Q: Depth to be cut in one step
FZ: Z-axis feed rate
E: Depth of cut for the finishing work

9-3.   ISLAND MILL

RECTANGLE   CHAMFER   CORNER RADIUS

INVERT RADIUS   CIRCLE

RECT ANGLE	CHAM FER	CORNER RADIUS	INVERT RADIUS	CIRCLE			PAGE BACK

This function can leave the desired types of shapes in the workpiece. The desired shape, drilling depth, and area to be cut can be selected. First, use the F1 to F5 keys to select the desired shape.
Press the PAGE BACK (F8 key) to return to the previous screen.

Typical Example

MDI	CHAMFER ISLAND	inch

P
W   W
(X2,Y2)   H
C
(X1,Y1)   H
E
R
I
Z=0   Z
Q

WORK CORNER X1 =   0.0000
WORK CORNER Y1 =   0.0000
WORK CORNER X2 =   3.0000
WORK CORNER Y2 =   2.0000
WORK SIZE W =   1.5000
WORK SIZE H =   1.5000
TOOL DIRECTION =   CW
TOOL PITCH P =   0.3000
FINISH CUT E =   0.1000
CHAMFER C =   0.3000
XY FEED RATE F =   15.0
TOOL No.=   1

CW	CCW	Z AXIS SETTING	TEACH	TOOL DATA	POSN DISP	USE	PAGE BACK

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

X1, Y1, X2, Y2: Coordinates for the opposite angles of the workpiece
W, H: The cutting range can be selected. (In this case, the machining shape is always left at the center of the cut-out piece.)
Z: Depth to be cut
I: Tool initial point (if not entered, this is taken as the current position)
R: Cutting starting point (if not entered, this is taken as the same position as I)
Q: Depth to be cut in one step
TOOL DIRECTION: This is the tool movement direction during cutting. You can select from clockwise (CW) and counterclockwise (CCW).
P: The STEP setting for the tool is variable.
E: Depth of cut for the finishing work
C: Chamfering amount for the workpiece
F: Feed rate for the X and Y axes during cutting
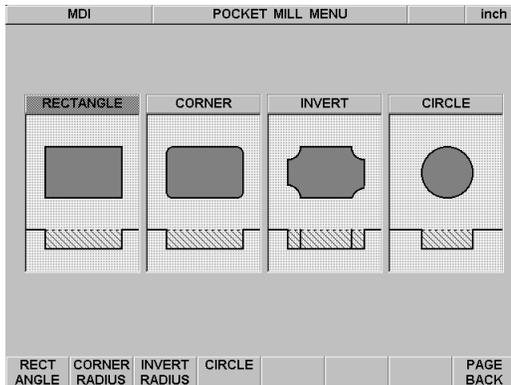
This function can cut the desired shapes into the workpiece. The desired shape and drilling depth can be selected. First, use the F1 to F4 keys to select the desired shape.
Press the PAGE BACK (F8 key) to return to the previous screen.

Typical Example

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

X1, Y1, X2, Y2: Coordinates for the opposite angles of the workpiece
TOOL DIRECTION: This is the tool movement direction during cutting. You can select from clockwise (CW) and counterclockwise (CCW).
P: The STEP setting for the tool is variable.
E: Depth of cut for the finishing work
r: Corner radius for the workpiece angle
F: Feed rate for the X and Y axes during cutting

• Z AXIS SETTING

The Z-axis can be controlled by using the Z AXIS SETTING. This is only enabled for the 3-axis control models. For 2-axis control models, instruction messages for Z-axis operation are displayed.

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

Z: Depth to be cut
I: Tool initial point (if not entered, this is taken as the current position)
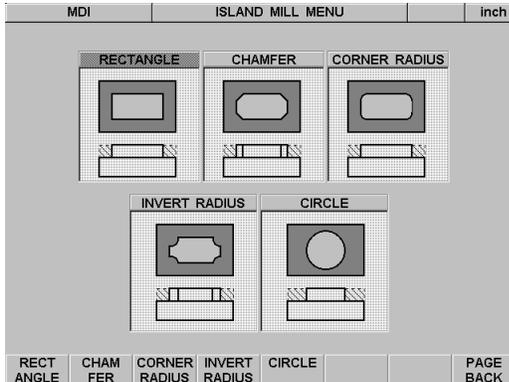R: Cutting starting point (if not entered, this is taken as the same position as I)
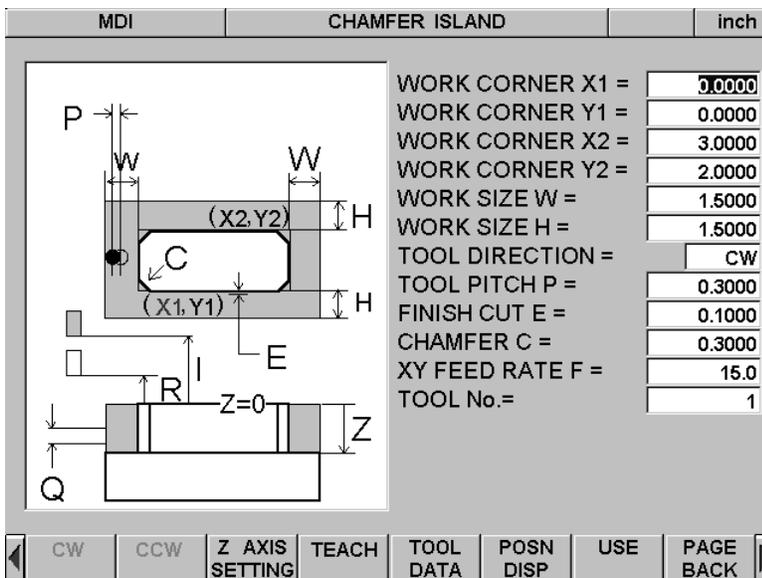Q: Depth to be cut in one step
FZ: Z-axis feed rate

9-5.   FRAME MILL
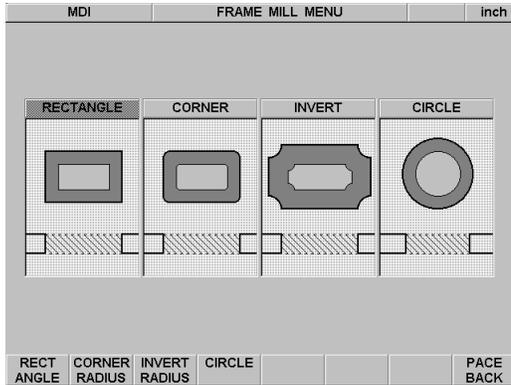
This function can open holes with the desired shapes into the workpiece. Since this function cuts only the perimeter, it can open holes in a short time. Also, this function can only be used for finishing Pocket Milling work. First, use the F1 to F4 keys to select the desired shape.
Press the PAGE BACK (F8 key) to return to the previous screen.

Typical Example

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

X1, Y1, X2, Y2: Coordinates for the opposite angles of the workpiece
TOOL DIRECTION: This is the tool movement direction during cutting. You can select from clockwise (CW) and counterclockwise (CCW).
E: Depth of cut for the finishing work
r: Inverted radius for the workpiece angle
F: Feed rate for the X and Y axes during cutting
Z: Workpiece plate thickness, or depth to be cut
I: Tool initial point (if not entered, this is taken as the current position)
R: Cutting starting point (if not entered, this is taken as the same position as I)
Q: Depth to be cut in one step

56

9-6.   RING MILL
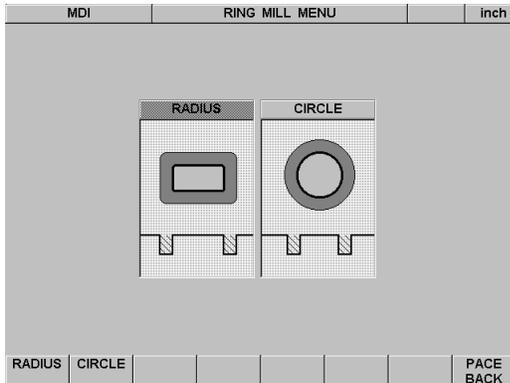
This function can cut grooves with the desired shapes into the workpiece. Use this function for machining grooves for O rings and similar work. The desired shape, groove width, and depth can be selected.

First, use the F1 or F2 key to select the desired shape. Press the PAGE BACK (F8 key) to return to the previous screen.

Typical Example

Key entry method:

The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.

Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.

Press the PAGE BACK (F8 key) to return to the previous screen.

X1, Y1, X2, Y2: Coordinates for the opposite angles of the workpiece

W: Width of groove

TOOL DIRECTION: This is the tool movement direction during cutting. You can select from clockwise (CW) and counterclockwise (CCW).

P: The STEP setting for the tool is variable.
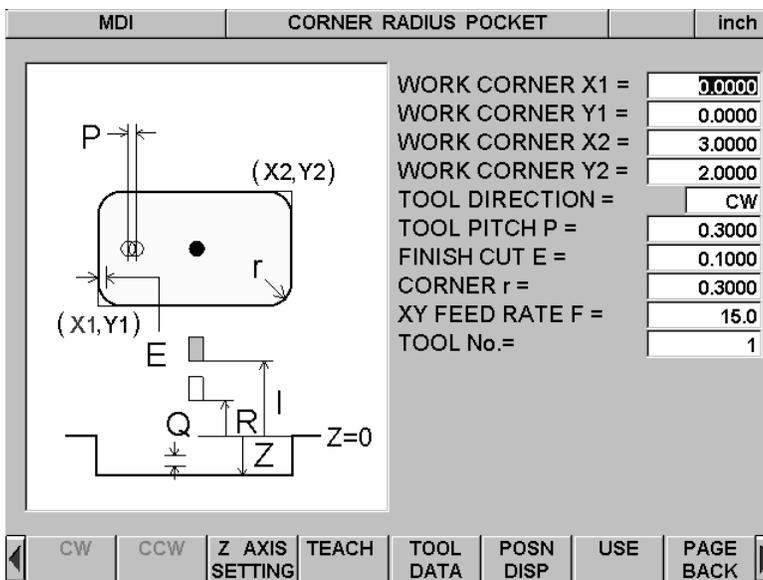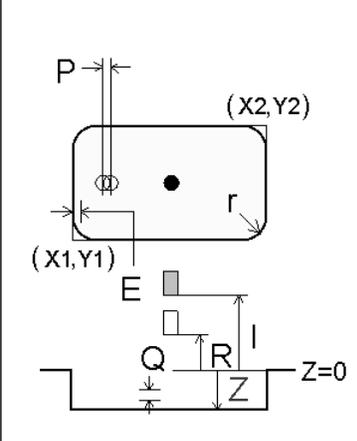
E: Depth of cut for the finishing work

r: Workpiece angle radius

F: Feed rate for the X and Y axes during cutting

Z: Depth to be cut

I: Tool initial point (if not entered, this is taken as the current position)

R: Cutting starting point (if not entered, this is taken as the same position as I)

Q: Depth to be cut in one step

• Z AXIS SETTING

MDI	CORNER RADIUS RING		inch

WORK CORNER X1 = 0.0000
WORK CORNER Y1 = 0.0000
WORK CORNER X2 = 3.0000
WORK CORNER Y2 = 2.0000
WIDTH W = 0.6000
TOOL DIRECTION = CW
TOOL PITCH P = 0.3000
FINISH CUT E = 0.1000
CORNER r = 0.3000

Z AXIS SETTING
DEPTH Z = -1.0000
INITIAL POINT I = 0.2000
SAFETY POINT R = 0.1000
PITCH/CYCLE Q = 0.2500
Z FEED RATE FZ = 15.0

		Z AXIS SETTING		TOOL DATA	POSN DISP	USE	PAGE BACK	

The Z-axis can be controlled by using the Z AXIS SETTING. This is only enabled for the 3-axis control models. For 2-axis control models, instruction messages for Z-axis operation are displayed.

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

Z: Depth to be cut
I: Tool initial point (if not entered, this is taken as the current position)
R: Cutting starting point (if not entered, this is taken as the same position as I)
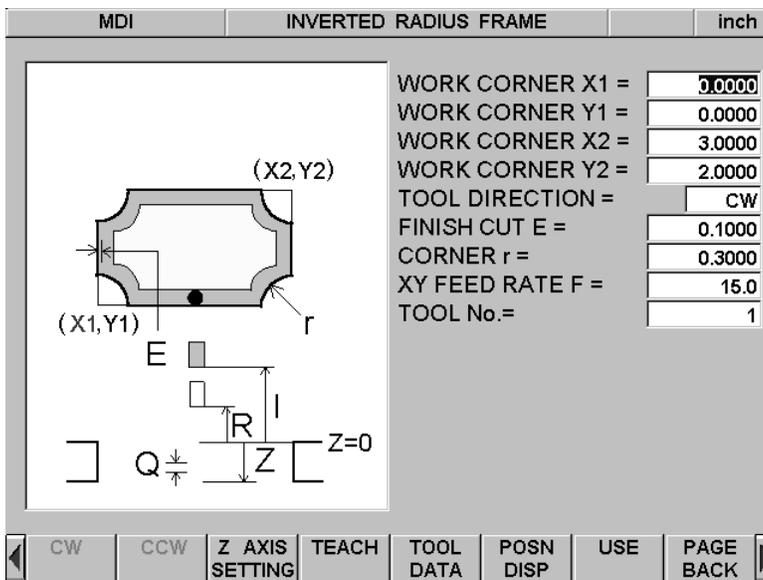Q: Depth to be cut in one step
FZ: Z-axis feed rate

• Addition Function

When selecting a machining shape (such as RECTANGLE in ISLAND MILL, for instance), pressing the [◄] and [►] function keys allows you to switch to another page. Pressing the CORNER or CENTER function keys on this next page lets you select a different entry method. Using these function keys allows machining with a designated angle with respect to the X+ direction.

Typical Example

ISLAND MILL - RECTANGLE - CORNER

Key entry method:
The input position is moved by using the [▲] or [▼] cursor keys. To change numerical settings, use the [►] or [◄] keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

X, Y: Corner point
U, V: Enter the machining shape size.
W, H: The cutting range can be selected. In this case, the machining shape is always left at the center of the cut-out piece.
TOOL DIRECTION: This is the tool movement direction during cutting. You can select from clockwise (CW) and counterclockwise (CCW).
P: The STEP setting for the tool is variable.
E: Depth of cut for the finishing work
A: The angle can be designated in degrees. The machine shape is straight at A=0, degree.
F: Feed rate for the X and Y axes during cutting
Z: Depth to be cut
I: Tool initial point (if not entered, this is taken as the current position)
R: Cutting starting point (if not entered, this is taken as the same position as I)
Q: Depth to be cut in one step

ISLAND MILL - RECTANGLE - CENTER
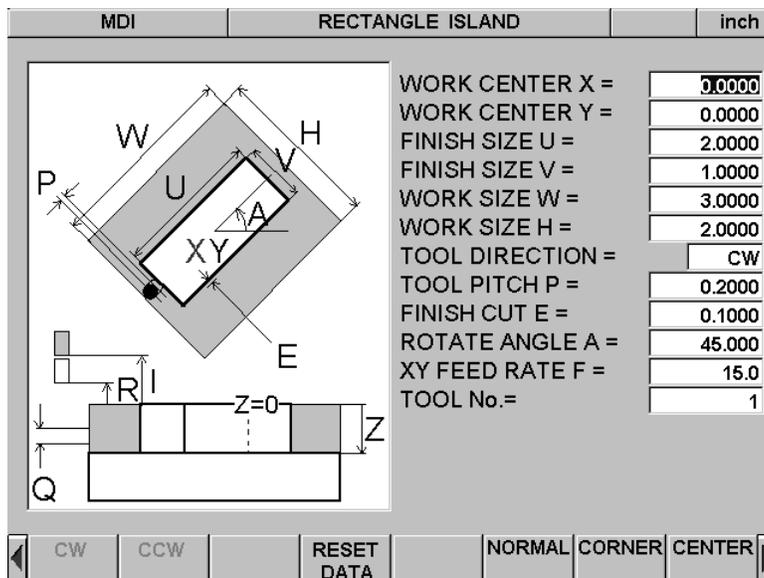
Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the TEACH key (F4) enters the current tool position. Pressing the USE key inputs the data to the program and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

X, Y: Center point

U, V: Enter the machining shape size.

H, W: The cutting range can be selected. In this case, the machining shape is always left at the center of the cut-out piece.

TOOL DIRECTION: This is the tool movement direction during cutting. You can select from clockwise (CW) and counterclockwise (CCW).

P: The STEP setting for the tool is variable.

E: Depth of cut for the finishing work

A: The angle can be designated in degrees. The machine shape is straight at A=0, degree.

F: Feed rate for the X and Y axes during cutting

Z: Depth to be cut

I: Tool initial point (if not entered, this is taken as the current position)

R: Cutting starting point (if not entered, this is taken as the same position as I)
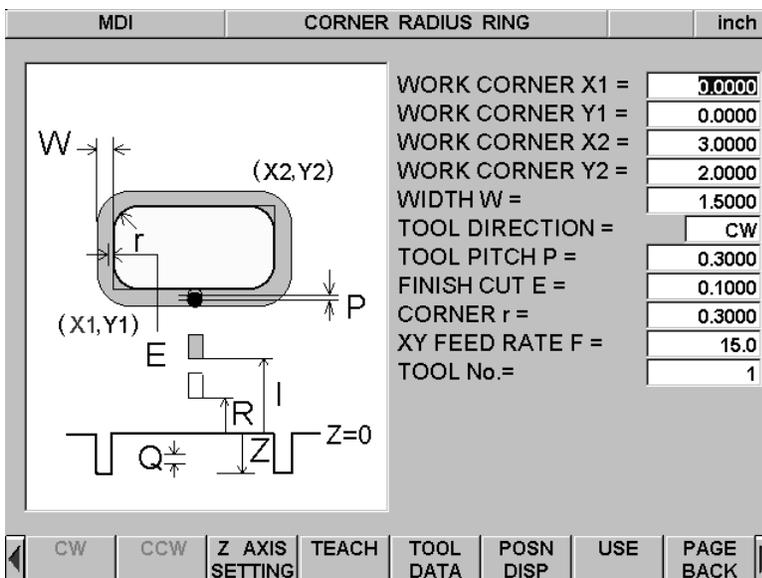
Q: Depth to be cut in one step

9-7.   LINE ARC

This function interpolates lines and arcs in MDI mode. RAPID TO (F1 key), LINE (F2 key), ARC (F3 key) are selectable. Press PAGE BACK (F8 key) to go back to the previous page.

