

INSTRUCTION

DMNC

A

SERIES

POWER

SUPPLY



DIMON BEIJING CNC TECHNOLOGY CO., LTD.

V01.1.

PRAT 1 SYSTEM INTRODUCTION.....	1-1
CHAPTER 1 SYSTEM CONFIGURATION.....	1-2
1.1 Introduction.....	1-2
1.2 Features.....	1-2
1.3 Notice.....	1-2
1.4 System Configuration.....	1-3
1.5 Component Name and Function.....	1-4
1.6 Mode Configuration.....	1-6
1.7 The mode Scene Layout.....	1-9
1.8 Accessories.....	1-10
CHAPTER 2 OPERATION.....	2-1
2.1 Introduction.....	2-1
2.2 Start of the CNC supply unit.....	2-1
2.3 End of the CNC supply unit.....	2-2
2.4 Emergency stop of Z power.....	2-2
2.5 The Basic Keys of the CNC supply unit and Its Operation.....	2-3
2.6 Operation Procedure of the CNC supply unit.....	2-5
2.7 JOG Operation (Remote Control Box Operation)	2-6
2.8 ENABLING SWITCH Operation.....	2-8
2-9Tool Holder Setting.....	2-9
PART 2 PREPARATION FOR OPERATION.....	3-1
CHAPTER 3 COORINATE SET.....	3-2
3.1 Coordinate System Change.....	3-2
3.2 Coordinate Value Set.....	3-3
3.3 [FUNC]&[COND].....	3-5
CHAPTER 4 MOVE.....	4-1
4.1 Move.....	4-1
4.2 Halfway Move.....	4-4
4.3 Limit Move.....	4-6
Chapter 5 ALIGNMENT.....	5-1
5.1 Approach Face Alignment.....	5-1
5.1.1 Approach face Alignment scene.....	5-1
5.1.2 Approach face detail alignment scene.....	5-2
5.2 Column Center Alignment.....	5-4
5.2.1 Column center alignment screen.....	5-4
5.2.2 Column center detail alignment.....	5-5

5.3 Corner Alignment.....	5-9
5.3.1 Comer alignment scene.....	5-9
5.3.2 Comer detail alignment scene.....	5-10
5.4 Hole Center Alignment.....	5-12
5.4.1 Hole center alignment scene.....	5-12
5.4.2 Hole center detail alignment scene.....	5-13
5.5 Other Alignment.....	5-16
5.5.1 Random 3 points alignment.....	5-16
5.5.1.1 Random 3 points manual alignment.....	5-16
5.5.1.2 Random 3 points auto alignment.....	5-17
5.5.2 Discharge alignment.....	5-19
PART 3 CUT.....	6-1
CHAPTER 6 ASSIST CUTTING.....	6-2
6.1 Assist cutting.....	6-2
6.2 Single Cutting.....	6-18
CHAPTER 7 MANUAL CUTTING.....	7-1
7.1 Line cutting.....	7-1
7.1.1 Line cutting scene.....	7-1
7.1.2 Line cutting detail scene.....	7-4
7.1.3 Line cutting code.....	7-7
7.2 Arc Cutting.....	7-8
7.2.1 Arc cutting scene.....	7-8
7.2.2 Arc cutting detail scene.....	7-11
7.2.3 Arc cutting code.....	7-13
7.3 Thread Cutting (Z,U axis).....	7-14
CHAPTER 8 MULTIPLE CUTTING.....	8.1
8.1 Random Multiple cutting.....	8-1
8.2 Lattice 1 Multiple Cutting.....	8-5
8.3 Lattice 2 Multiple Cutting.....	8-9
8.4 Circle Multiple Cutting.....	8-13
CHAPTER 9 USER CUTTING.....	9-1
9.1 HD User Cutting.....	9-1
9.2 FD User Cutting.....	9-3
CHAPTER10 EDIT.....	10-1
10.1 Edit Screen Configuration.....	10-1
10.2 Edit function.....	10-2

PART 4 UTILITY.....	10-1
CHAPTER 11 PASSWORD.....	11-1
11.1 Password Screen Configuration.....	11-2
CHAPTER 12 SET.....	12-1
12.1 Introduction.....	12-1
12.2 FLAG SET.....	12-3
12.2.1 FLAG SET FLAG SET.....	12-4
12.2.2 FLAG SET MACHINE SPEC SET.....	12-8
12.2.3 FLAG SET. NC SET.....	12-9
12.2.4 FLAG SET DISCHARGE SPEC SET.....	12-12
12.2.5 FLAG SET MOTOR SPEC SET.....	12-15
12.2.6 FLAG SET AXIS SPEC SET.....	12-22
12.3 CHECK.....	12-25
12.3.1 IN/OUT (A0000H).....	12-26
12.3.2 IN/OUT (A0060H).....	12-27
12.3.3 IN/OUT (A00A0H).....	12-28
12.3.4 IN/OUT (A00E0H).....	12-29
12.3.5 IN/OUT (A0130H).....	12-30
CHAPTER 13 File Operation.....	13-2
13.1 File Operation.....	13-2
13.2 File Operation Method.....	13-3
CHAPTER 14 LIST OF ERROR/HALT MESSAGES AND	
COMMENTS.....	14-1
14.1 Introduction.....	14-1
14.2 Message Display.....	14-2
14.3 Causes of Error/Halt Messages and Their Remedies.....	14-3
14.4 Causes of Comments and Their Remedies.....	14-20

Preface

First, thank you for your purchase and use of CNC supply unit for EDM machine of Beijing Institute of Electro-Machining and DIMON BEIJING CNC TECHNOLOGY CO., LTD.