RAPID TO

Key entry method:
The input position is moved by using the ▲ or ▼ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the USE (F7) key fixes data and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

Position the cursor on input items and corresponding menu is displayed above function keys (F1 – F6).
SPINDLE : Power ON/OFF of the spindle can be controlled by function keys SPINDLE ON (F4)/SPINDLE OFF (F5). Please note that this command becomes valid only when Spindle automatic control is selected in manual mode.
X, Y, Z : Enter the coordinates for of the axis to be moved.
COMPENSATION : Tool length compensation can be set to valid or invalid. LENGTH data in TO referred to here.
+ LENGTH (F4 key) interprets Z-axis travel plus tool length being travel value.
– LENGTH (F5 key) interprets Z-axis travel minus tool length being travel value.
L CANCEL (F6 key) invalids tool length compensation.
TOOL No. : This function displays the number of the currently selected tool. Designates the number of the tools to be changed.

LINE

MDI	LINE ARC MENU		inch

LINE

WORK POSITION

X:　　　0.0000 inch

Y:　　　0.0000 inch

Z:　　　0.0000 inch

TOOL No.　1
RADIUS　0.5000 inch
LENGTH　0.5000 inch
TOOL NAME　DRILLBF

TO

X		ABS
Y		ABS
Z		ABS
COMPENSATION		
TOOL No.	1	
FEED RATE		inch/min

RAPID TO	LINE	ARC			PLANE X-Y	USE	PAGE BACK

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the USE (F7) key fixes data and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

Position the cursor on input items and corresponding menu is displayed above function keys (F1 – F6).

SPINDLE : Power ON/OFF of the spindle can be controlled by function keys SPINDLE ON (F4)/SPINDLE OFF (F5). Please note that this command becomes valid only when Spindle automatic control is selected in manual mode.

X, Y, Z : Enter the coordinates for of the axis to be moved. By pressing PLANE (F6 key), planes can be selected in the order of X-Y, Z-Y and Y-Z.

COMPENSATION : Tool length compensation can be set valid or invalid. DIA/RAD and LENGTH values in TOOL DATA will be referred to.

LEFT (F1 key) offsets the tool by its diameter on the left side of the cutting line on the selected plane.

RIGHT (F2 key) offsets the tool by its diameter on the right side of the cutting line on the selected plane.

DIA/RAD CANCEL (F3 key) invalids tool diameter compensation.

+ LENGTH (F4 key) interprets Z-axis travel plus tool length being travel value.

– LENGTH (F5 key) interprets Z-axis travel minus tool length being travel value.

L CANCEL (F6 key) invalids tool length compensation.

TOOL No. : This function displays the number of the currently selected tool. Designates the number of the tools to be changed.

FEED RATE : Designate the feed rate.

ARC

Key entry method:
The input position is moved by using the ⬆ or ⬇ cursor keys. To change numerical settings, use the ➡ or ⬅ keys and the CAN, INS, and DEL keys.
Pressing the USE (F7) key fixes data and starts the Program Run screen for machining preparation.
Press the PAGE BACK (F8 key) to return to the previous screen.

Position the cursor on input items and corresponding menu is displayed above function keys (F1 – F6).
SPINDLE : Power ON/OFF of the spindle can be controlled by function keys SPINDLE ON (F4)/SPINDLE OFF (F5). Please note that this command becomes valid only when Spindle automatic control is selected in manual mode.
X, Y, Z : Enter the coordinates for of the axis to be moved. By pressing PLANE (F6 key), planes can be selected in the order of X-Y, Z-X and Y-Z, and the axis on the designated plane only will be displayed.
Press CENTER (F4 key) to designate the CENTER for radius input.
Press R (F5 key) to designate the R for radius input. One of CENTER or R is selectable.
CW/CCW : Select the cutting direction of the circle from CW (F4 key) or CCW (F5 key).
HELIX (F6 key) is for the helical cutting. Pressing this key, orthogonal axes on the currently selected plane will be displayed.
TOOL No. : This function displays the number of the currently selected tool. Designates the number of the tools to be changed.
FEED RATE : Designate the feed rate.

10. Program Mode

In Program Mode, you can create new programs, edit existing ones, and save and load programs from FD and the HDD.

1. Press the |MODE| key. The system enters the Mode Selection screen.

2. Press the | F2 | (PROG) key.

3. The system enters the Program MODE screen. The following function keys can be used.

<	CREATE	EDIT	RUN			DRIVE CHANGE		PAGE BACK	>
<	COPY FROM	CREATE FOLDER	DELETE	RENAME		DRIVE CHANGE		PAGE BACK	>
<	PROG SEND	PROG RECEIV			DNC				>

- At this screen, you can select CREATE, EDIT, RUN programs, and perform other operations for handling program files.

Key Name	Function
CREATE:	This function is to create new programs. Use this function to assign a name to the program. The system switches to the Program Input screen after the name is assigned.
EDIT:	This function is used to edit existing programs.
RUN:	This function is used to run existing programs.
DRIVE CHANGE:	This function is used to select the HDD or FD as the medium for saving or loading the program.
COPY FROM:	This function is used to copy programs.
CREATE FOLDER:	This function is used to create new folders.
DELETE:	This function is used to delete existing programs.
RENAME:	This function is used to change the name of program.
PROG SEND:	Send program data from RS-232C port.
PROG RECEIV:	Receive program from RS-232C port.
DNC:	Executes G code program received through RS-232C port directly.

List of PROGRAM Function

```
PROG ─┬─ CREATE ─┬─ RAPID TO
      │          ├─ LINE
      │          ├─ ARC
      │          ├─ MACH. CYCLE ──── (SAME AS MDI TREE)
      │          │
      │          │
      │          ├─ MISC. SET UP ──┬─ PROGRAM
      │          │                 ├─ PLANE SELECT
      │          │                 ├─ DWELL
      │          ├─ COPY/MOVE       ├─ TOOL OFFSET
      │          ├─ CORNER BLEND    ├─ TOOL CHANGE
      │          ├─ DO NOT SAVE     ├─ ROTATE
      │          │                  ├─ SPINDLE
      │          │                  ├─ POSITION SET
      │          │                  ├─ SCALING
      │          ├─ REDRAW          ├─ REPEAT
      │          ├─ CLEAR DISP      ├─ IRREGULAR
      │          ├─ VIEW INIT       ├─ MIRROR
      │          └─ ROTATE VIEW     └─ inch/mm
      │
      ├─ EDIT ──── (SAME AS CREATE TREE)
      │
      ○
```

```
         ○
         │
         │
  ┌──────────┐        ┌──────────────┐        ┌──────────────┐
  │   RUN    │────┬───│   DRY RUN    │   ┌────│   DRY RUN    │
  └──────────┘    │   └──────────────┘   │    └──────────────┘
                  ├───│ MACHINE LOCK │   ├────│ MACHINE LOCK │
                  ├───│    Z LOCK    │   ├────│    Z LOCK    │
                  ├───│    SINGLE    │   ├────│    SINGLE    │
                  ├───│ OPTNAL STOP  │   ├────│ OPTNAL STOP  │
                  ├───│    START     │   ├────│    START     │
                  ├───│    PAUSE     │   ├────│    PAUSE     │
                  ├───│  POSN DISP   │◄─►├────│GRAPHIC DISPLAY│
                  ├───│   FEED ↑     │   ├────│    FEED ↑    │
                  ├───│   FEED SET   │   ├────│   FEED SET   │
                  ├───│   FEED ↓     │   ├────│    FEED ↓    │
                  ├───│  OVERRIDE ↑  │   ├────│  OVERRIDE ↑  │
                  ├───│    100%      │   ├────│    100%      │
                  ├───│  OVERRIDE ↓  │   ├────│  OVERRIDE ↓  │
                  ├───│ GRPH OPTION  │─┐ ├────│ SPINDLE FWD  │
                  ├───│ SPINDLE FWD  │ │ ├────│ SPINDLE REV  │
  ┌──────────────┐├───│ SPINDLE REV  │ │ └────│ SPINDLE STOP │
  │  COPY FROM   ││   │ SPINDLE STOP │ │      └──────────────┘
  └──────────────┘│   └──────────────┘ │
  ┌──────────────┐│                    │      ┌──────────────┐
  │ CREATE FOLDER│┤                    └──┬───│ SIZE CHANGE  │
  └──────────────┘│                       ├───│   REDRAW     │
  ┌──────────────┐│                       ├───│ CLEAR SCREEN │
  │   DELETE     │┤                       └───│ EXIT OPTION  │
  └──────────────┘│                           └──────────────┘
  ┌──────────────┐│
  │   RENAME     │┤
  └──────────────┘│
  ┌──────────────┐│
  │ DRIVE CHANGE │┤
  └──────────────┘│
  ┌──────────────┐│
  │  PROG SEND   │┤
  └──────────────┘│
  ┌──────────────┐│
  │ PROG RECEIV  │┤
  └──────────────┘│
  ┌──────────────┐│
  │    DNC       │┘
  └──────────────┘
```

Flowchart for Program Creation

Steps	Keys used

Steps	Keys used
Enter Program mode	(PROG)
Assign a file name	(CREATE)
Designate the machining start point	(RAPID TO)
Designate the tool	(MISC. SETUP) (TOOL OFFSET)
Create the program	
Designate the machining shape	(LINE) (ARC) (ROTATE) (MACH. CYCLE) (CORNER BLEND)
Designate the machining conditions	(RAPID TO) (DWELL) (TOOL OFFSET) (TOOL CHANGE) (inch /mm)
Edit the program	(SUB CALL) (PROGRAM) (COPY/MOVE)
Save the program	(PAGE BACK)

67

10-1. Creating and Editing Programs

1) Key Description

The keys for Creating Programs are described below.

1. Press the ⌷MODE⌷ key.

2. Press the ⌷ F2 ⌷ (PROG) key.

(EDIT)

(CREATE)

3. Select the folder by using ⌷▲⌷ and ⌷▼⌷ keys.

4. Press the ⌷ F1 ⌷ (CREATE) key.

5. The software keyboard appears on the screen. To assign the file name, use the cursor keys to select the desired letters. You can use the numeric keys to enter numbers.

6. Press the ⌷ ENTER ⌷ key to set the file name. To modify the entered file name, press the ⌷DEL⌷ key and enter the correct file name.

7. After the file name is entered, press the ⌷ F1 ⌷ (CREATE) key again to enter the Program Creation screen. Use the keys on the next page when creating programs.

3. Press the ⌷ F2 ⌷ (EDIT) key after selecting the program file by using ⌷▶⌷, ⌷▲⌷ and ⌷▼⌷ keys when editing programs.

Note

If you enter a file name which already exists, it is overwritten when you save the program. To create a new program, be sure to enter a file name which is not being used.

<	RAPID TO	LINE	ARC	MACH. CYCLE	MISC. SETUP	COPY/ MOVE	CORNER BLEND	PAGE BACK	>
<	DO NOT SAVE	RE-DRAW	CLEAR DISP	VIEW INIT	ROTATE VIEW			PAGE BACK	>

Key Name	Function
RAPID TO:	This command performs a rapid traverse movement from the current position to the designated position.
LINE:	This command performs straight line movement. You can select either to enter the X, Y, Z coordinates for the intended point or to designate the line length and angle. Designating by the line length and angle can be performed only in the XY plane. If you enter a value for FEED RATE, the machine then moves at the designated rate. Once the value is entered for FEED RATE, the machine continues to move at this rate for linear and circular movement until the next event is designated. If no value for FEED RATE is designated in the program, linear or circular movement is performed at the JOG FEED rate designated in Manual mode.
ARC:	This command performs circular movement. You can select either the radius and X and Y coordinates of the arc end point or the center point and the X and Y coordinates of the arc end point. The arc direction can be selected as clockwise (CW) or counterclockwise (CCW).
MACH. CYCLE:	Pressing this key allows you to use all of the functions in MDI mode. Select the machining event. See the section entitled "MDI Mode" for the MDI setting procedure.
MISC. SETUP:	With this function, you can set various functions needed for execution of programs. This includes subprogram loading, DWELL settings, tool offset settings, tool selection, and rotation functions, among others.
COPY/MOVE:	This function is used to edit the program. This allows you to move and copy within the program.
CORNER BLEND:	With this function, you can crate an arc with radius R bordering two segments with a common point of intersection. The FRONT BLEND and BLACK BLEND functions allow you to select the line segment of the line before or after the line with the currently selected segment. You can designate the radius R to be assigned for BLEND RADIUS.
DO NOT SAVE :	Press this key before you leave the Creating and Editing mode by PAGE BACK key if the program does not need to be saved.
REDRAW :	This function is used to draw the machining shapes in programs again.
CLEAR DISP :	This function is used to delete the drawings of machining shapes in programs.

VIEW INIT :

> This function is used to change the angle of the machining shapes drawings in programs to the initial setting.

ROTATE VIEW :

> This function is used to rotate the machining shape drawings automatically.

2) Setting the Basic Functions for Creating and Editing Programs

1. Entering the rapid traverse commands

On the PROGRAM EDITOR screen

Press the ⌑ F1 ⌑ (RAPID TO) key and the system enters the Input mode for positioning command.

						USE	PAGE BACK

"FROM window" on the top right of the screen displays the final position which was performed by the previous movement command. ("FROM window" is blank at the beginning of the program.)

1.1 Input method of the target position (or distance)

Input the target position to the "TO window" at the bottom right of the screen, by using numerical keys, cursor keys and X/Y/Z keys.

Display ABS on the right side of the numeric value with ABS key to input the finishing point. Display INC on the right side of the numeric value with INC key to input the distance to the finishing point.

2.2 How to define the input values and complete

Press the ⌑ F7 ⌑ (USE) key after entering values. Designated values are converted to blocks and displayed on the right side of the screen.

Press the ⌑ F8 ⌑ (PAGE BACK) key to discontinue input.

2. Entering the line movement commands

On the PROGRAM EDITOR screen

Press the [F2] (LINE) key and the system enters the linear interpolation command mode.

TARGET POSITION	?BLOCK	ANGLE LENGTH	ANGLE XYZ	LENGTH XYZ		USE	PAGE BACK

"FROM window" on the top right of the screen displays the final position which was performed by the previous movement command. ("FROM window" is blank at the beginning of the program.)

2.1   Input method of the target position (or distance)

Press the [F1] (TARGET POSITION) key. Input the target position to the "TO window" at the bottom right of the screen, by using numerical keys, cursor keys and X/Y/Z keys.
Display ABS on the right side of the numeric value with ABS key to input the finishing point. Display INC on the right side of the numeric value with INC key to input the distance to the finishing point.

Press the [F7] (USE) key after entering values. Designated values are converted to blocks and displayed on the right side of the screen.

Press the [F8] (PAGE BACK) key to discontinue input.

2.2   Input method of the feed rate
Certain feed rate (cutting speed) can be set in the item of "FEED RATE" in the "TO window" at the bottom right. Once the value is set, it will be effective in the following blocks. If the input is omitted or set to 0, feed rate will be the value of JOGFEED (x OVERRIDE) which was set during operation.

2.3   How to define the input values and complete

Press the [F7] (USE) key after entering values. Designated values are converted to blocks and displayed on the right side of the screen.

Press the [F8] (PAGE BACK) key to discontinue input.
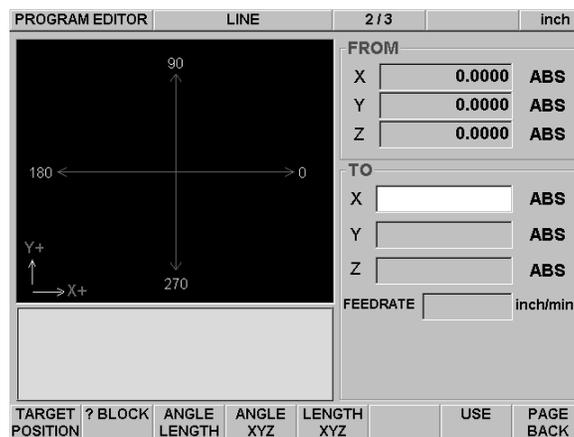
2.4   Input method of ANGLE LENGTH

Press the [F3] (ANGLE LENGTH) key. Enter FEED RATE, ANGLE and LENGTH into the "TO window" at the bottom right of the screen. Set ANGLE and LENGTH in the coordinates which was designated in MISC. SETUP (see the item 3) – 2.).
[Example]
X-Y PLANE, ANGLE = 30 DEGREES, LENGTH = 10 inches
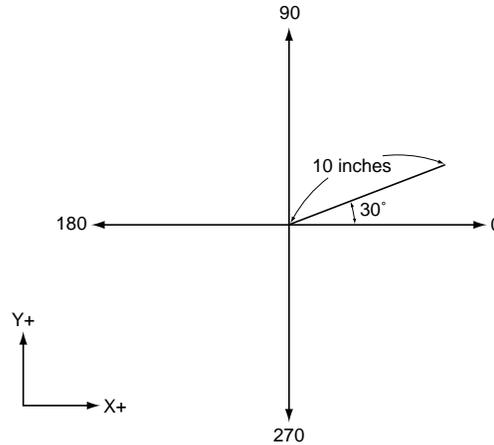
2.5   Input method of ANGLE XYZ

Press the [F4] (ANGLE XYZ). Enter FEED RATE, ANGLE and LENGTH in coordinates into the "TO window" at the bottom right of the screen. Select LENGTH X, LENGTH Y and LENGTH Z with X/Y/Z keys. Set ANGLE and LENGTH in the coordinates which was designated in MISC. SETUP (see the item 3) – 2.).
[Example]
X-Y PLANE, ANGLE = 30 DEGREES, LENGTH = 10 inches

72
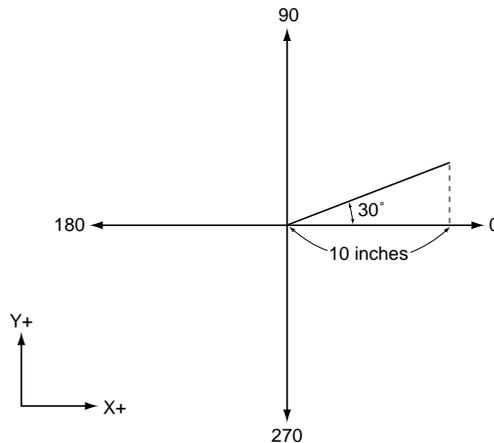
2.6   Input method of LENGTH XYZ

Press the $\boxed{\text{F5}}$ (LENGTH XYZ).