CNC supply unit is the crystallization of the latest scientific research achievements of our technical staff. It has excellent machining performance, reliability and convenient operation to ensure the perfect realization of your processing design.

This manual describes the basic use of CNC digital control power supply, working order, matters need attention. Please read carefully the operator and his supervisor. Any adverse consequences arising from the violation of the instructions during the operation, the company assumes no responsibility.

With the continuous research and development of CNC series power supply, the contents of this manual may be further improved and added.

If this specification can be used to make full use of your CNC series power supply, we will be honored.

PART 1 SYSTEM INTRODUCTION

This part will introduce the system introduction, the name of each part, uses and basic operation methods of the CNC supply unit.

Chapter 1 System Configuration

Chapter 2 Basic Operation

Chapter 1 SYSTEM CONFIGURATION

1.1 introduction

This chapter describes the system of the A power supply unit designation of its components and their functions.

1.2 Features

1. To use the "touch screen" advanced technology to replace the complicated keyboard operation. All processes from alignment to cutting can be done conveniently by "screen" operation.
2. It provides excellent user interface, dialogue ways. The users can operate the CNC supply unit correctly even though they don't understand the NC program.
3. According to the assist cutting, user need only input cutting demands prompted by screen of each mode, then the system can create the NC codes automatically from rough cutting to finish cutting, and do cutting automatically, and finish the process with optimum.

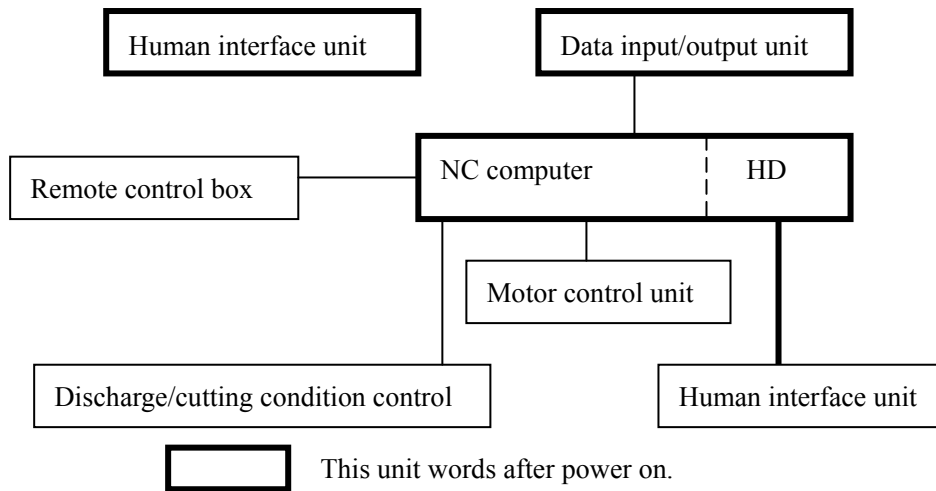
1.3 Notice

The A power supply unit need be note as shown below:

- 1) Only 4 axes of X,Y, Z, U can be set.
- 2) PATTEN CUTTING is possible, but the "offset" fiction is not available the codes of G40, G41, G42 can not be used.
- 3) The ruction of changing screen and modifying the parameter data during the cutting operation is not available.
This fiction can be used after press the "halt".
- 4) The SINGLE RUN ruction can be turned ON/OFF in the SET mode.
- 5) NC codes can be executed only in user programs.
- 6) When using the "LORAN PATTERN",SERVO 1 and retraction along the pattern can not be used. Retracting to the center only by SERVO 2.
- 7) The editing instructing can only be used in the user's program. In addition, the U axis Mechanism(Rotation indexing axis)and ATC (Automatic tool changer) are optional accessories. The operation concerning the U axis and ATC are possible only after selecting the two optional accessories.

1.4 System Configuration

The CNC supply unit consists of various units. The NC computer is the center of the CNC supply unit.



Diagram

(1) Remote control box

This box carries JOG, OFF, HALT, ACK and other keys which are to be frequently used during program operation process.

(2) Human interface unit

This unit is made up of a display, machine operation control key and others, and it transmits operator's command to the NC unit and informs the operator of the EDM operating status.

(3) Data input/output unit

This unit is composed of a FD driver, PTR (paper tape reader) and serial port (RS-232C) for input/output of NC programs for the CNC supply unit.

(4) NC computer

The NC unit constitutes brains of the CNC supply unit severing 'not only to analyze the input data, but also to command, monitor, control and manage the whole of EDM operations.