Enter FEED RATE, LENGTH and LENGTH in coordinates into the "To window" at bottom right of the screen.
Select LENGTH X, LENGTH Y and LENGTH Z with X/Y/Z keys. Set LENGTH in the coordinates which was designated in MISC. SETUP (see the item 3) – 2.).
[Example]
X-Y PLANE, LENGTH = 10 inches, LENGTH X = 5 inches
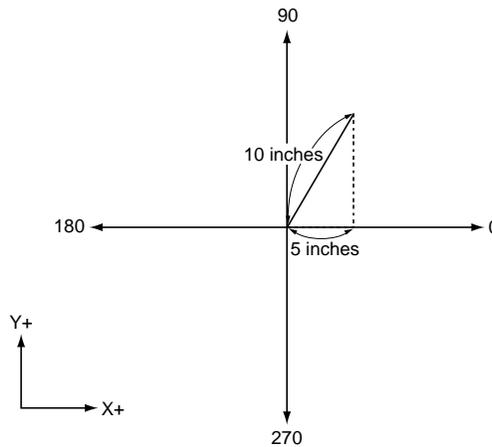
2.7   Input method of ?BLOCK key
In machine programming there is a case where you can define the shape by internal calculation even when some input data lacks. In a block you are programming when input data of LINE lacks you can move on to the editing of next block without determining the shape by ?BLOCK function.

After pressing $\boxed{\text{F2}}$ (?BLOCK) key follow the message to input necessary values and ? mark in the unknown item.
Error will be indicated when it is unable to automatically calculate with this function.
?BLOCK can be used up to 3 blocks continuously by the combination of LINE and ARC commands.

3. Entering the circular (ARC) movement commands

On the PROGRAM EDITOR screen

Press the ☐ F3 (ARC) key and the system enters the circular interpolation command mode.

CENTER TARGET	RADIUS TARGET	?BLOCK	HELIX	X-Y PLANE		USE	PAGE BACK

"FROM window" on the top right of the screen displays the final position which was performed by the previous movement designation. ("FROM window" is blank at the beginning of the program.)

3.1   Selection method of the circular plane

Planes on which the circular interpolation is performed are switched by pressing the ☐ F5 (X-Y PLANE) key. Selected plane is indicated by input item, XY/ZX/ YZ, in the "TO window" at the bottom right of the screen.

3.2   Input method of the center point of circles and finishing point

Input method of the center point of circles is selectable from center point designation and radius resignation.

3.2.1. Input by the center designation method

Press the ☐ F1 (CENTER TARGET) key. Input the target position to the "TO window" at the bottom right of the screen, by using numerical keys, cursor keys and X/Y/Z keys.

Display ABS on the right side of the numeric value with ABS key to input the finishing point. Display INC on the right side of the numeric value with INC key to input the distance to the finishing point.

Press the ☐ F7 (USE) key after entering values. Designated values are converted to blocks and displayed on the right side of the screen.

Press the ☐ F8 (PAGE BACK) key to discontinue input.

TO (X, Y), (X, Z) or (Y, Z)

FROM (X, Y, Z)

TO
CENTER (X, Y), (X, Z) or (Y, Z)
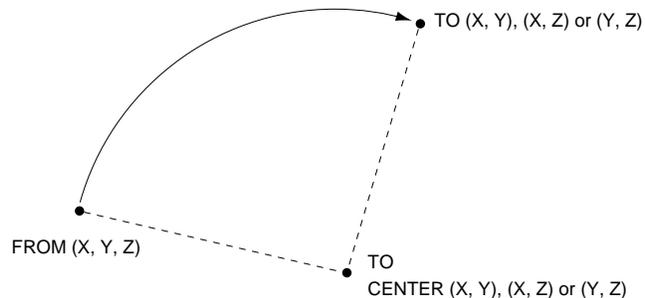
3.2.2. Input by the radius designation method

Press the $\boxed{\text{F2}}$ (RADIUS TARGET) key. Input the target position to the "TO window" at the bottom right of the screen, by using numerical keys, cursor keys and X/Y/Z keys.
Display ABS on the right side of the numeric value with ABS key to input the finishing point. Display INC on the right side of the numeric value with INC key to input the distance to the finishing point.
Set the radius R with positive values when the arc is smaller than 180°, and with negative values when the arc is larger than 180°. Radius designation method is not available for setting a circle as its center is indefinite. Use the center designation method.

Press the $\boxed{\text{F7}}$ (USE) key after entering values. Designated values are converted to blocks and displayed on the right side of the screen.

Press the $\boxed{\text{F8}}$ (PAGE BACK) key to discontinue input.

FROM (X, Y, Z)

RADIUS
(+R)

TO (X, Y)

RADIUS
(–R)

FROM (X, Y, Z)

TO (X, Y)
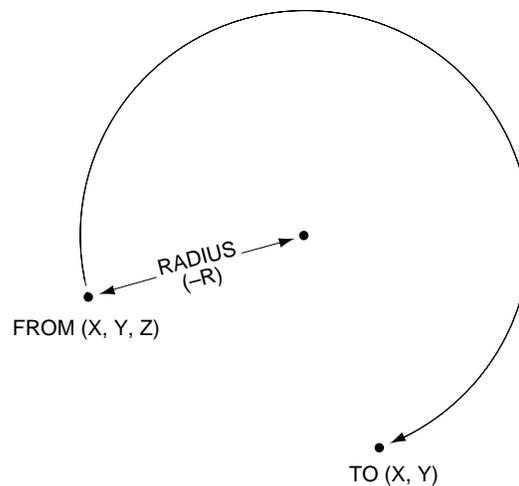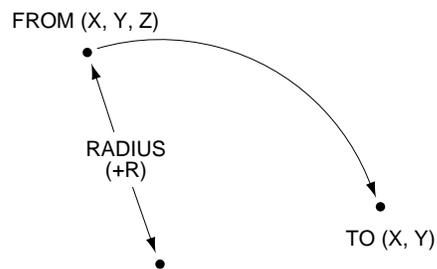
3.3   Input method of the feed rate
Certain feed rate (cutting speed) can be set in the item of "FEED RATE" in the "TO window" at the bottom right. Once the value is set, it will be effective in the following blocks. If the input is omitted or set to 0, feed rate will be the value of JOGFEED (x OVERRIDE) which was set during operation.

3.4 Input the path
Designate the path to the designated target value is whether CW or CCW.
Move the cursor to "CW/CCW" input space in the "TO window" at the bottom

right, F1 changes into CW and F2 into CCW.
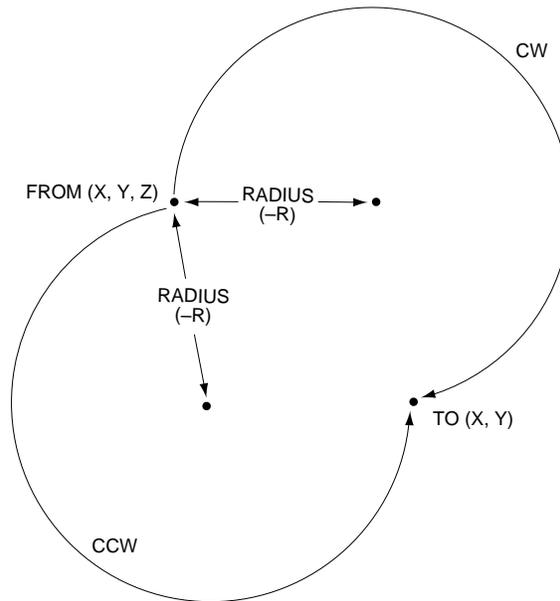
3.5 Input method of the helical interpolation
This function is used to perform the helical interpolation by adding linear

interpolation command to the normal circular interpolation. Press the F4

(HELIX) key. Another axis than ones on the selected plane, and the helical
interpolation input becomes available. Press ABS key to enter the finishing point
of linear movement, and INC key to enter the distance to the finishing point.

Press the F4 (HELIX) key again, and the system goes back to the usual

circular interpolation designation.

3.6 Input method of ?BLOCK key
In machine programming there is a case where you can define the shape by
internal calculation even when some input data lacks. In a block you are
programming when input data of ARC lacks you can move on to the editing of
next block without determining the shape by ?BLOCK function.

After pressing F3 (?BLOCK) key follow the message to input necessary

values and ? mark in the unknown item.

Error will be indicated when it is unable to automatically calculate with this
function.

?BLOCK can be used up to 3 blocks continuously by the combination of LINE
and ARC commands.

4. Entering the machine cycle (MACH. CYCLE) commands

On the PROGRAM EDITOR screen

Press the ☐ F4 ☐ (MACH. CYCLE) key and the system enters the machine cycle command mode.

DRILL	FACE MILL	ISLAND MILL	POCKET MILL	FRAME MILL	RING MILL	LINE ARC	PAGE BACK

All functions in MDI mode are available.
See the "MDI mode" for setting method.

3) Miscellaneous Function for Creating and Editing Programs

On the PROGRAM EDITOR screen

Press the ☐ F5 ☐ (MISC. SETUP) key to enter the Miscellaneous Function input mode.

After selecting functions with ☐ ∧ ☐ ☐ ∨ ☐ keys and commands with ☐ F1 ☐ - ☐ F6 ☐ keys,

press the ☐ F7 ☐ (USE) key to input commands to programs.
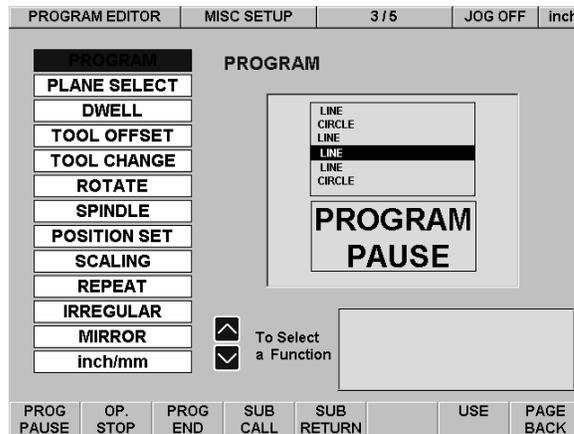
1. PROGRAM function

PROG PAUSE	OP. STOP	PROG END	SUB CALL	SUB RETURN		USE	PAGE BACK

This function has the following commands.

Key Name	Function

PROG PAUSE:

To insert this command into the program line, pauses execution of the program. The program is then executed again when the START key is pressed.

Key Name	Function

OP. STOP:

Selecting the OP. STOP on the PROGRAM RUN screen, execution of the program is stopped on the OP. STOP command line during the program execution. In this condition, press the START key to restart execution of the program.

PROG END:

Select this setting to end the program which is being created. When this command is given, the program contents following this line are not executed.

SUB CALL:

Selecting and inserting the sub routine files into the program, other programs can be executed as sub routines. Up to four nested subprograms can be loaded.

SUB RETURN:

Insert this command at the end of the SUB PROGRAM line. When SUB RETURN is executed, the program returns to the program which loaded the SUB PROGRAM, and execution is continued. If SUB RETURN is not inserted, program execution ends when the program loaded by the SUB PROGRAM is executed.

2. PLANE SELECT Function

X-Y PLANE	Z-X PLANE	Y-Z PLANE				USE	PAGE BACK

This function sets basic operation axes such as LINE and ARC by plane selection.

3. DWELL Function

This function lets you delay the time until the program moves to the next block operation. Use the numeric keys to enter the dwell time. The unit is second.

4. TOOL OFFSET Function

LEFT	RIGHT	RADIUS CENTER	LENGTH +	LENGTH –	LENGTH CANCEL	USE	PAGE BACK

This function commands compensation of the tool diameter and tool length.
When LEFT and RIGHT commands are inserted into the program, selected tool diameter in the next RAPID or LINE operation is compensated and positioning is made.
When LENGTH + and LENGTH – commands are inserted into the program, selected tool length in the next Z-axis operation is compensated and positioning is made.

Key Name	Function
LEFT:	
	This command offsets the tool by a radius length to the left of the tool advancement direction.
RIGHT:	
	This command offsets the tool by a radius length to the right of the tool advancement direction.
RADIUS CENTER:	
	This command does not offset the tool. The tool cuts a radius length inside of the designated value.
LENGTH+:	
	This command shifts the Z-axis to "+" direction when the length of the selected tool is longer than the standard tool length.
LENGTH–:	
	This command shifts the Z-axis to "–" direction when the length of the selected tool is shorter than the standard tool length.
LENGTH CANCEL:	
	This command cancels the tool length compensation.

5. TOOL CHANGE Function

This function is used to select the tool.

TOOL SELECT:	
	Pressing this key moves to TOOL DATA mode where you can select the desired tool. You can select the tool by entering the tool diameter or the tool radius, and so be careful not to mistake the setting value.

6. ROTATE Function

This function rotates the program from ROTATE SET to ROTATE END at the designated angle.

ROTATE START:	
	Inserting this command allows you to rotate the machining commands which follow this line. You can designate the coordinates of the center of rotation and the angle.
ROTATE END:	
	This command is inserted to end the angle designation. The ROTATE function is enabled up to this block.

7. SPINDLE Function

Forward rotation, reverse rotation or stop of spindle is selected when spindle control unit (NZ31) is attached.

Besides, when SPEED SET command is selected and SPINDLE speed is input, message is indicated to set Spindle speed during program execution.

Key Name	Function
FWD:	
	Spindle motor rotates in forward direction.
REV:	
	Spindle motor rotates in reverse direction.
STOP:	
	Spindle motor stops.
SPEED SET:	
	Input Spindle speed.

8. POSITION SET Function

Work position is set to optional value at the time of command execution. Following two kinds of setting are available.

Key Name	Function
POSITION SET:	
	Changes current work position to the setting value of X, Y and Z.
OFFSET SELECT:	
	Selects a coordinate from six work coordinates which have been set on OFFSET CHANGE, then applies it.
OFFSET CHANGE:	
	Enables to set six work coordinates based on machine coordinate zero point, regardless of the current position. The coordinates which were set with this command will be saved in system parameter when they are once executed.

(Ex.)...

    0010 DIAM INPUT LZX-10.000 Y-20.000 Z-7.000P2   Made on OFFSET CHANGE
    0011 WORK POSITION 2   Made on OFFSET SELECT
    ...

Set X=-10.000, Y=-20.000, Z=-7.000, P2 on OFFSET CHANGE.

Work coordinate of machine coordinate zero point based on P2 (-10, -20, -7).
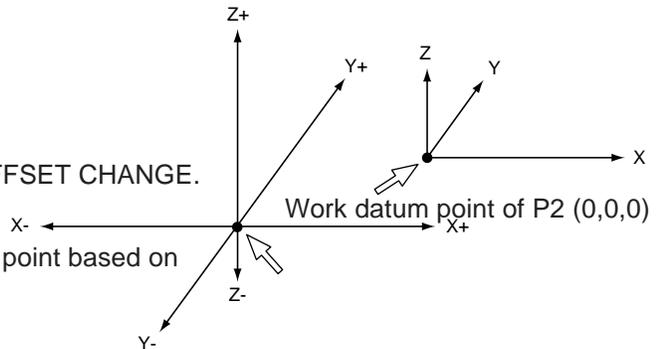
9. SCALING Function

This function zooms in/out the processing data after SCALING START command inserted, with the reference point designated by X, Y and Z as the center.

Magnifications are set by X0.1 unit from X0.1 to 9.9999.

SCALING END command cancels the zoom in/out. The machine does not move, and a tool path is displayed on the screen. Check the machine operation by observing this tool path.

10. REPEAT Function

This function repeats the designated blocks in the programs by designated number.

Enter the block by no. of START line and END line of the program.

11. IRREGULAR Function

This function performs the POCKET milling in a certain shape of the block which is surrounded by IRREG. START command and IRREG. END command.

11.1   Program a closed contour using LINE and ARC commands.

11.2   Insert an IRREGULAR START block immediately before the first block in the closed contour.

11.2.1 Move the cursor onto the first block of the closed contour using UP or DOWN arrow keys.

11.2.2 Press INS key to insert a blank block before the first block of the closed contour.

11.2.3 Press MISC. SETUP to show the miscellaneous setup menu, then select the IRREGULAR item and press IRREG. START (F1) key to show the parameter input menu.

Enter the needed parameters for your pocket cutting here.

The following prompt messages for each parameter are also displayed on the screen.

(X,Y)	Start Position
Z	Pocket Bottom Position
I	Initial Point (Z-axis)
R	Start Point (Z-axis)
Q	Cut Pitch (Z-axis)
P	Cut Pitch (XY plane) (P < Tool Radius)
E	Finish Cut Pitch (XY plane)
ER	Finish Cut Approach Radius
FN	Finish Cut Approach Block No. (See Note 5.)
F	Feedrate
	If omitted, the JOGSPEED will be used.
FE	Feedrate for Finish Cut
	If omitted, the F will be used.
FZ	Feedrate for Z-axis
FTD	Finish Cut Direction
	0 – Pocket Definition
	1 – CW
	2 – CCW

11.2.4 Press USE(F7) key to go back to the top menu of program editor.

11.3 Enter an IRREGULAR END block immediately following the last block of the closed contour.
The detailed description is omitted here for it follows almost the same procedure as entering an IRREGULAR START block.

Note

1. The Start Position (X,Y) must be located inside the pocket contour.
2. The straight path connecting the Start Position (X,Y) and the contour's start position should not cross with any contour
blocks in order to avoid over cutting.
3. The feedrate in each LINE or ARC block of the pocket contour will be overridden by the designated F feedrate.
4. The Finish Cut will be conducted after the inside of the pocket has be cleared out. A Zero-valued or omitted Finish Cut Pitch E implies that no Finish Cut will be done.
5. Finish Cut Approach Block No. FN is counted from the first block of the pocket contour. An improper value will cause an alarm when the program is run.

12. MIRROR Function

This function is used to perform the MIRROR shape processing by designating the program block with MIRROR ON command and MIRROR OFF command.
Target position on the coordinates can be changed to the opposite location based on the designated line or point as the center.

[Example]
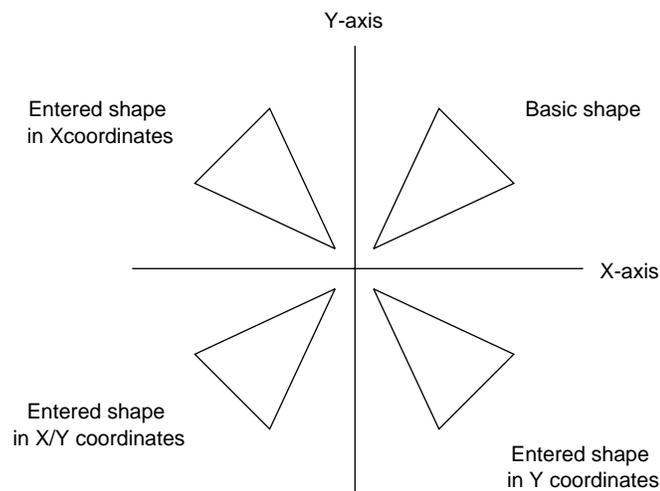
13. inch / mm Function

This function switches inch commands and mm commands during program operation.

4) COPY/MOVE Function for Creating and Editing Programs

On the PROGRAM EDITOR screen

Press the [F6] (COPY/MOVE) key to enter into the COPY/MOVE screen.

COPY BLOCK	DELETE BLOCK	MOVE BLOCK			INPUT LINE	USE	PAGE BACK

This function performs COPY/DELETE/MOVE in the already created parts in the programs.

1. COPY BLOCK

1.1 Press [F1] (COPY BLOCK) key.

1.2 Programs which are already created are displayed on the left of the screen, and cursor can be moved by [∧][∨] keys.

1.3 Cursor can be moved by [▲][▼] keys to START NO. and END NO. of the COPY FROM, and START NO. and COPY COUNT of the COPY TO.

1.4 Move the cursor by [▲][▼] keys to the desired position, and enter the no. of the line by [∧][∨] keys or [0] – [9] keys and press [F6] (INPUT LINE) key to determine it.

1.5 Press the [F7] (USE) key to execute the COPY.

2. DELETE BLOCK

2.1 Press [F2] (DELETE BLOCK) key.

2.2 Move the cursor by [▲][▼] keys to the desired position, and enter the no. of the line by [∧][∨] keys or [0] – [9] keys and press [F6] (INPUT LINE) key to fix it.

2.3 Press the [F7] (USE) key to execute the DELETE.

3. MOVE BLOCK

3.1   Press the ☐F3 (MOVE BLOCK) key.

3.2   Move the cursor by ▲▼ keys to the desired position, and enter the no. of the line by ∧∨ keys or ☐0 – ☐9 keys (this is not the number of N****) and press ☐F6 (INPUT LINE) key to fix it.

3.3   Press the ☐F7 (USE) key to execute the MOVE.

5)   CORNER BLEND Function for Creating and Editing Programs

1. First, designate the angle that you want to assign an arc.
2. The section with two intersecting lines at the screen left line can be used for blending arc. For example, these are displayed as "5 : LINE" and "6 : LINE".
3. Move the cursor to "5 : LINE".
4. In this state, press the ☐F7 (CORNER BLEND) key. The Input screen appears.
5. Next, the two line segments to be designated are displayed.
6. Enter the desired value in the BLEND RADIUS box. If the numeric value is correct, the result is instantly calculated and displayed on the screen. A message is displayed when an unacceptable value for blending is entered.
7. Press the ☐F6 (USE) key if the result is OK.

10-2. Handling of the Program Files

This section describes the handling of the program files.

1. Press the [MODE] key.

2. Press the [F2] (PROG) key to enter the PROGRAM MODE screen.

<	CREATE	EDIT	RUN			DRIVE CHANGE		PAGE BACK	>
<	COPY FROM	CREATE FOLDER	DELETE	RENAME		DRIVE CHANGE		PAGE BACK	>
<	PROG SEND	PROG RECEIV			DNC				>

1) Copy Function

This function is used to copy a stored program file and save it to a floppy disk or the hard disk. This function can also be used to assign a different program file name to the same program.
1. First, select the program file to be copied.
2. The file folder on the left side is highlighted with a yellow outline. To select a program file, use the [➡] cursor key on the numeric keyboard to move to the files on the right side.
   The selected program file is shown in blue. Use the [⬆][⬇] cursor keys on the numeric keyboard to select the desired program file.

Note
If you select the FDD, make sure that the floppy disk is already inserted in the drive. Do not close the lid on the front panel when a floppy disk is inserted.

3. Press the [⬅] or [➡] function key.

4. Press the [F1] (COPY FROM) key. The program file name for the FROM option appears.

5. Select either HDD or FDD, and then press the [F1] (COPY TO) key again.

Note
If the same file is used, a message appears asking whether it is OK to save the file. Press OK. The end of the file name is automatically underlined, and then it is saved.

2)   Delete

This function is used to delete program files.

1. At first, the file folder on the left side is highlighted with a yellow outline. To select the program file to delete, use the ⬛➡ cursor key on the numeric keyboard to move to the files on the right side. The selected program file is shown in blue. Use the ⬛⬆ ⬛⬇ cursor keys on the numeric keyboard to select the desired program file.

    Note
    If you select the FDD, make sure that the floppy disk is already inserted in the drive.

2. Press the ⬛⬅ or ⬛➡ function key.

3. Press the [F3] (DELETE) key. A message then appears asking you to confirm the deletion (OK or CANCEL). To delete the file, press the [ENTER] key.

3)   Renaming the File Name

This function allows you to change the file name for an existing program. Use this function to change file names for copied or edited files.

1. At first, the file folder on the left side is highlighted with a yellow outline. To select the program file to rename, use the ⬛➡ cursor key on the numeric keyboard to move to the files on the right side. The selected program file is shown in blue. Use the ⬛⬆ ⬛⬇ cursor keys on the numeric keyboard to select the desired program file.

    Note
    If you select the FDD, make sure the floppy disk is already inserted in the drive.

2. Press the ⬛⬅ or ⬛➡ function key.

3. Press the [F4] (RENAME) key. The software keyboard appears.

4. Use the ⬛⬆ ⬛⬇ ⬛⬅ ⬛➡ cursor keys to select the text to enter.

5. Press the [ENTER] key to register the text.

6. After using these steps to enter the new file name, press the [F4] RENAME key again to register the changed file name.

10-3. Running the Program

This section describes items pertaining to running the created program. To run a program, the program file must first be loaded.

1. Press the MODE key to enter the Mode Selection screen.

2. Press the F2 (PROG) key.

3. The system enters the Program MODE screen.

1) Loading the Program File

1. Select the program file to be run.
2. At first, the file folder on the left side is highlighted with a yellow outline. To select the program file, use the ➡ cursor key on the numeric keyboard to move to the files on the right side. The selected program file is shown in blue. Use the ⬆ ⬇ cursor keys on the numeric keyboard to select the desired program file.

3. Press the F3 (RUN) key. The program is loaded, and the system enters the Program Run state.

2) Checking the Program Operation

When machining for the first time, as an added precaution, perform a program operation check before the actual machining. A description of the display screen and checking functions is given below.
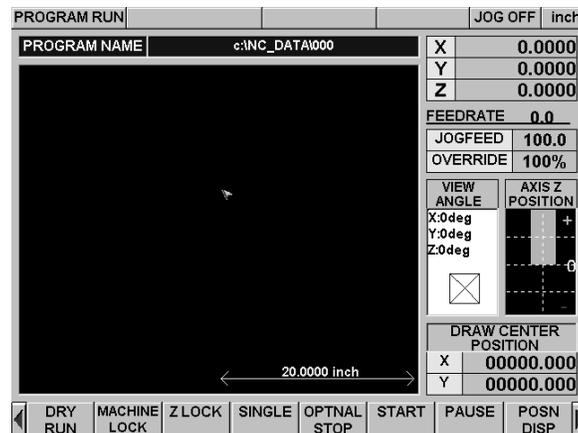
87

<	DRY RUN	MACHINE LOCK	Z LOCK	SINGLE	OPTNAL STOP	START	PAUSE	POSN DISP	>
<	SPINDLE FWD	SPINDLE REV	SPINDLE STOP						>
<	FEED ↑	FEED SET	FEED ↓	OVER RIDE ↑	100%	OVER RIDE ↓	GRAPH OPTION	PAGE BACK	>

Press EXIT OPTION ↑ ↓ Press GRAPH OPTION

<	SIZE CHANGE	REDRAW	CLEAR SCREEN				EXIT OPTION		>

Key Name	Function

DRY RUN:

This function allows you to change the feed rate to JOG FEED, ignoring FEED RATE written in the program.
This check allows the checking of long programs to be finished in a short time. When the DRY RUN (F1) key is pressed, the key on the screen changes to blue and DRY RUN is enabled.

`Note`

Be aware that when the START key is pressed in this state to execute the program, the machine will actually move at the speed set at Jog Feed. Pressing the DRY RUN (F1) key again disables this function, and the machine will then move at the speed for FEED RATE written in the program.

MACHINE LOCK:

This function allows you to run only the program without moving the machine. When the MACHINE LOCK (F2) key is pressed, the key on the screen changes to blue and MACHINE LOCK is enabled. When the START key is pressed to execute the program, the machine does not move, and a tool path is displayed on the screen. Check the machine operation by observing this tool path.

`Warning`

Pressing the MACHINE LOCK key again while the program is running or during MDI operation will disable MACHINE LOCK, and the machine will start to move.

`Caution`

In case MACHINE LOCK is disabled in the middle of the program the program will once stop.
Then, when the program is restarted the difference between programmed position on MACHINE LOCK and actual position will be added to the value of movement.

Z LOCK:

This function invalidates the commands to the Z-axis in the program operation, and Z-axis stays still.

`Caution`

In case Z LOCK is disabled in the middle of the program the program will once stop.
Then, when the program is restarted the difference between programmed position on Z LOCK and actual position will be added to the value of movement.
On 2 axis model (NU20-20)
At time of Z axis command 2 axis model will stop with a message displayed.
With Z LOCK on the machine will not stop at time of Z axis command.
This enables to execute program verification without stopping the function verification of program.

SINGLE:

The machine stops after executing one line of commands in a program. When the SINGLE (F4) key is pressed, the key on the screen changes to blue and SINGLE is enabled. When the START key is pressed to execute the program, the machine executes the events in the first line and then pauses. To go to the next operation, press the START key again. If the program includes MDI items, all of the MDI items are executed (such as CIRCLE for POCKET MILL), and then the machine is paused.

OPTNAL STOP:

When this function is ON, program operation stops on the OPTIONAL STOP command line in the program.

The program restarts by pressing the START key.

START:

Pressing this key runs the program. During execution, "RUN" is displayed at the top of the screen.

PAUSE:

Pressing this key while a program is running pauses the movement of the X and Y axes. When a program is paused, "PAUSE" is displayed at the top of the screen. To restart a program, press the START key.

Note

The START/PAUSE operation can be performed even if the optional START/PAUSE switch is being used.

POSN DISPLAY:

This switches the screen display. When this key is pressed, the screen changes from Graphic Display to Position Display. The program to be run is displayed at the right side. When the Graphic Display key is pressed, the screen returns to the original Graphic Display screen.

3) Other Functions

Press the cursor keys at both ends of the function keys. This allows you to make settings for the feed rate and display screen.

FEED SET, OVERRIDE:

When using DRY RUN, use these functions to change the feed rate. For more information, see "7-2. Jog Feed Function" in "7. Basic Operatoin".

GRAPH OPTION:

This functions is used for making display screen settings. Press this key first. The following functions are displayed.

SIZE CHANGE:

The function enlarges or reduces the screen size. The screen size can be changed while a program is running, but the display is redrawn starting from the changed section.

REDRAW:

Press this button when the screen has become hard to see due to SIZE CHANGE or other function. The tool path line where the program starts is redrawn.

CLEAR SCREEN:

When there are many tool paths drawn on the screen and the current tool path is hard to see, press this key to erase the tool path lines on the screen. If this key is pressed while a program is being executed, the lines are erased, and then the remaining path is drawn from the current position.

DRAW CENTER POSITION:

This is the coordinate position for the drawing screen center. The coordinate values are displayed at the bottom right of the screen. To change a coordinate value, press the

$\boxed{X}$ or $\boxed{Y}$ key after pressing the $\boxed{F7}$ (GRAPH OPTION) key.

SPINDLE FWD:

Spindle motor rotates in forward direction.

SPINDLE REV:

Spindle motor rotates in reverce direction.

SPINDLE STOP:

Spindle motor stops.

`Note`

SPINDLE FWD, SPINDLE REV, SPINDLE STOP commands are available with optional NZ31.

10-4.   RS-232C Communication

With a serial port on the controller employed you can run the machine in DNC mode such as LOAD, SAVE, etc. of program. Please contact Sony Precision Technology for further information.

1)   Connection

Personal computer

SERIAL PORT

STOP SW INPUT   To Electrical Box
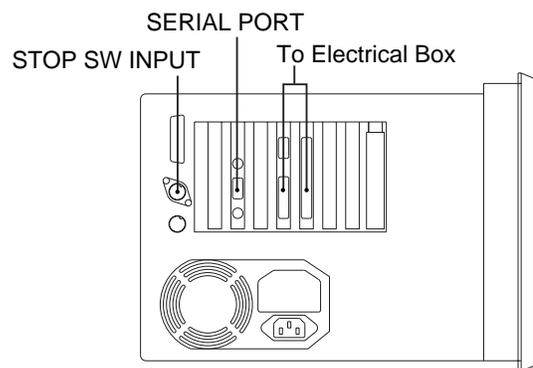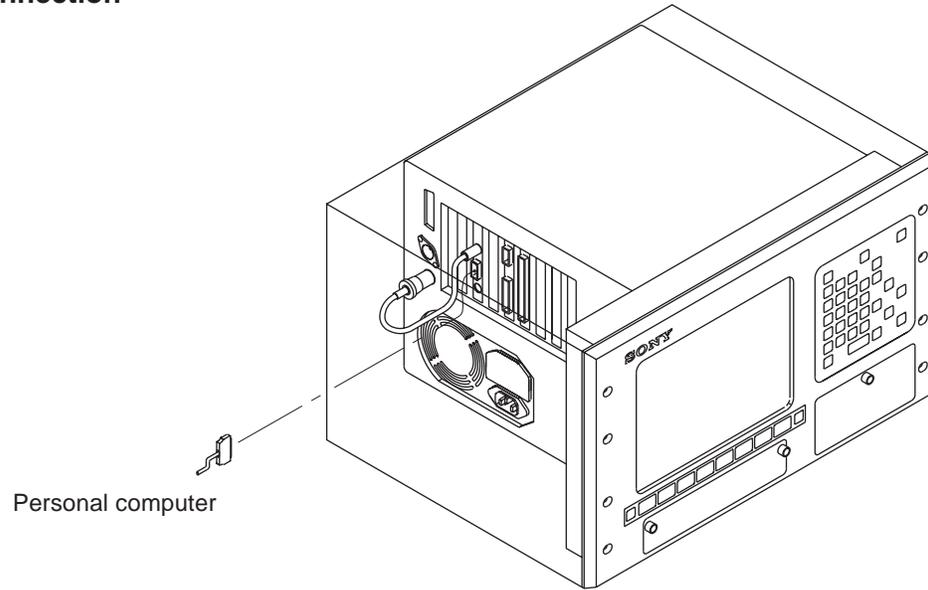
2) Connecting cable

Cables with following configuration is recommended.

D-Sub 9P			D-Sub 9P	D-Sub 25P	
RD	2		2	3	RD
SD	3		3	2	SD
GND	5		5	7	GND
RTC	7		7	4	RTC
CTS	8		8	5	CTS

or

RD	2		2	3	RD
SD	3		3	2	SD
GND	5		5	7	GND
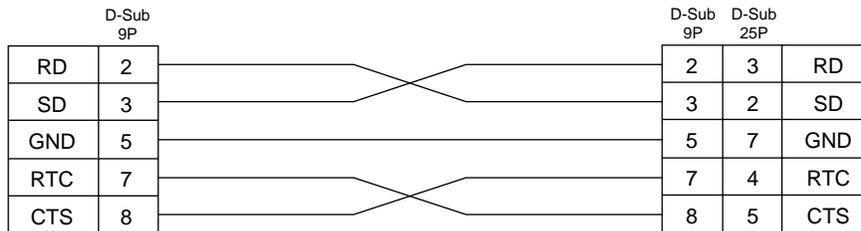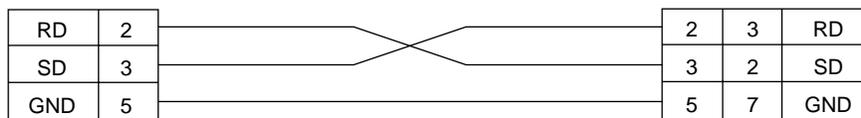
3) Settings

Default setting up of this unit is as follows.
Please refer to Sony Precision Technology or its dealers when any change in setting is required.