(5) Discharge/cutting condition control unit

This unit provides the optimal discharge waveform and cutting conditions to fit in with the cutting status

(6) Motor control unit

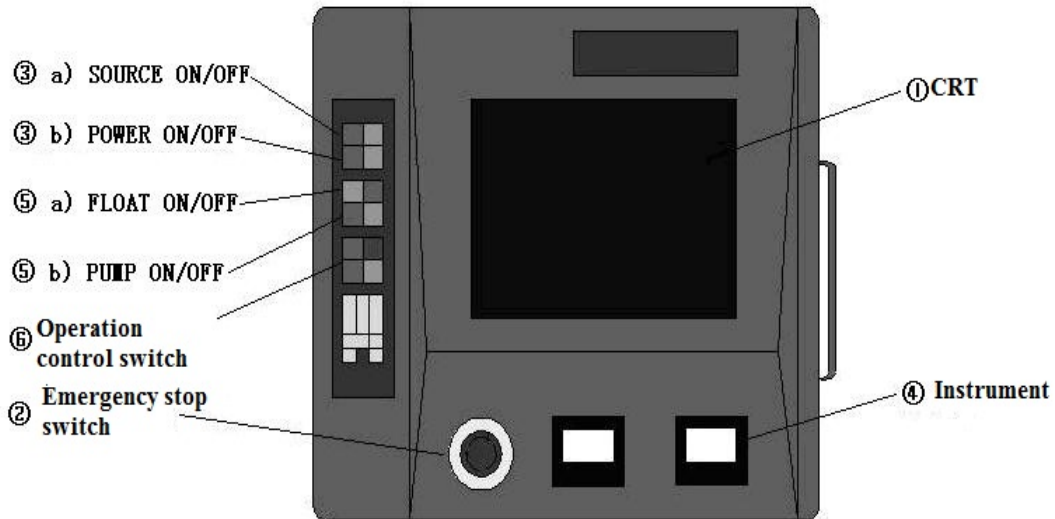
This unit controls the motor according to the command from the NC unit to perform high-speed and high-accuracy operation such as electrode alignment

(7) Machine input/output unit

This unit transmits commands from NC unit to machine and informs the window of NC unit about the machine status.

1.5 Component Name and Function

The outline of the CNC supply unit, the name of its components and their individual functions are shown in following diagram.



① CRT

This 14 inch CRT display is used not only to provide various information on power and Machine operation.

② EMERGENCY STOP switch

This switch is used to turn OFF the CNC supply unit both to the machine and power unit at the same time. Never activate this switch except in case of emergency

③ SOURCE switch

a) This switch is used to turn ON/OFF the NC power.

(1) Never activate the [SOURCE OFF] key while the CNC supply unit is ON. Be sure to turn the CNC supply unit OFF before activating the [SOURCE OFF] key.

(2) When repeating the ON/OFF operation of the SOURCE switch, which causes a burden.

b) POWER switch

This switch is used to turn ON/OFF the CNC supply unit of the machine. This switch can be turned ON after the starting of the NC unit completed.

When repeating the ON/OFF operation of the POWER switch, which cause a burden on the CNC supply unit and machine, be sure to leave at least two minutes between the repetitions.

④ Instrument

Ammeter is on the left. Voltage is on the right.