Bits per second : 9600
Data bits : 7
Parity : Even
Stop bits : 2
End of block : CR + LF
Flow control : X on / X off
DC code : Disable

4)   Flow chart of RS-232C communication

1. PROGRAM SEND

| NU20 | → | External equipment |

Steps   **Key used**

| Enter Program mode | MODE → | F2 | (PROG)

↓

| Select file | ⬅ ➡ ⬆ ⬇

↓

| External equipment starts to revcive |

↓

⬅ (Function key)

| Start sending | | F1 | (PROG SEND)

↓

| Complete | | ENTER |

2. PROGRAM RECEIVE

```
┌──────────┐          ┌──────────────────────┐
│  NU20    │◄─────────│  External equipment  │
└──────────┘          └──────────────────────┘
```

Steps   **Key used**

```
┌──────────────────────────┐
│   Enter Program mode      │          [MODE] ──► [F2]   (PROG)
└──────────────────────────┘
             │
             ▼
┌──────────────────────────┐
│      Select folder        │          [↑]  [↓]
└──────────────────────────┘
             │
             ▼
┌──────────────────────────┐
│      Start receiving      │          [F2] (PROG RECEIV)
└──────────────────────────┘
             │
             ▼
┌────────────────────────────────────┐
│  External equipment starts to send  │
└────────────────────────────────────┘
             │
             ▼
      Yes    ╱◆╲
  ◄─────────╱ * Name of file ╲
           ╲   received    ╱
            ╲◆╱
             │ No
             ▼
┌──────────────────────────┐
│      Enter file name      │      [X] [Y] [Z] [0] - [9]
└──────────────────────────┘
             │
             ▼
┌──────────────────────────┐
│        Complete           │          [ENTER]
└──────────────────────────┘
```
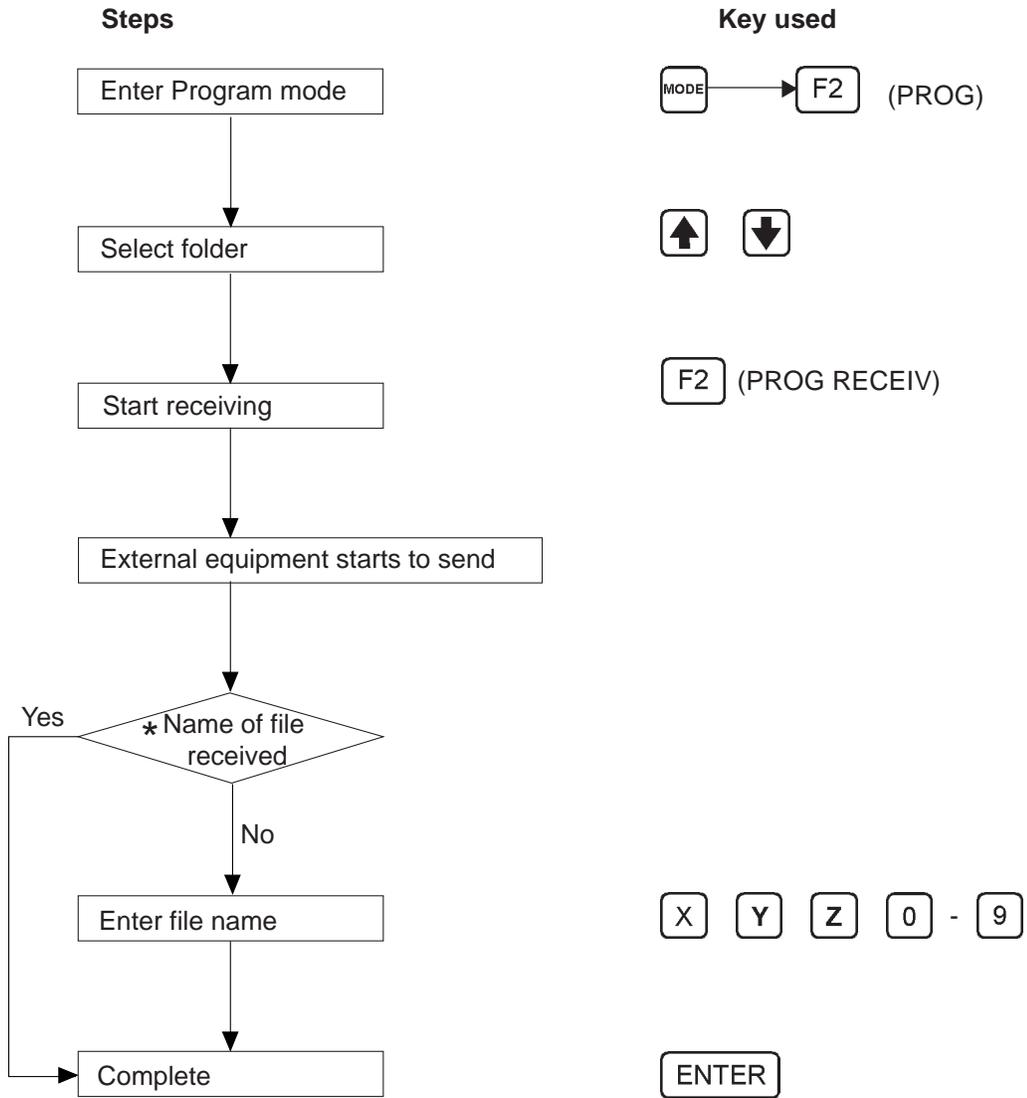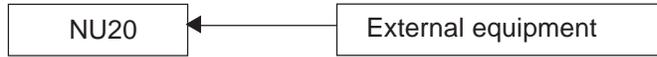
* Characters surrounded between ":" and "CR+LF" at the front line become file name.

3. DNC

Please contact Sony Precision Technology or its distributors as to the G codes which can be executed on NU20.

```
┌─────────┐        ┌──────────────────────┐
│  NU20   │◄───────│  External equipment  │
└─────────┘        └──────────────────────┘
```

Steps   **Key used**

```
        ┌────────────────────────┐              ┌──────┐      ┌──────┐
        │  Enter Program mode    │              │ MODE │─────►│  F2  │   (PROG)
        └────────────────────────┘              └──────┘      └──────┘
                    │
                    ▼
        ┌────────────────────────┐                    ┌──────┐
        │   Enter DNC mode       │                    │  ◄   │   (Function key)
        └────────────────────────┘                    └──────┘
              *     │                                  ┌──────┐
                    ▼                                  │  F5  │   (DNC)
                                                       └──────┘
                            ┌──────────────────────────────┐
                            │  External equipment starts    │
                            │         to send               │
                            └──────────────────────────────┘
                                            │
        ┌────────────────────┐      ┌──────────────────────┐        ┌──────┐
        │ Starting program   │      │  Starting program     │        │  F6  │  (START)
        │       run          │      │        run            │        └──────┘
        └────────────────────┘      └──────────────────────┘
                    │                           │
                    ▼
┌──────────────────────────────┐
│  External equipment starts    │
│         to send               │
└──────────────────────────────┘
                    │
                    ▼
              ┌────────────────────────┐
         ┌───►│      Machining         │      * Either of these two procedures operable
         │    └────────────────────────┘
         │                │
      No │                ▼
         │          ╱─────────────╲
         └─────────╱  M02 or M30 or ╲
                   ╲   % received   ╱
                    ╲─────────────╱
                          │
                         Yes
                          ▼
              ┌────────────────────────┐
              │      Complete          │
              └────────────────────────┘
```
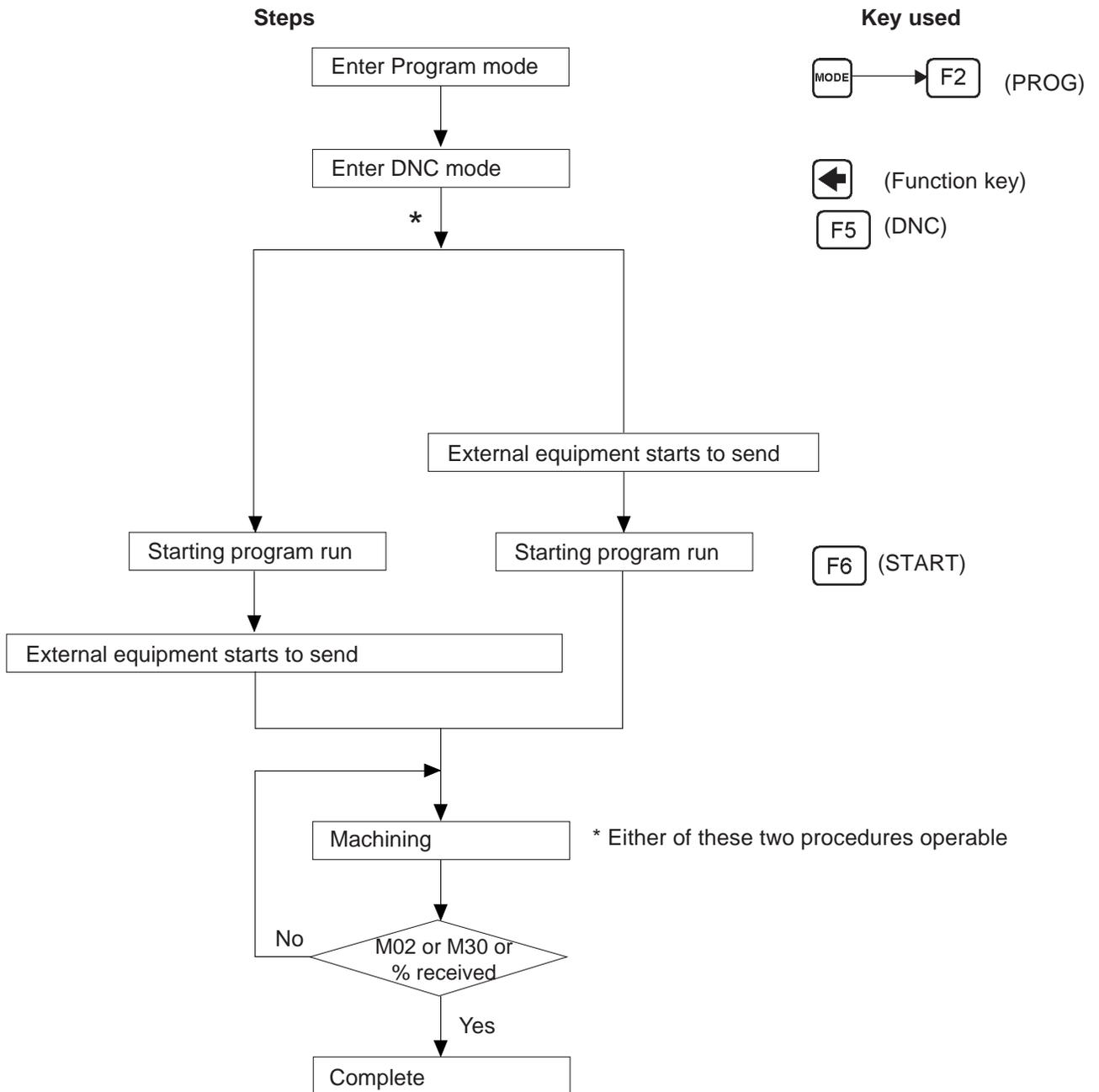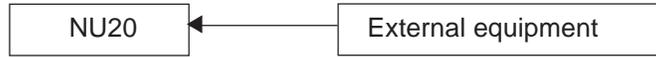
5)　Example of communication with Hyper Terminal used

Please follow this procedure to communicate with NU20 by starting Hyper Terminal included in Windows® from the external PC.

1. PROGRAM SEND

```
┌──────────┐                      ┌──────────────┐
│   NU20   │  ──────────────────▶ │ External PC  │
└──────────┘                      └──────────────┘

                                  ┌────────────────────────┐
                                  │ Starting Hyper Terminal │
                                  └────────────────────────┘
                                              │
                                              ▼
                                  ┌────────────────────────┐
                                  │   Setting Properties    │
                                  └────────────────────────┘

                                  • Bits per second    9600
                                  • Data bits           7
                                  • Parity             Even
                                  • Stop bits           2
                                  • Flow control     Xon/Xoff
                                              │
                                              ▼
┌────────────────────┐            ┌────────────────────────┐
│ Enter Program mode │            │ Starting receiv command │
└────────────────────┘            └────────────────────────┘
          │                          (Transfer/Capture Text)
          ▼                                   │
┌────────────────────┐                        ▼
│    Select file     │            ┌────────────────────────┐
└────────────────────┘            │    Enter file name      │
          │                       └────────────────────────┘
          ▼                                 (Open)
┌────────────────────┐                        │
│   Start sending    │   File transfer        │
└────────────────────┘   ------------------▶  │  *
          │                                    │
          ▼                                    ▼
┌────────────────────┐  (Enter)   ┌──────────┐
│     Complete       │            │   Exit   │
└────────────────────┘            └──────────┘
```
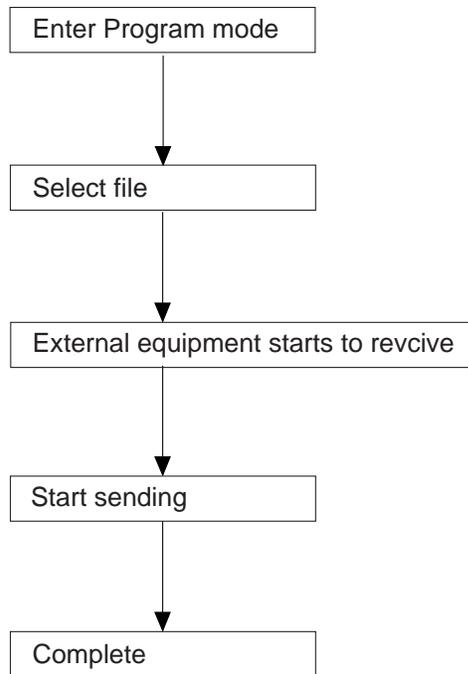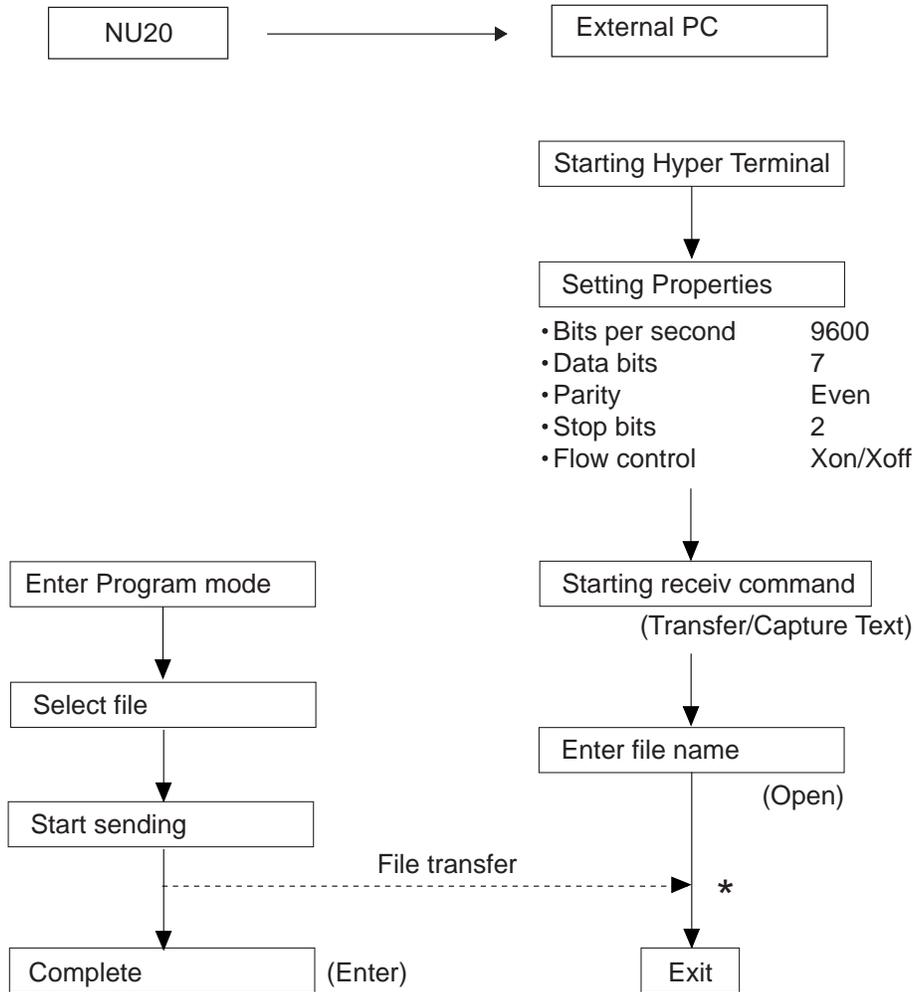
* When the file can not be received properly please check Properties again.