⑤ a) Float on/off

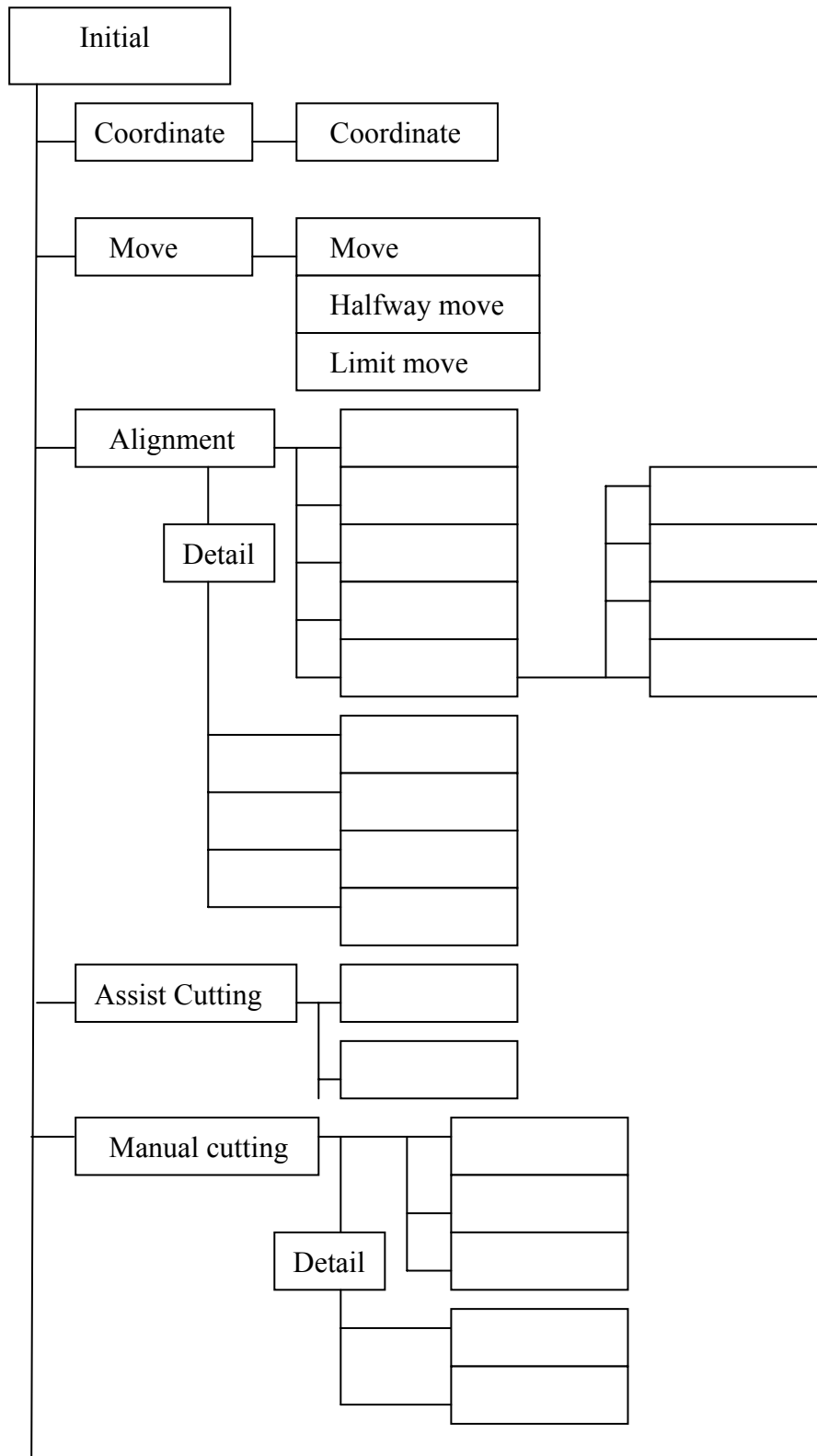
b) Pump on/off

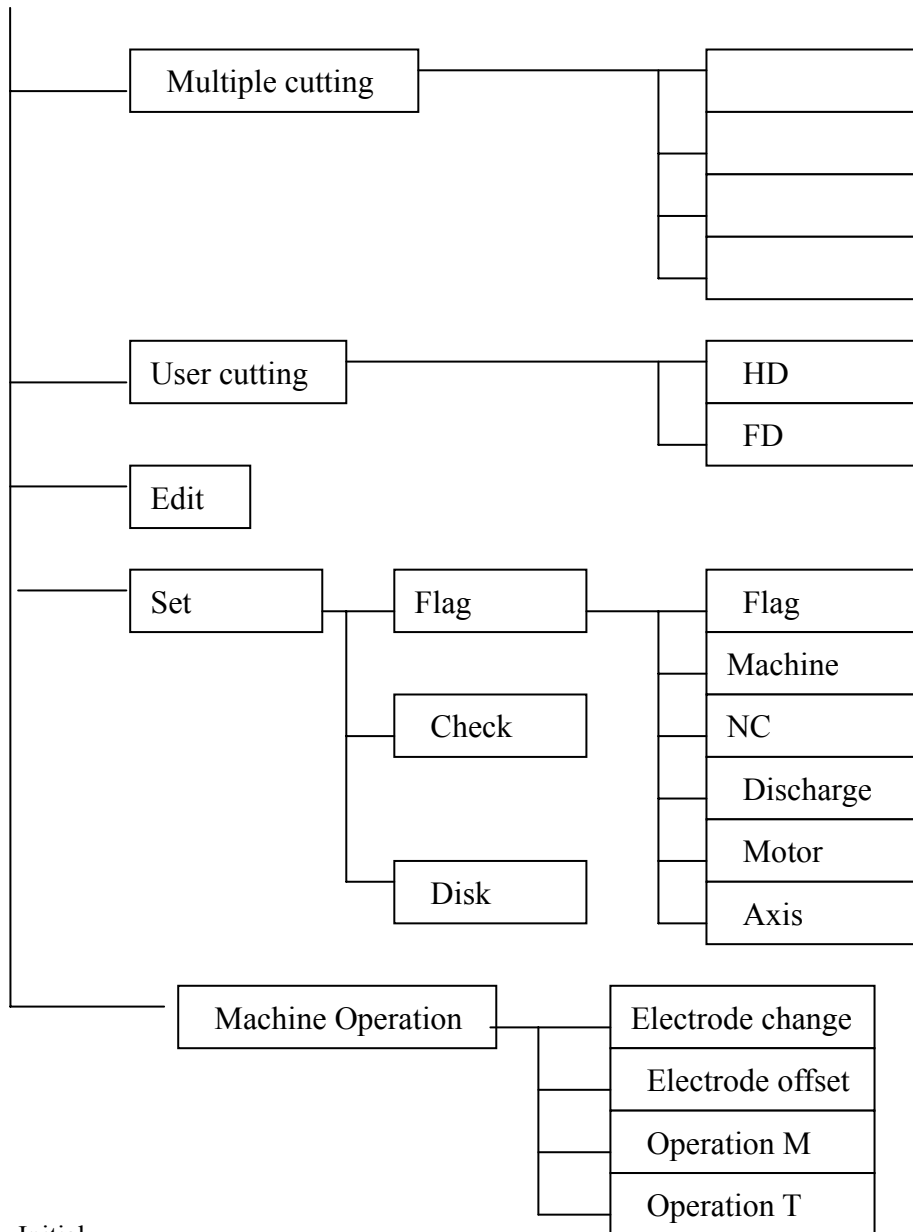
⑥ Operation control switch

This switch insists of the [ENT], [ACK], [HALT], [OFF] key.

1.6 Mode Configuration

The operation modes of the CNC supply unit are configured as shown below with their division into two major groups: one is [Z modes] categorized according to the type of operation and other is [Detail] mode provided by sub-dividing each Z mode for its detail setting.





- Initial
When the CNC supply unit is turned ON, it is necessary to move the X-, Y- and Z- axis to their machine limits, respectively, to set the coordinate origin point.
- Coordinate set
This mode is used to change the coordinate system and the coordinate values of the axes in the current system.
- Move
This mode is used to set the position to which move the electrode and move it to the set position according to the following three sub modes: Move, Halfway move and Limit move.

- **Alignment**
This mode is used to measure the electrode and work piece positions for alignment according to the following five sub modes: Approach face, Column center, Corner, hole center and other alignment.
- **Assist cutting**
This mode is used to make the CNC supply unit create automatically the optimal cutting conditions and cutting mode according to the input information such as the cutting depth and work piece material. There are two cutting sub modes: assist cutting and single cutting.
- **Manual cutting**
This mode is used to do manual cutting according to manual input information such as the number of cutting and cutting condition. It consists of line cutting, arc cutting, thread cutting, spiral cutting and PIKA10 cutting.
- **Multiple cutting**
This mode is used to allow cutting of more than one hole by the cutting position and the number of hole to be cut. There are four sub modes multiple position: Random, Lattice (two patterns) and circle.
- **User cutting**
This mode is used to perform a NC program created by user outside the CNC supply unit.
- **Program**
This mode is used to combine the operation modes from Alignment to Cutting to execute them as a series of operations or to carry out more complex multiple cutting.
- **Set**
 - 1) **Flag**
This mode provides the setting methods of power and machine.
 - 2) **Check**
This mode is used to check the input/output state of power and machine.
 - 3) **Disk**
This mode provides the disk operation.
- **Machine operation**
This mode is used to perform electrode change, electrode offset saving

1.7 Scene Layout

There are "Z scene" and "Z detail scene" displayed by Z screen. The user can input more data and cutting demands in that scene. The division of two scene contents is similar, and it can be shown as Fig. 1.7.1.

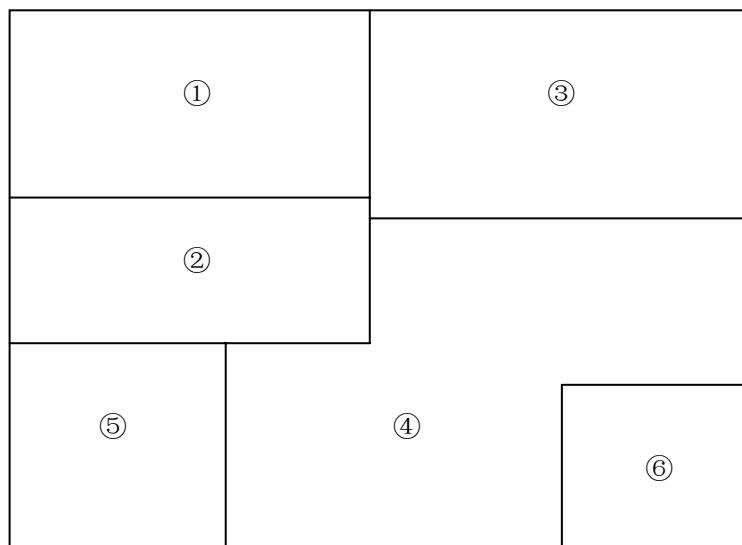
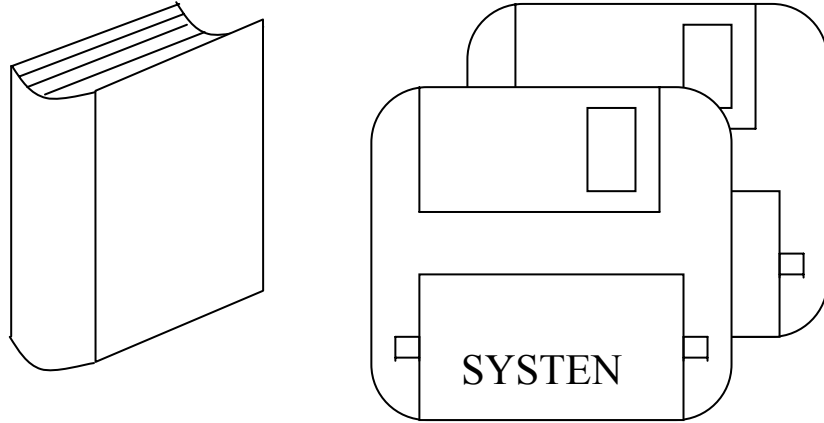


Fig1.7.1

- ① Coordinate value display area
Displays the current coordinate values of the X,Y,Z and U axes
- ② Message display area
Displays a message of the machine/NC unit operating state.
- ③ Mode button key display area
Displays the selection change key between mode and sub mode
- ④ Data input area
Input s the cutting data and demands.
- ⑤ Help description scene area
Displays the help scene and text description of each operation.
- ⑥ Simulation key area
Displays the input button key of simulation keyboard such as data key.