2. PROGRAM RECEIV

```
┌──────────┐            ┌──────────────┐
│   NU20   │ ◄───────── │ External PC  │
└──────────┘            └──────────────┘
```

```
                        ┌─────────────────────────┐
                        │ Starting Hyper Terminal │
                        └─────────────────────────┘
                                    │
                                    ▼
                        ┌─────────────────────────┐
                        │ Setting Properties      │
                        └─────────────────────────┘
```

- Bits per second   9600
- Data bits   7
- Parity   Even
- Stop bits   2
- Flow control   X on / X off

```
┌──────────────────────┐         ┌──────────────────────┐
│ Enter Program mode   │         │ Starting send command│
└──────────────────────┘         └──────────────────────┘
           │                       (Transfer/Send Text File)
           ▼                                  │
┌──────────────────────┐                      ▼
│ Select folder        │         ┌──────────────────────┐
└──────────────────────┘         │ File selection       │
           │                     └──────────────────────┘
           ▼                                  (Open)
┌──────────────────────┐
│ Start receiving      │
└──────────────────────┘
           │◄╌╌╌╌╌ File transfer ╌╌╌╌╌╌╌╌╌╌╌╌╌┐
           ▼                                  ▼
                                        ┌──────────┐
Yes     ◇ Name of file ◇                │   Exit   │
        ◇  received    ◇                └──────────┘
           │ No
           ▼
┌──────────────────────┐
│ Enter file name      │
└──────────────────────┘
           │
           ▼
┌──────────────────────┐
│ Complete             │  (Enter)
└──────────────────────┘
```
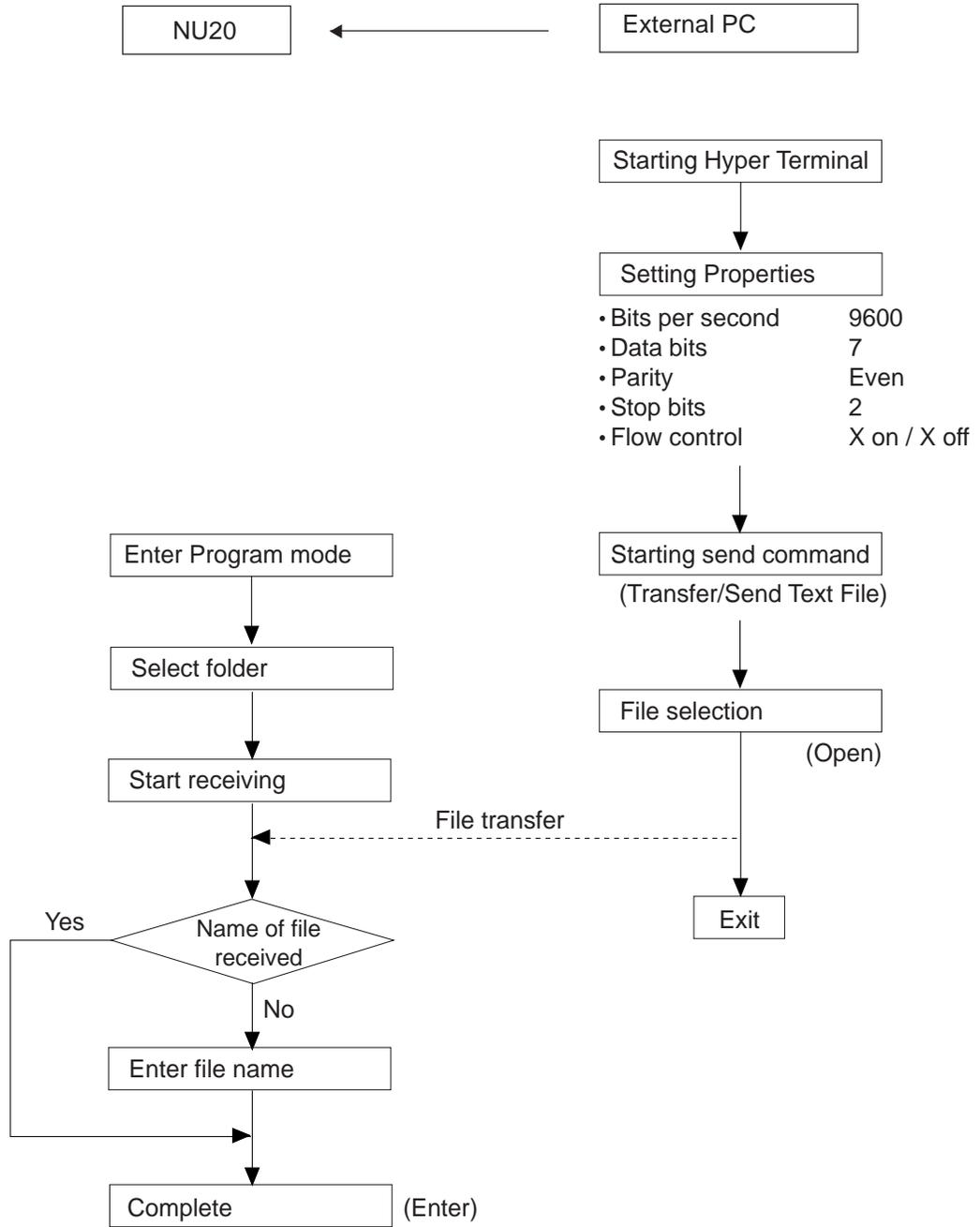
11. Tool Data

TOOL DATA mode is for storing the necessary tool information for machining. The settings entered here are used as the tool diameter compensation values during MDI machining and program machining. These values are not used for manual machining in MANUAL mode.

1)   Key Description

1. Press the [MODE] key.

2. Press the [F4] (TOOL DATA) key.

3. The system enters Tool Data Input mode. The following function keys can be used.

TOOL DATA	SOFT KEY	TOOL LIST		LENGTH SET	DIA. /RAD.	SELECT TOOL	PAGE BACK	

Key Name	Function

TOOL DATA:

This is the selector key used for entering tool data.

SOFT KEY:

This key is used when entering the tool name or comment. The entry must be in alphabet characters.

TOOL LIST:

Normally, frequently used tools are shown in the tool name list. Use this list when entering the tool name.

LENGTH SET:

This function is used to enter the Z-axis work position as the length data.

DIA./RAD.:

This setting is used to set the tool width display to diameter or radius.

2)   Entering Tool Data

1. Entering the tool name
2. Entering the tool width
3. Entering the tool length
4. Entering a comment

1. Entering the tool name

    1.1   Press the `MODE` key.

    1.2   Press the `F4` (TOOL DATA) key.

    1.3   Press the `F1` (TOOL DATA) key.

    1.4   Enter the tool name. Use the cursor keys to move to the Tool Name box.

    1.5   Press the `F2` (SOFT KEY). You can enter the name now. Use the cursor keys to select the letters, and then press the ENTER key to register the entry.

             To enter the tool name from TOOL LIST, press the `F3` (TOOL LIST) key. Use the cursor keys to select the tool name, and then press the ENTER key to register the entry.

    1.6   Use the `CAN` or `DEL` keys to make corrections if necessary.

2. Entering the tool width

    2.1   Entering the tool width

    2.2   Press the `F1` (TOOL DATA) key.

    2.3   Use the cursor keys to move to the Tool Width box.

    2.4   First, check whether RAD. (radius) or DIA. (diameter) is displayed at the top of the screen, and then use the numeric keys to enter the tool width.

3. Entering the tool length

    3.1   Entering the tool length

    3.2   Press the `F1` (TOOL DATA) key.

    3.3   Use the cursor keys to move to the Tool Length box.

    3.4   Use the numeric keys to enter the tool length.

4. Entering a comment

    4.1   If necessary, enter a comment in the comment column.

    4.2   Use the cursor keys to move to the Comment box.

    4.3   Press the `F2` (SOFT KEY) key. You can enter the comment now. Use the cursor keys to select the letters, and then press the ENTER key to register the entry.

3) Selecting the Tool to be Used

Be sure to always check the tool selected in TOOL DATA before performing actual machining. For MDI machining and program machining, add an offset by referring to the tool diameter selected in TOOL DATA.

1. Go to the TOOL DATA Setting screen.
2. The tool currently being used is displayed at "SELECTED TOOL No." at the top of the screen.
3. To change the tool, first, press the F1 key.
4. Use the ⬆ ⬇ cursor keys to select the desired tool.
5. Press the F7 (TOOL SELECT) key. This selects the tool.

4) Entering Tool length with Teaching function

Enter the length data with Teaching instead of numeric keys.
Move Z-axis by the mechanical handle or NZ30 (JOGMPG unit), display the difference of lengths of the standard tool and the selected tool as the work position, and set the length data of the selected tool.

1. Move and adjust the tool's end to the reference position by the mechanical handle or NZ30, and clear Z-axis work position to "0".
2. Enter the tool name, tool width and comment in TOOL DATA mode.
3. Enter the tool length by Teaching.

3.1 Press the F1 (TOOL DATA) key.

3.2 Press the F5 (LENGTH SET) key.

3.3 Adjust the tool's end, which was selected by the mechanical handle or NZ30, to the reference position.

3.4 Press the F4 (TEACH) key.

3.5 Now the length data of the selected tool is entered.

12. Touch Sensor

Touch Sensor mode allows you to bring the Touch Sensor (option) in contact with the workpiece to set the workpiece coordinate system based on the workpiece edge surface. This mode also allows you to measure the workpiece dimensions and angle. The Mill Station is provided with seven special-purpose menus to respond to a variety of different measuring needs. These seven special menus each have a PROMPT MESSAGE window at the bottom right of the screen. This PROMPT MESSAGE window guides you through the measurement process by displaying messages asking you step by step how you want the measurement to be performed. This allows measurements to be taken easily and quickly without errors.

12-1.   Before Operation

Stop the spindle, and install the Touch Sensor on the spindle. Connect the Touch Sensor to the machine using the special cable provided for that purpose. As shown in the figure below, the center of the Touch Sensor ball measures the workpiece at the contact point with the workpiece. Jog feed, Manual Pulse Wheel feed, or the machine hand wheel can be used.
Before entering Touch Sensor mode, set JOG FEED to 1.5 inch/min (38.1 mm/min) or less in Manual mode.
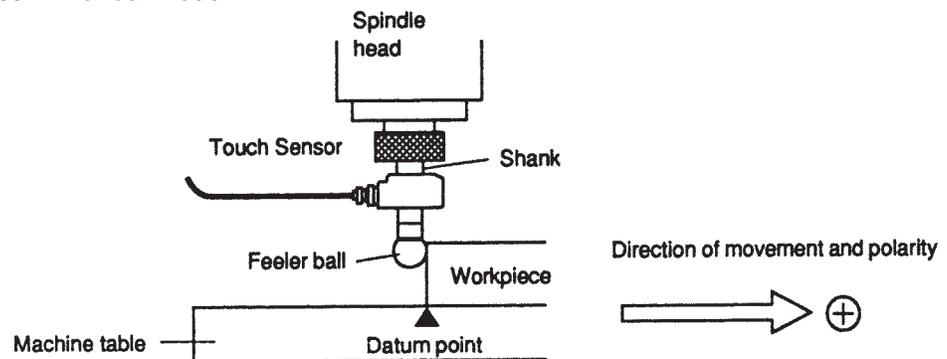
Caution
• Be careful not to rotate the spindle by mistake. This is extremely dangerous. If the spindle is rotated when the Touch Sensor is installed, the Touch Sensor or cable can break.

Note
• The Touch Sensor cannot be used for workpieces with no conductivity.
• The Z-axis workpiece coordinate system settings and workpiece height direction measurement cannot be performed with the Touch Sensor.
• When the Touch Sensor is brought into contact with the workpiece, a buzzer sounds for about 0.5 seconds. Be careful when handling the Touch Sensor since it can be damaged if moved an additional 0.02 inches (0.508 mm) once contact is made.
• If the Touch Sensor is brought into contact with the workpiece at Jog Feed, the Touch Sensor decelerates and brings the workpiece to a stop as soon as it contacts the workpiece. Set the Jog Feed speed so that the workpiece does not continue to move after contact and damage the Touch Sensor.
• To make accurate measurements, it is recommended that the jog feed rate be set to 1.5 inch/min (38.1 mm/min) or less.
• Changes to the jog feed rate and override cannot be made in Touch Sensor mode. The jog feed rate and override settings must be made beforehand in Manual mode.
• If Manual Pulse wheel feed or the machine hand wheel is used to bring the Touch Sensor into contact with the workpiece, be aware that the Touch Sensor will not automatically decelerate the workpiece and bring it to a stop.

12-2.   Touch Sensor Modes

This section describes the seven measurement menus provided in Touch Sensor mode.

1.   Press the [MODE] key.

2.   Press the [F1] (MANUAL) key.

3.   Press the [◀] or [▶] function key.

4.   Press the [F5] (TOUCH SENSOR) key.

5.   The system enters TOUCH SENSOR/FUNCTION MENU.
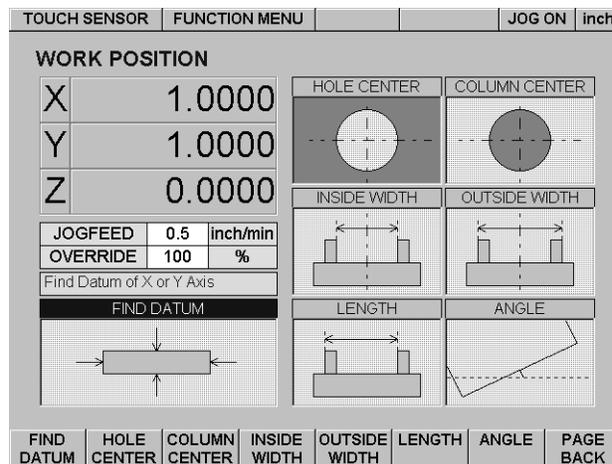
2) Key Description

Use the F1 to F7 keys, cursor keys, and ENTER key to select the intended measurement menu. To exit the TOUCH SENSOR/FUNCTION MENU mode, press the F8 (PAGE BACK) key.

Key Name	Function
FIND DATUM (F1):	
	This function is used to set the workpiece coordinates (ABS coordinates) based on the workpiece edge surface.
HOLE CENTER (F2):	
	This function is used to measure the hole center.
COLUMN CENTER (F3):	
	This function is used to measure the center of the column.

Key Name	Function
DRINSIDE WIDTH (F4):	
	This function is used to measure the inside width and find the center.
OUTSIDE WIDTH (F5):	
	This function is used to measure the outside width and find the center.
LENGTH (F6):	
	This function is used to measure the length between two points.
ANGLE (F7):	
	This function is used to measure the angle between two points.

12-3. Setting the Workpiece Coordinate System (FIND DATUM)

Set the workpiece coordinate system based on the workpiece edge surface.

1. Press the MODE key.

2. Press the F1 (MANUAL) key.

3. Press the ← or → function key.

4. Press the F5 (TOUCH SENSOR) key.

5. Press the F1 (FIND DATUM) key.

	FIND DATUM	HOLE CENTOR	COLUM CENTOR	INSIDE WIDTH	OUTSIDE WIDTH	LENGTH	ANGLE	PAGE BACK	

1) Key Description

Key Name	Function
START:	
	This key is used to start the measurement.
AXIS SELECT:	
	This key is used to select either the X-axis or the Y-axis as the measurement axis. The setting changes each time the key is pressed.
DIR. SELECT:	
	This key is used to select the direction of the measurement surface. You can select either to measure from the position direction or negative direction.

TS DIA SET:

> This key is used to enter the diameter of the Touch Sensor. When this setting is entered, the system automatically performs diameter offset for the measurement value.

ABORT:

> This key is used to abort a measurement while it is in progress.

2)   Operation Procedure

1. Entering the diameter of the Touch Sensor

2. Selecting the axis to be measured

3. Selecting the edge contacting the workpiece

4. Measuring

---

1.   Entering the diameter of the Touch Sensor

1.1   Press the ⎍F6⎍ (TS DIA SET) key.

1.2   Use the numeric keys to enter the diameter for the Touch Sensor edge.

1.3   Press the ⎍ENTER⎍ key to register the setting.

2.   Selecting the axis to be measured

2.1   Press the ⎍F4⎍ (AXIS SELECT) key. The selected axis is changed each time the key is pressed.

2.2   Select the axis to be measured.

3.   Selecting the edge contacting the workpiece

3.1   Press the ⎍F5⎍ (DIR. SELECT) key. The direction is changed each time the key is pressed.

3.2   Select the direction to be measured.

4.   Measuring

4.1   Press the ⎍F1⎍ (START) key.

4.2   The system enters the Measurement Standby state.

4.3   Jog feed the table, and then slowly feed the Touch Sensor to the workpiece surface using the Manual Pulse Wheel or the machine hand wheel. The feed rate at this time should be 1.5 inch/min (38.1 mm/min) or less.

4.4   The measurement is finished when a beeping sound is heard. Perform the measurement for the other axis in the same way.

12-4.   Setting the Hole Center (HOLE CENTER)

Find the center of the hole opened in the workpiece.

1.   Press the |MODE| key.

2.   Press the [F1] (MANUAL) key.

3.   Press the [←] or [→] function key.

4.   Press the [F2] (TOUCH SENSOR) key.

5.   Press the [F2] (HOLE CENTER) key.

1)   Key Description

Key Name	Function
START:	
	This key is used to start measurement.
SELECT P1 - P4:	
	This is the input start key for the measurement points. This screen appears when the START key is pressed. The measurement points are displayed in the following order: P1 → P2 → P3 → P4.
ABORT:	
	This key is used to abort a measurement while it is in progress.
DATUM SET:	
	This key sets the center value found by measurement as the zero point of the workpiece coordinates.

2)   Operation Procedure

1.   Starting P1 input

2.   P2 input

3.   P3 input

4.   P4 input

5.   Setting the measured value as the zero point of the workpiece coordinate system

1. **Starting P1 input**

    1.1   Press the ☐F1☐ (START) key.

    1.2   The F1 key display changes to the SELECT P1 display. The setting input box flashes to indicate that the system is waiting for the P1 input. Jog feed the table, and then slowly feed the Touch Sensor to the workpiece surface using either the Manual Pulse Wheel or the machine hand wheel. The feed rate at this time should be 1.5 inch/min (38.1 mm/min) or less. Bring the Touch Sensor into contact with the side of the hole corresponding to the position shown on the display screen. The measurement is finished when a beeping sound is heard.