1.8 Accessories

The CNC supply unit provides following accessories. It is only a filling of other mode such as the CNC supply unit and machine.



1. Operation description
It is this book you are reading.
2. System floppy disk to be used to installation.

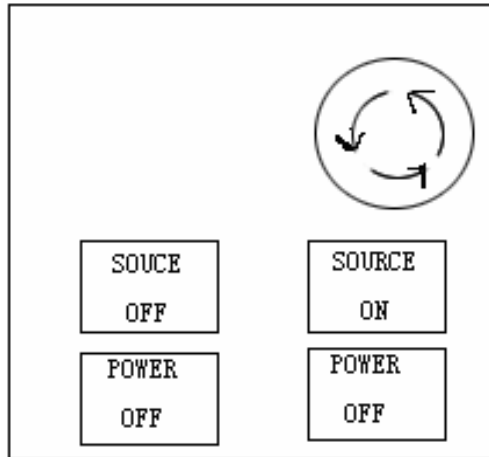
Chapter 2 OPERATION

2.1 Introduction

This chapter describes the procedures for starting and ending the A power supply unit.

2.2 Start the CNC supply unit

The position of the power switch is shown as following.



The starting procedure of the CNC supply unit is as follows:

- 1、 Press the [SOURCE ON] key at the upper right of the device as shown the diagram. In a little while after the CNC supply unit is turned ON, the scene will display the starting scene. After completing the display of the starting scene, the screen will prompt "Press the POWER switch.

(1) The POWER switch cannot be turned ON until the above prompt appears on the screen.

NOTE!

(2) If the starting screen displays "Press <F1> to resume', please press the [SOURCE ON] again after fuming off the CNC supply unit, and restart again.

(3) Never press the [SOURCE ON] key while the screen is displaying the starting scene.

(4) When repeating the ON/OFF operation of the power switch, leave at least 30 seconds between the repetitions.

- 2、 Press the [POWER ON] key

Press the [POWER ON] key and the machine .can be operated.

NOTE! Press leave the ON/OFF POWER switch at least two minuets.

- 3、 Perform the machine origin point setting operation

Use the keys provided at the JOG or use the mode of the [Move] to set the limits movement. To cause all the axes to move to their respective set limits for the machine origin point setting

The JOG key can also be used to move all the axes to their (-) or (+) limits for the machine origin point setting.

NOTE! When the origin point setting, make sure that the tool and electrode cause noninterference.
The starting of the CNC supply unit is completed with help of above necessary operations.

The starting of the CNC supply unit is completed with help of above necessary operation.

2.3 End of the CNC supply unit

The CNC supply unit will be ended as following procedures.

- 1) Make sure that neither the CNC supply unit nor machine is operating. If the CNC supply unit or machine is operating, stop its operation.
- 2) Press the [POWER OFF] key at the left bottom of display. This causes the CNC supply unit to the machine to be turned OFF.
- 3) Press [SOURCE OFF] key. This causes the CNC supply unit to be turned OFF.
 - (1) Perform the above procedure when ending the CNC supply unit.
 - (2) Turn off the CNC supply unit when the EDM machine is not in use.

2.4 Emergency stop of Z power

When any emergency occurs, requiring urgent shutdown of the machine, press the [EMERGENCY STOP] switch at the upper left of the screen. This causes the CNC supply unit both to the machine and the power unit to be turned OFF at the same time.

- NOTE!
- (1) Once the [EMERGENCY STOP] switch is activated, it will hold its activated state. The switch can be released from this state by turning it clockwise.
 - (2) Never use the [EMERGENCY STOP] switch except in case of emergency

2.5 The Basic Keys of the CNC supply unit and Its Operation

The user interface of the CNC supply unit is very friendly. The interchange between user and machine can complete very conveniently in "screen", by using the "touch screen" technology to replace "keyboard" to the button of "screen".

There are four types of the "touch key" in the Z screen.

1) Mode button

Using finger (or other tools) to press "touch" the button of mode and sub mode, the button is in "selected color" state, at this time, the scene is changed to selected mode (and sub mode). ;

The mode buttons also include:

- [Detail] button: Press this button, the scene will change to the detail scene of current scene.
- [FILE] button: Press this button, the scene will change to file operation scene.

2) Data input button

This is a input, select cutting parameter button. Press (or "touch" the button, make it in concave state) input item button, and then press the digit key, sign key of simulation keyboard can input the cutting demands and parameters.

3) Simulation keyboard button

The "simulation keyboard button" in the screen is shown as Fig. 2.5.1.

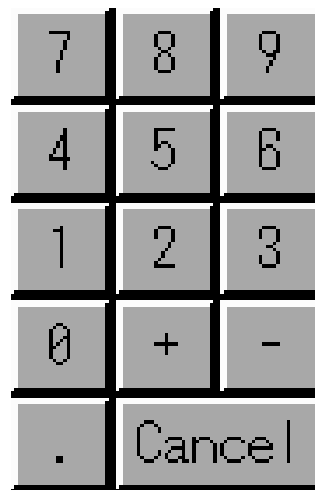


Fig 2.5.1

- Digit key
There are [0], [1], [2], [3], [4], [5], [6], [7], [8], [9] keys to be used for selecting and inputting cutting parameters.
- Sign key
There are [+],[-]keys to be used for setting+/- of cutting parameters and moving direction.