2. **P2 input**

    Press the ☐F1☐ key, and perform the measurement for P2 in the same way.

3. **P3 input**

    Press the ☐F1☐ key, and perform the measurement for P3 in the same way.

4. **P4 input**

    Press the ☐F1☐ key, and perform the measurement for P4 in the same way. The measurements are now complete.
    When the measurements are finished, the center value is calculated and then displayed on the screen.

5. **Setting the measured value as the zero point of the workpiece coordinate system**

    When the ☐F5☐ (DATUM SET) key is pressed, the center point which was measured is set as the zero point.

12-5.　Setting the Column Center (COLUMN CENTER)

This function is used to find the column center.

1. Press the [MODE] key.

2. Press the [F1] (MANUAL) key.

3. Press the [←] or [→] function key.

4. Press the [F2] (TOUCH SENSOR) key.

5. Press the [F3] (COLUMN SENSOR) key.

1)　Key Description

Key Name	Function
START:	
	This key is used to start measurement.
SELECT P1 - P4:	
	This is the input start key for the measurement points. This screen appears when the START key is pressed. The measurement points are displayed in the following order: P1 → P2 → P3 → P4.
ABORT:	
	This key is used to abort a measurement while it is in progress.
DATUM SET:	
	This key sets the center value found by measurement as the zero point of the workpiece coordinates.

2)　Operation Procedure

1. Starting P1 input

2. P2 input

3. P3 input

4. P4 input

5. Setting the measured value as the zero point of the workpiece coordinate system

1. **Starting P1 input**

    1.1   Press the [F1] (START) key.

    1.2   The F1 key display changes to the SELECT P1 display. The setting input box flashes to indicate that the system is waiting for the P1 input. Jog feed the table, and then slowly feed the Touch Sensor to the workpiece surface using either the Manual Pulse Wheel or the machine hand wheel. The feed rate at this time should be 1.5 inch/min (38.1 mm/min) or less. Bring the Touch Sensor into contact with the side of the column according to the diagram on the display screen. The measurement is finished when a beeping sound is heard.

2. **P2 input**

    2.1   Press the [F1] key, and perform the measurement for P2 in the same way.

3. **P3 input**

    3.1   Press the [F1] key, and perform the measurement for P3 in the same way.

4. **P4 input**

    4.1   Press the [F1] key, and perform the measurement for P4 in the same way. The measurements are now complete.

5. **Setting the measured value as the zero point of the workpiece coordinate system**

    5.1   When the [F5] (DATUM SET) key is pressed, the center point which was measured is set as the zero point.

12-6.   Inside Width Measurement and Center Calculation (INSIDE WIDTH)

This function is used to measure the inside width and find the center.

1. Press the [MODE] key.

2. Press the [F1] (MANUAL) key.

3. Press the [←] or [→] function key.

4. Press the [F2] (TOUCH SENSOR) key.

5. Press the [F4] (INSIDE WIDTH) key.

1)   Key Description

Key Name	Function
START:	
	This key is used to start measurement.
SELECT P1 - P2:	
	This is the input start key for the measurement points. This screen appears when the START key is pressed. The measurement points are displayed in the following order: P1 → P2.

ABORT:	
	This key is used to abort a measurement while it is in progress.
DATUM SET:	
	This key sets the center value found by measurement as the zero point of the workpiece coordinates.
TS DIA SET:	
	This key is used to enter the diameter of the Touch Sensor. When this setting is entered, the system automatically performs diameter offset for the measurement value.

2) Operation Procedure

1. Entering the diameter of the Touch Sensor
2. Starting P1 input
3. Selecting the axis to be measured
4. P1 input
5. P2 input
6. Setting the measured value as the zero point of the workpiece coordinate system

1. Entering the diameter of the Touch Sensor

1.1 Press the $\boxed{\text{F6}}$ (TS DIA SET) key.

1.2 Use the numeric keys to enter the diameter of the Touch Sensor edge being used.

1.3 Press the $\boxed{\text{ENTER}}$ key to register the settings.

2. Starting P1 input

2.1 Press the $\boxed{\text{F1}}$ (START) key.

3. Selecting the axis to be measured

3.1 Use the $\boxed{\text{X}}$ key or the $\boxed{\text{Y}}$ key to select the axis to be measured.

3.2 The F1 key display changes to the SELECT P1 display.

4. Starting P1 input

4.1 Press the $\boxed{\text{F1}}$ (SELECT P1) key.

4.2 The setting input box flashes to indicate that the system is waiting for the P1 input. Jog feed the table, and then slowly feed the Touch Sensor to the workpiece surface using either the Manual Pulse Wheel or the machine hand wheel. The feed rate at this time should be 1.5 inch/min (38.1 mm/min) or less. Bring the Touch Sensor into contact with the side of the workpiece shown on the display screen. The measurement is finished when a beeping sound is heard.

4.3 The F1 key display changes to the SELECT P2 display.

5. P2 input

    5.1   Press the F1 (SELECT P2) key, and perform the measurement for P2 in the same way as P1. When measurement is finished, the inside width and center value are displayed on the screen.

6. Setting the measured value as the zero point of the workpiece coordinate system

    6.1   When the F5 (DATUM SET) key is pressed, the center point which was measured is set as the zero point.

12-7.   Outside Width Measurement and Center Calculation

This function is used to measure the outside width and find the center.

1. Press the MODE key.

2. Press the F1 (MANUAL) key.

3. Press the ← or → function key.

4. Press the F2 (TOUCH SENSOR) key.

5. Press the F5 (OUTSIDE WIDTH) key.

1)   Key Description

Key Name	Function
START:	
	This key is used to start measurement.
SELECT P1 - P2:	
	This is the input start key for the measurement points. The measurement points are displayed in the following order: P1 → P2.
ABORT:	
	This key is used to abort a measurement while it is in progress.
DATUM SET:	
	This key sets the center value found by measurement as the zero point of the workpiece coordinates.
TS DIA SET:	
	This key is used to enter the diameter of the Touch Sensor. When this setting is entered, the system automatically performs diameter offset for the measurement value.

2) Operation Procedure

1. Entering the diameter of the Touch Sensor
2. Starting P1 input
3. Selecting the axis to be measured
4. P1 input
5. P2 input
6. Setting the measured value as the zero point of the workpiece coordinate system

---

1. Entering the diameter of the Touch Sensor

1.1 Press the $\boxed{\text{F6}}$ (TS DIA SET) key.

1.2 Use the numeric keys to enter the diameter of the Touch Sensor being used.

1.3 Press the $\boxed{\text{ENTER}}$ key to register the settings.

2. Starting P1 input

2.1 Press the $\boxed{\text{F1}}$ (START) key.

3. Selecting the axis to be measured

3.1 Use the $\boxed{\text{X}}$ key or the $\boxed{\text{Y}}$ key to select the axis to be measured.

3.2 The F1 key display changes to the SELECT P1 display.

4. Starting P1 input

4.1 Press the $\boxed{\text{F1}}$ (SELECT P1) key.

4.2 The setting input box flashes to indicate that the system is waiting for the P1 input. Jog feed the table, and then slowly feed the Touch Sensor to the workpiece surface using either the Manual Pulse Wheel or the machine hand wheel. The feed rate at this time should be 1.5 inch/min (38.1 mm/min) or less. Bring the Touch Sensor into contact with the side of the workpiece as shown in the diagram on the display screen. The measurement is finished when a beeping sound is heard.

5. P2 input

5.1 Press the $\boxed{\text{F1}}$ (SELECT P2) key, and perform the measurement for P2 in the same way as P1. When measurement is finished, the outside width and center are displayed on the screen.

6. Setting the measured value as the zero point of the workpiece coordinate system

6.1 When the $\boxed{\text{F5}}$ (DATUM SET) key is pressed, the center point which was measured is set as the zero point.

12-8. Measuring the Length Between Two Points (LENGTH)

This function is used to measure the length between any two points on the workpiece.

1. Press the [MODE] key.

2. Press the [F1] (MANUAL) key.

3. Press the [←] or [→] function key.

4. Press the [F2] (TOUCH SENSOR) key.

5. Press the [F6] (LENGTH) key.

1) Key Description

Key Name	Function
START:	
	This key is used to start measurement.
SELECT P1 - P2:	
	This is the input start key for the measurement points. The measurement points are displayed in the following order: P1 → P2.
ABORT:	
	This key is used to abort a measurement while it is in progress.

2) Operation Procedure

1. Selecting the axis to be measured
2. P1 input
3. P2 input

1. Selecting the axis to be measured

1.1 Press the [F1] (START) key.

1.2 Use the [X] key or the [Y] key to select the axis to be measured.

1.3 The F1 key display changes to the SELECT P1 display.

2. Starting P1 input

2.1 Press the ⌊ F1 ⌋ (SELECT P1) key.

2.2 The setting input box flashes to indicate that the system is waiting for the P1 input. Jog feed the table, and then slowly feed the Touch Sensor to the workpiece surface using either the Manual Pulse Wheel or the machine hand wheel. The feed rate at this time should be 1.5 inch/min (38.1 mm/min) or less. Bring the Touch Sensor into contact with the side of the workpiece as shown in the diagram on the display screen. The measurement is finished when a beeping sound is heard.

3. P2 input

3.1 Press the ⌊ F1 ⌋ key, and perform the measurement for P2 in the same way as P1. When measurement is finished, the distance (length) between the two points is displayed.

12-9. Measuring the Angle Between Two Points (ANGLE)

This function is used to measure the angle between any two points on the workpiece.

1. Press the ⌊MODE⌋ key.

2. Press the ⌊ F1 ⌋ (MANUAL) key.

3. Press the ⌊◀⌋ or ⌊▶⌋ function key.

4. Press the ⌊ F2 ⌋ (TOUCH SENSOR) key.

5. Press the ⌊ F7 ⌋ (ANGLE) key.

1) Key Description

Key Name	Function
START:	
	This key is used to start measurement.
SELECT P1 - P2:	
	This is the input start key for the measurement points. The measurement points are displayed in the following order: P1 → P2.
ABORT:	
	This key is used to abort a measurement while it is in progress.

2) Operation Procedure

1. Selecting the axis to be measured
2. P1 input
3. P2 input

1. Selecting the axis to be measured

      1.1   Press the ⎡F1⎤ (START) key.

      1.2   The F1 key display changes to the SELECT P1 display.

2. Starting P1 input

      2.1   Press the ⎡F1⎤ (SELECT P1) key.

      2.2   The setting input box flashes to indicate that the system is waiting for the P1 input. Jog feed the table, and then slowly feed the Touch Sensor to the workpiece surface using either the Manual Pulse Wheel or the machine hand wheel. The feed rate at this time should be 1.5 inch/min (38.1 mm/min) or less. Bring the Touch Sensor into contact with the side of the workpiece as shown in the diagram on the display screen. The measurement is finished when a beeping sound is heard.

3. P2 input

      3.1   Press the ⎡F1⎤ key, and perform the measurement for P2 in the same way as P1. When measurement is finished, the angle between the two points is displayed.

13. More Information

This section describes the various diagnostic functions, compensation functions, and system parameters in the NU20. Generally, the functions described here are internal functions which cannot be set or changed by the user. However, the functions are described here to enhance understanding of NU20 capabilities and to aid in troubleshooting if ever a fault of any kind occurs.

13-1. Diagnostic Function

• How to display the diagnostic function

The diagnostic functions allows you to view the I/O check, alarm check, alarm list, and other indicators of the internal system state. When the customer makes a service inquiry due to a fault, the serviceman will check this diagnostic information. Refer to the descriptions below in case he asks you to check the diagnostic function.

1. Press the `MODE` key.

2. Press the `F7` (DIAGNOSTIC) key.

I/O CHECK:

This function is used to display the status of the internal I/O contact.

ALARM CHECK:

This function is used to display detailed information when alarm occurs.

ALARM LIST:

This function is used to display the history of alarms which have occurred.

ABOUT:

This function is used to display the date and software version number.

• How to set the date.

1. Press the `MODE` key.

2. Press the `F7` (DIAGNOSTIC) key.

3. Press the `F6` (ABOUT) key.

4. Move the cursor by `↑` `↓` keys to the desired position, and enter the date by `0` – `9` keys and press the `F1` (STORE TIME) key to determine it.

13-2.   System Parameters

⚠️ **Warning**

The system parameters are important data which determine the operation of the machine. Do not change these settings without the permission of Sony Precision Technology. Doing so could result in a malfunction.

Note

At present, the system parameters have been set so that they cannot be changed by the user. To change them, please contact Sony Precision Technology or your local dealer.

• How to load and exit the system parameters

1. Press the MODE key.

2. Press the F6 (SET SYSTEM) key.

3. Use the ∧ ∨ keys to select the category, ⬆ ⬇ keys to select the number, and the ⬅ ➡ keys to select the settings.

   Press the F1 (FIND) key, and enter the number. You can also search by pressing the ENTER key.

4. Press the F8 (BACK PAGE) key. This returns you to the previous screen.

13-3.   Compensation Function

Generally a machine tool has its inherent geometric error. For example, with a knee type milling 8 machine, the knee is slightly tilted as the table moves and the horizontal component of this inclination is added to the scale displacement as an error. The ball screws for moving the table also have a certain amount of play, so that an error also results between the advancing and returning of the table.
The NU20 has internal functions for compensating for these errors. These functions are described below.

1)   Backlash Compensation

This function compensates for backlash due to the ball screws. It works only during semi-closed loop control. This function is disabled until the zero point is set. If the zero point sensor is not installed, perform the reference point return operation manually.

The compensation amount can be set from 0 to 32,767 (μm). The compensation amount and other settings are input as system parameters during NU20 installation.

2)   Linear Compensation

This function compensates for the error due to table tilting.
Error compensation is made by adding or subtracting a compensation amount from/to the encoder reading when a fixed distance is moved.

The compensation amount can be set from −999 to +999 μm/m. The compensation amount and other settings are input as system parameters during NU20 installation.

3)   Sectional Error Compensation

The measuring scale length can be divided into a maximum of 32 sections. Error compensation is then performed for each of these sections. This function is disabled until the zero point is set. Compensation is based on the zero point position.
If the zero point sensor is not installed, the reference point return position will not be the same each time that the power is turned on. Use of the section error compensation function is not recommended in this case.

1) Axis to be compensated: X, Y axis
2) Number of points to be compensated: Up to 32 points/axis
3) Maximum compensation amount: ±127 (least command increment)
4) Minimum compensation section width: 256 (least command increment)

The compensation amount and other settings are input as system parameters during NU20 installation.

Note
At present, the system parameters have been set so that they cannot be changed by the user. To change them, please contact Sony Precision Technology or your local dealer.

14. Troubleshooting

14-1. Before Assuming a Fault

Symptom	Cause	Remedy
Rapid traverse cannot be performed in JOG.	The reference point return operation has not been performed.	Perform the reference point return operation.
The table was supposed to be moving at a constant rate during Jog Feed or while running a program, but the table speed slowed in the middle of the movement.	If SLOW is displayed at the top of the screen, the table is being slowed by the SOFTWARE LIMIT function.	Allow the table to move inside the SOFTWARE LIMIT area. However, it is normal for the speed to change when tracing a circular path.
The table was supposed to be moving at a constant rate during Jog Feed or while running a program, but the table came to a sudden stop in the middle of the movement.	The table is being stopped by the SAFE ZONE function.	Either change the SAFE ZONE settings or disable the SAFE ZONE function.
No operation is performed when the JOG key is pressed.	The machine is in a mode other than MANUAL mode.	Set to MANUAL mode.
The NOT READY display is not shown at the top of the screen, and no operation is performed.	The STOP switch has been pressed.	If the JOGMPG unit is installed at the NU20, turn the STOP switch clockwise to unlock it.
The table moves faster (slower) than the programmed setting speed.	The OVERRIDE value is not at 100%.	Set the OVERRIDE value to 100%.
When the machine is turned on, the blue screen is displayed and the system enters the SCAN DISC status.	The power was not turned off normally last time for some reason. To turn off the power, press the POWER OFF key on the NU20 front panel.	At the blue screen, press the ENTER key. After the operating system performs scan disc, the system starts normally. If the system does not start normally, contact Sony Precision Technology.

14-2.   List of Alarms and Messages

Alarm	Description	Cancel
Axis Servo Alarm X, Y, Z	This alarm occurs when the drive system (motor, amplifier, mechanical system) scale is abnormal and the position deviation limit value set by the controller system parameter. Check the drive system.	Press the STOP switch, and then remove the cause of the alarm before turning the power on again.
Feedback Alarm X, Y, Z	This alarm occurs drive system (motor amplifier, mechanical system) scale or connection wiring is abnormal, or when the servo direction set by the controller system parameter is reversed. Check the drive system, scale, and connection wiring.	Same as above.
XYZ Amp Alarm	Either the servo amplifier is in alarm status, or the main power supply for the servo amplifier is off. Check the servo amplifier main power supply and drive system (motor, servo amplifier, and mechanical system). The servo amplifier alarm occurs when the machine has overrun.	Same as above.
Axis Over Speed X, Y, Z	This alarm occurs when the machine has moved at an excessive speed. This alarm also occurs when the position sensor is faulty or a counter error occurs due to noise. Check the drive system (motor, servo amplifier, mechanical system).	Same as above.
Axis Scale X, Y, Z	This alarm occurs when an alarm signal (AL, *AL) is detected from the position sensor. Check the connections for the position sensor (scale).	Same as above.
Axis Cut X, Y, Z	This alarm occurs when a break has been detected in the signal wire from the position sensor. Check the connection between the position sensors (scale).	Same as above.
Soft Over Speed X, Y, Z	This alarm occurs when the speed pulse from the position sensor exceeds the limit value set by the controller system parameter (No. 109).	Same as above.