- [Cancel] key

This key is used to cancel the input of last time.

4)Function button

In some modes ,the function buttons will appear at bottom line of scene .They are used according to their name and meaning.

Besides this ,there keys in operating board of the CNC supply unit as.

- [OFF] KEY

This key is used to stop the action of machine and NC. The machine can not carry out.

- [ACK] key

After [OFF] key is pressed or an error occurs, pressing this key releases from previous state.

- [HALT] key

Pause the operation of machine and NC. After pressing the [ENT] key, the operation will start again.

- [ENT] key

This key is used to start operation of machine and NC. If you want to work again after pressing the [HALT] key, use this key.

NOTE! Make sure the state of machine and environment when the machine is in operation and discharge after pressing[ENT]key.

2.6 Operation Procedure of the CNC supply unit

The A SERIES procedure from craying preparation to operation is roughly shown in Fig.2.6.1. This section will describe the axis movement, operation of work piece set, coordinate changeand coordinate set.

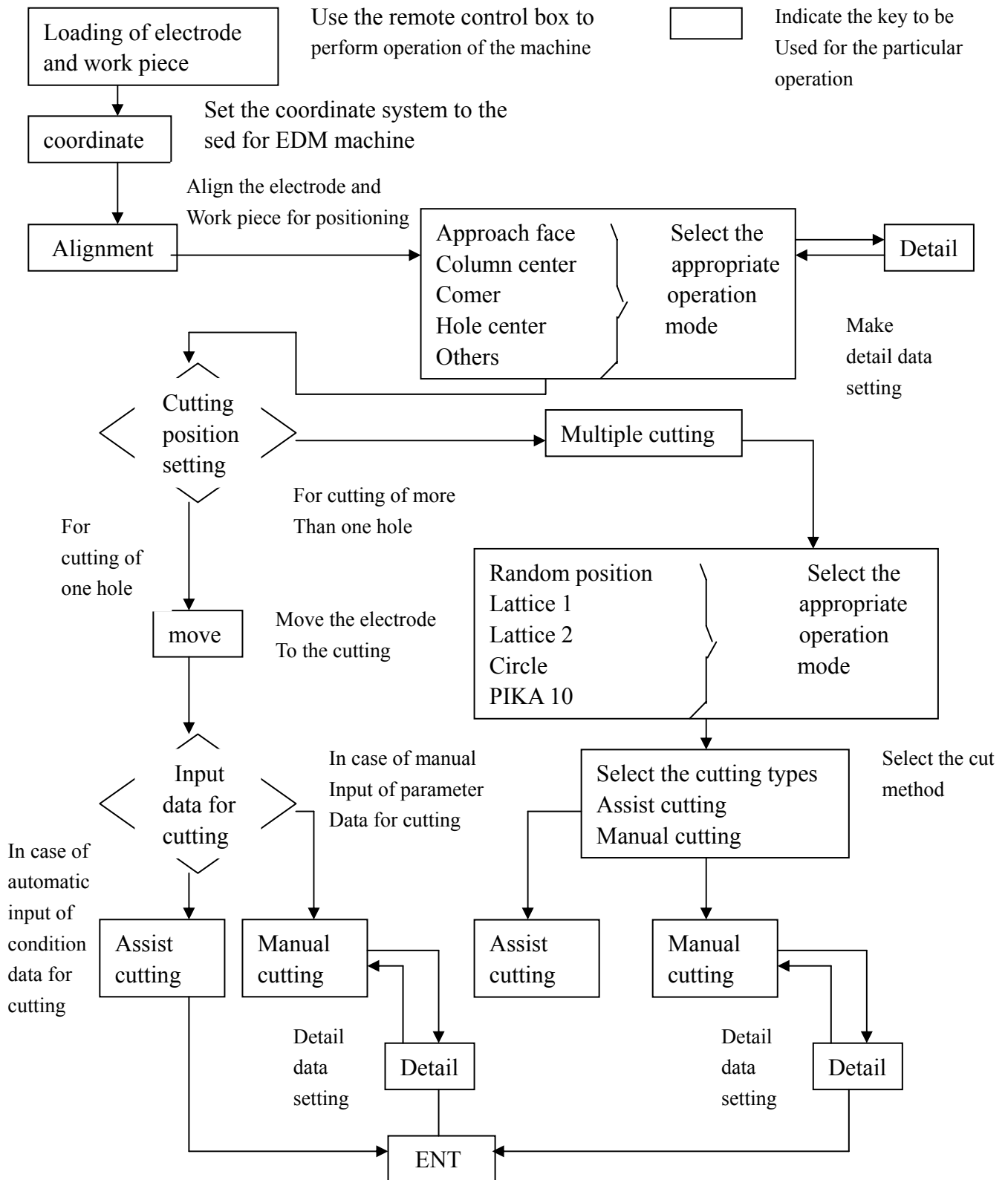


Fig.2.6.1

2.7 JOG Operation (Remote Control Box Operation)

The remote control box has necessary switches which are used for cutting preparation of the machine. The diagram is shown as Fig.2.7.1.