Message	Description	Remedy
LIMIT X+ LIMIT X– LIMIT Y+ LIMIT Y– LIMIT Z+ LIMIT Z–	This message is displayed when the Overtravel Limit Switch is reached. This message is displayed when the Software Limit position is reached. This message is displayed when the Safe Zone is reached.	Use JOG or the MPG to move the machine in the opposite direction on the axis shown in the message until LIMIT +/– no longer appears.
SLOW X+ SLOW X– SLOW Y+ SLOW Y– SLOW Z+ SLOW Z–	This message is displayed when the deceleration start position before the Software Limit position is reached.	If the deceleration is detrimental to the machining, change the workpiece position.
Illegal number format	This message is displayed when unrecognized text is input and then ENTER is pressed.	Correct the text.
Illegal address	This message is displayed when the same text has been entered two or more times and then ENTER is pressed.	Correct the text.
Illegal address value	This message is displayed when an unsuitable value is set and then ENTER is pressed.	Correct the variable.
Illegal G-code same group gcode in one-block	This message is displayed when the same group G code exists in the block.	Input the G code correctly.
Illegal G-code	This message is displayed when an unrecognized G code is input and then ENTER is pressed.	Input the G code correctly.
Illegal loop count L > 99	This message is displayed when an incorrect repeat count is input and then ENTER is pressed.	Input the correct repeat count.
Illegal dowell time	This message is displayed when an incorrect dwell time is entered and then ENTER is pressed.	Correct the dwell time.
Subprogram not find	This message is displayed when the subprogram which was called does not exist.	Designate the correct subprogram.
Illegal M98	This message is displayed when the subprogram exceeds four nested levels.	The nested levels should be four or less.
Illegal G-code in oneshot	This message is displayed when one-shot G codes are designated and executed in the same block.	Execute the G code correctly.
Fixed cycle rpoint < zpoint	This message is displayed when Z has been set and executed in a higher position than R.	Input the correct position.

Message	Description	Remedy
Illegal axis in 2-axis mode	This message indicates that movement of axes which cannot be moved simultaneously was executed. This message also appears when combinations of two or more axes unable to be designated by the tool diameter compensation are set and executed.	Designate the correct axes.
Illegal fixed cycle G-code	This message appears when the a fixed cycle is executed without cancelling the designated fixed cycle.	Cancel the fixed cycle, and then designate the other fixed cycle.
Fixed cycle without X, Y, Z, R	This message appears when an insufficient or incorrect setting value for the designated fixed cycle is executed.	Correct the setting value.
R with 'IJK' in G02/03	This message appears when two are more locations are calculated as the arc center in G02 or G03.	Correct the setting value.
Can't claculate IJ from R	This message appears when a value unrecognized as R is set and the ENTER is pressed.	Correct the value.
R-check error in G02/03	This message appears when the values for the arc end point and R (or I, J) are entered and executed without correctly calculating to 1 μm.	Correct the value.
Illegal plane in G02/03	This message appears when two axes other than the axes selected in the planes by G17, G18, or G19 are executed.	Correct the designation.
Illegal Bcode value	This message appears when a value not recognized as B is set and executed.	Correct the value.
Illegal Dcode value	This message appears when a value not recognized as D is set and executed.	Correct the value.
Illegal Fcode value	This message appears when a value not recognized as F is set and executed.	Correct the value.
Illegal Hcode value	This message appears when a value not recognized as H is set and executed.	Correct the value.
Illegal Lcode value	This message appears when a value not recognized as L is set and executed.	Correct the value.
Illegal Mcode value	This message appears when a value not recognized as M is set and executed.	Correct the value.
Illegal Scode value	This message appears when a value not recognized as S is set and executed.	Correct the value.
Illegal Tcode value	This message appears when a value not recognized as T is set and executed.	Correct the value.
Illegal Pcode value	This message appears when a value not recognized as P is set and executed.	Correct the value.

Message	Description	Remedy
Illegal plane G68	This message is displayed when the selected planes for coordinate rotation are not correct.	Select the correct plane.
E > Tool Radius	This message is displayed when E is larger than the tool radius.	Correct the setting.
P > Tool Diameter	This message is displayed when P is larger than the tool radius.	Correct the setting.
Illegal cut length value	This message is displayed when the workpiece cutting depth is incorrect.	Correct the setting.
Coner R error	This message is displayed when the corner R value is incorrect.	Correct the setting.
TD value error	This message is displayed when the value for the cutting direction is invalid.	Correct the value.
WW value error	This message is displayed when the cutting width for the ring mill is 0 or less.	Correct the value.
U value error	This message is displayed when the U value cannot be cut.	Correct the value.
V value error	This message is displayed when the V value is a value for which cutting cannot be performed.	Correct the value.
RP value error	This message is displayed when the return point RP value is invalid.	Change to a valid value.
A value error	This message is displayed when the angle setting value is not within the 0 to 360° range.	Change to a valid value.
K value error	This message is displayed when the angle setting value is not within the 0 to 360° range.	Change to a valid value.
R > I	This message is displayed when the R setting value is larger than the I value.	Correct the position.
Z > R	This message is displayed when the Z setting value is larger than the R value.	Correct the position.
Z > I	This message is displayed when the Z setting value is larger than the I value.	Correct the position.
Q value error	This message is displayed when the value for the cutting depth Q is invalid.	Correct the value.
Q > Cut Length	This message is displayed when the cutting depth Q is larger than the depth to be cut.	Correct the setting.
FZ value error	This message is displayed when the value for the Z-axis speed FZ is invalid.	Correct the setting.
F value error	This message is displayed when the value for the X, Y-axis speed F is invalid.	Correct the setting.
Tool Diameter > (W – 2E)	The cutting width (WW -2E) for the ring mill is smaller than the tool diameter.	Correct the setting.

Message	Description	Remedy
Number over	This message is displayed when the number of holes to be cut by the random drill exceeds 20.	Change the setting so that the number of holes do not exceed 20.
Tool Diameter > (W – E)	The cutting width (W – E) of the island mill is smaller than the tool diameter.	Correct the value.
d value error	The cycle drill diameter d is not a valid value.	Change to a valid value.
Illegal plane G68 in MDI	This message is displayed when the coordinate rotation (G68) before MDI operation has selected a plane other than the G17 plane.	Change to a valid rotation plane.
Tool Diameter > (H – E)	The cutting width (H – E) of the island mill is smaller than the tool diameter.	Correct the value.
Memory full	This message is displayed when the text string memory is insufficient.	Change the number of characters entered in a single line.
File not open	This message is displayed when the selected file cannot be found.	Select a valid file.
Not found file name	This message is displayed when the parameter file cannot be found.	Set the parameter file to a correct position.
File seek error	This message is displayed when a file reading error occurs.	Select a valid file.
Tool compensation error	An error has occurred in the tool diameter compensation.	Enter the program or tool diameter correctly.

15. Specifications

15-1. Electrical and Mechanical Specifications

Item	NU20-20 U7 / NU20-30 U7
Display Unit	10.4-inch TFT color liquid-crystal display
CPU	Pentium 133 MHz + SH3 (NC board)
RAM	32 MB + 4 MB (NC board)
HDD	2.1 GB
FDD	1.44 MB
OS	Windows®95
Input voltage range	AC 115 V ± 10%, 60 Hz
Certified Operating Temperature Range	32–104°F (0–40°C)
Storage Temperature Range	–4–140°F (–20–60°C)
Certified Operating Humidity Range	20–80% RH (no condensation)
Storage Humidity Range	10–90% RH (no condensation)
Rated Power	200 VA
Dimensions	483 x 362 x 266 mm (not including protruding parts)
Maximum Operating Altitude	1000 m
Mass	Approx. 17 kg

15-2.   Functional Specifications

Item	NU20-20 U7	NU20-30 U7
Display axes	X, Y and Z Z axis being connected with the scale unit functions as a DRO.	X, Y and Z
Control axes	X and Y	X, Y and Z
Number of simultaneously controlled axes	2	3
Least input increment (least command increment)	0.001 mm / 0.0001 inch	
Min. movement increment	0.001mm / 0.0001 inch	
Feedback setting units	Set in integers or fractions (selected using the parameters) 0.0001 to 0.01 mm	
Min. display units	0.001 mm / 0.0001 inch 0.001 deg.	
Max. setting values	±99999.999 mm / ±9999.9999 inches ±99999.999 deg.	
Rapid traverse speed	Actual speed is determined by the machine tool and motor specifications. Rapid traverse speed depends on the feedback setting units.	
Rapid traverse override	0 to 100% in 10% steps	
Cutting feed speed	Cutting feed speed is programmed directly by the F code. Max. cutting feed speed is set by the parameters.	
Cutting feed speed override	0 to 200% in 10% steps	
Dry run	In Dry run mode, feed rate speed consistent with jog feed speed. Parameters can be used to select either rapid traverse speed or jog feed speed for positioning (RAPID TO) operation.	
Automatic acceleration/deceleration	For positioning, linear interpolation, and jog feed, acceleration/deceleration before distribution is automatically performed (linear and exponential).	
Zero point passing speed (when using the optional Zero point sensor)	Zero point is passed at the speed set by parameters during zero point setting.	
Overtravel (each axis) (when using the optional OT unit)	Movement is stopped using overtravel signal for each axis, and then alarm is displayed.	
Manual continuous feed	(1) Jog feed The jog feed speed can be set by the function keys or the numeric keys. (2) Manual rapid traverse If the Rapid traverse key is pressed during jog feed, the machine will move at the rapid traverse speed set by the parameters.	

Item	NU20-20 U7 / NU20-30 U7
Manual pulse wheel feed (When the optional JOG MPG unit is used)	A manual pulse generator can be used to move the machine the corresponding distance. (Option)
Automatic coordinate system settings (when the optional Zero point sensor is used)	The mechanical coordinate system is set automatically when the zero point return operation is performed.
Manual zero point return	The zero point is set manually.
Zero set (DRO function)	The current point is set as the coordinate reference point for an incremental coordinate system. This does not affect program operation.
Preset (DRO function)	The incremental coordinate system is set so that the current position value becomes the setting value entered with the numeric keys. This does not affect program operation.
Coordinate system settings	The workpiece coordinate system is set so that the current position value becomes the setting value entered with the numeric keys.
Program memory capacity	Depends on the capacity of the hard disk or program disk. (Hard disk : Approx.1GB; Program disk : 1.44 MB)
Number of programs	Max. : 1000
Decimal input	Calculator-type decimal input
Input format	Original format Operation is also possible in NC language.
Operation mode	Jog, Manual, Program, MDI
Workpiece edge surface reference coordinate system Workpiece dimension measurement Workpiece centering	The Touch sensor (sold separately) can be used to set, measure, and find the center of the workpiece coordinate system.
Program name search	Programs to run can be selected from the programs stored in the memory or on floppy disk.
Single block	The program can be executed block by one block.
Z-axis lock	No Z-axis lock in NU20-20 U7. Z-axis feed is ignored with Z-axis lock On in NU20-30 U7.
Machine lock	The movement commands can be executed without moving the machine during Machine On.
Start	Automatic operation starts by pressing the Start key, after entering the program name, menu data, and other operating information.
Pause	By pressing the Pause key decelerates each axis and brings them to a stop. By pressing the Start key during the pause, operation will restart where it left off.
All axes stop (Stop switch)	By pressing the Stop switch, all operation and the machine will stop instantly. (The spindle motor does not stop.)

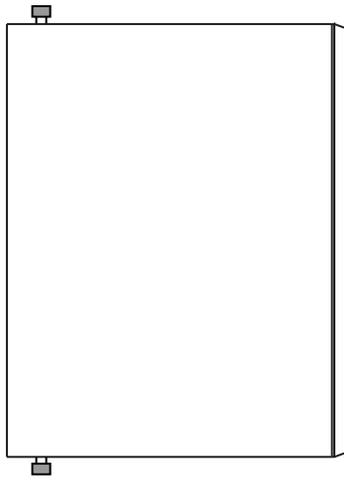| Item	NU20-20 U7 / NU20-30 U7
Remote start-pause switch (External switch connection is required)	During the program and MDI operation, by pressing the Remote start-pause switch cuts off operation and decelerates and stops the machine. By pressing the Remote start-pause switch again, operation will start again where it left off.
Software limit	The machine decelerates and stopped when it passes the preset area. Once the power is turned on, manual rapid traverse is prohibited until this area is set using the system parameters.
Safe zone function	This function allows to determine two types of movement prohibited zone for the inside or outside of a rectangular zone determined by two points.
Sectional error compensation (Optional Zero point sensor is required)	This function allows to compensate for the desired amount of linear error in the desired section. For an axis of 32 points and section of at least 256μm, the compensation amount is ±127μm or less.
Linear error compensation	The linear error compensation is automatically set by measuring the reference length and entering the correct length.
Backlash compensation	This function compensates for backlash of the ball screws.
Self-diagnostic function	This function serves as an aid during troubleshooting and adjustment by displaying the status of the I/O signals and servo error amount.
Positioning (RAPID TO)	Positioning is performed at rapid traverse speed using the RAPID TO command. The machine decelerates and stops at the end point, and an in-position check is performed.
Linear interpolation (LINE)	Linear interpolation is performed at the feed speed designated by F in the LINE command.
Circular interpolation (ARC)	Desired circular interpolation from 0 to 360 degrees CW : Clockwise CCW : Counter-clockwise
Helical cutting	No Helical cutting for NU20-20 U7. Designate the ARC command, Plane selection command and right angle axis to operate the Helical cutting.
Radius (R) designation	Circular interpolation can be designated by the arc end point and the radius R.
Dwell	Time until the program moves to the next block operation, can be delayed using the DWELL command. (units = s)

Item	NU20-20 U7 / NU20-30 U7
Plane selection (PLANE)	No Plane selection (NU20-20 U7). The plane on which Circular interpolation, Tool diameter compensation and Rotate will be operated can be selected using the Plane selection command (NU20-30 U7).
Inch/mm selector	Adding this command to the program allows you to select either inches or mm as the units.
Tool diameter compensation (TOOL OFFSET)	This function offsets the movement so that the tool outer circumference moves along the programmed path.
Tool length compensation (TOOL OFFSET LENGTH)	No Tool length compensation in NU20-20 U7. This function offsets the movement so that the tool tip moves along the programmed path in NU20-30 U7.
Work coordinate system setting	Set the work coordinate systems with the program. Several work coordinate systems are selectable.
Rotate (ROTATE)	The machining commands which follow can be rotated using the Rotate command. The center of rotation and the angle are designated.
Scaling	Enable to reduce/enlarge the programmed machining pattern.
Mirror image	Enable to reverse the programmed machining pattern.
Repeat	Enable to repeat the programmed machining pattern.
Irregular pocket	Enable to do pocket machining of any shape programmed.
Hole fixed cycle	When designated, a message is displayed at the hole positions in NU20-20 U7. Drill and Step cycle are available in NU20-30 U7.
Absolute/Incremental command (ABS/INC)	Absolute settings can be designated by ABS, and incremental settings can be designated by INC.
Program stop (PROGRAM STOP)	When Program stop is run, the program stops after the current block is executed. The program restarts when the Start key is pressed.
Optional stop	When the Optional stop is ON, the program stops after the designated block. The program will be restarted if the start key is pressed.
Program end (PROGRAM END)	When Program end is run, the program stops running and waits for startup.
Subprogram (SUB PROGRAM)	Up to four nested programs can be used.
Spindle command (SPINDLE COMMAND)	The four figures which follow the S in this command designate a spindle. This command stops automatic operation and displays the rotation number of the designated spindle.
Spindle FWD/REV/STOP (Available with optional Spindle control unit NZ31 used)	Enable to designate Spindle FWD or REV rotation and stop by program. NU20-30 U7 only.

128

Item	NU20-20 U7 / NU20-30 U7
Tool command	The two figures which follow the T in this command designate a tool number. This command stops automatic operation and displays the designated tool number.
Program operation	The following operations can be performed : Renaming, copying, and deletion of programs, reading programs from floppy disks, and saving programs to floppy disk.
Menu programming Menu editing (MDI/MACHINE CYCLE)	Programs can be created, edited, and automatically performed by referring to the diagram information. 1. Drilling cycles : Single hole drill cycle, Line drill cycle, Random hole drill cycle, Bolt circle drill cycle, Partial bolt circle drill cycle, Frame drill cycle, Grid pattern drilling cycle 2. Face milling cycles : Upper and lower 3. Pocket milling cycles : Circle, Rectangular, Corner with radius, Corner with inverted radius, Irregular 4. Frame mill cycles : Circular, Rectangular, Rectangular with corner radius and rectangular with inverted corner radius 5. Island mill cycles : Circle, Rectangular with corner radius, Rectangular with inverted corner radius, Rectangular with square corner, Rectangular with corner chamfer 6. Ring : Circle corner with radius 7. Positioning 8. Linear cutting 9. Circular cutting
Coordinate calculations	Calculate the coordinate of the intersection over 3 events. 1. Intersection of two lines 2. Intersection of a line and an arc 3. Tangent point of a line and an arc 4. Intersection of two arcs 5. Tangent point of two arcs 6. R insertion (line and line, line and arc, arc and line)
Graphic representation in editing	Display the programmed image by graphic.
Operating simulation	The created program operation is displayed graphically.
Machining time display	Display the required machining time when the program is checked.

15-3.　Accessories

Item/Product ID	NU20-20 U7 / NU20-30 U7
NU20 instruction manual	1
Power cable	1
STOP switch	1
Windows®95 disks and documentation	1 set
PCI IDE driver disk and instruction manual	1 set
MPC106A driver disk and instruction manual	1 set
MSC-256 instruction manual	1
MTB-001 wiring board instruction manual	1
NC control program disk	1 set

26.6/1.05"

130.6/5.14"

170.3/6.7"

166.0/6.5"

362.0/14.3"

266.0/10.47"

483.0/19"

463.0/18.23"

SONY

57.15/2.25"

57.15/2.25"

76.2/3"

94.9/3.74"

160.0/6.3"

170.3/6.7"

28.7/1.13"

130.6/5.14"

unit:mm/inch

131