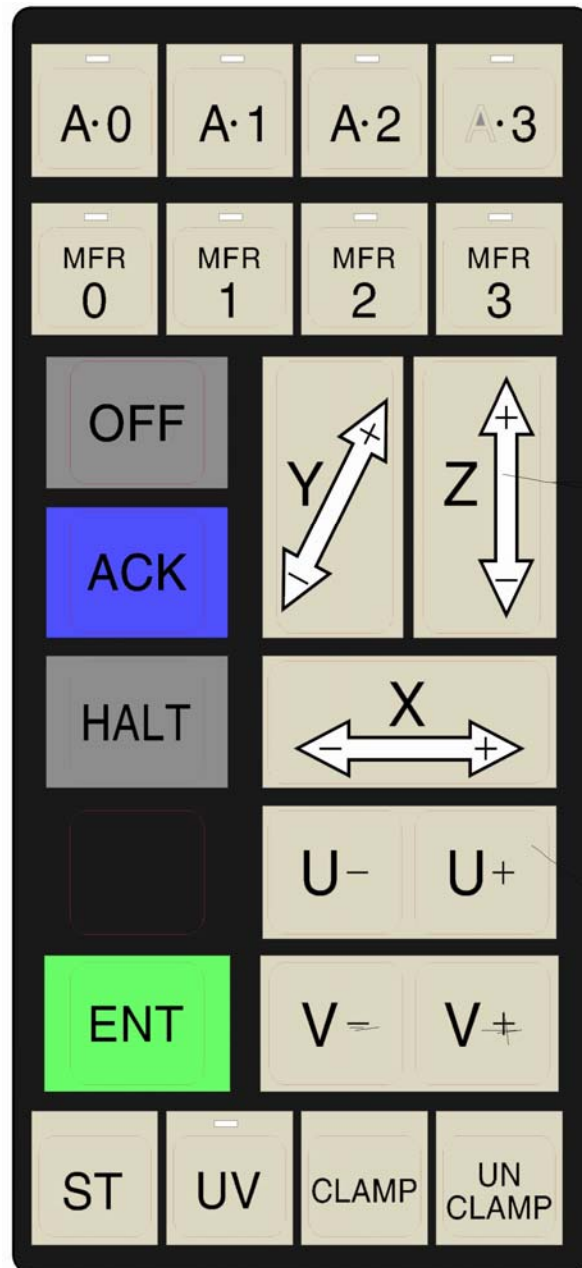


Fig .2.7.1

1、 [JOG] key

This key includes [X-][Y-], [Y+], [Z-], [Z+], [U-], [U+] keys. When pressing this key, it can specified axis to be moved in the “-” or “+” direction.

[Operation example]

Press the [JOG] key [X-], the X axis moves in the '-' direction. All this time:

- 1) The current coordinate value of the axis displayed at the upper left of the screen will change with its movement.
- 2) The movement speed can be selected in four steps by operating [M.F.R] knob.
- 3) If the machine moves beyond its movable range. the limit switch will be activated to stop its movement automatically.
- 4) At this time mentioned above, none of the axes can be operated. To release the machine from this state, press the [ACK] key.
 - [M.F.R] key (JOG speed selection)
The speed at which JOG key moves can be Selected in four steps.

Speed No.	Operation	Remarks
0	high-speed movement	1 pulse will be produced pressing each time the
1	Medium-speed movement	JOG key. (move μm for general, and $0.5\mu\text{m}$
2	Low-speed movement	for-specified machine) Continued pressing of
3	Inching movement	This key causes the low-speed movement continuously

- [ENT] key
To start machine and NC operation.
- [OFF] key
Press this key during the axis operation or machine operation executed by pressing the [ENT] key causes this operation to be Stopped at this time, when the key is pressed sounding the buzzer to. -indicate -the-pause -with _current , display message on the screen.” Press [OFF] key to stop. Please Press [ACK] key". During pause process, a new operation can not be done.
- [HALT] key
Press this key during the axis operation or machine operation executed by pressing the [ENT] key causes this operation to be at this time, when the key is pressed, sounding the buzzer to indicate the pause with current display message on the screen. Temporarily stopped. Please to press [ENT] key. Press [OFF] key to stop" During pause process, a new operation can not be done. The axis move can be clone by pressing the [JOG] key of the remote control BOX.
- [ACK] key
When the machine occurs an error or pressing[OFF]key, pressing this key cause the machine to be released from the stopped state.
- [ST]key
Pressing the [JOG] key with the [ST] key held down for axis movement causes the axis to move with the "ST" function ignored. (In general, the axis operation will unconditionally stop when the electrode comes into contact with the work piece during the axis movement. It is called as "ST" function.)

- [Unclamp]key(for tool holder unloading)

This key is used for manual unloading of the tool holder set in the machine head. (For R axis type machine, the IT. CPOINT] key is operated, for tool change position shifting before the above operation

NOTE! Pressing this key will cause the tool holder to be dropped from the machine head at this moment.

- [Clamp] key (for tool holder loading)

This key is used for manual unloading of the tool holder set in the machine head.

NOTE! When the key is pressing manually push the tool holder to position that you want to set

2.8 ENABLING SWITCH Operation

Enabling switch is a safe switch For C axis make so that are useful constant rotation (20Rpm).

Enabling switch is useful to do vertically of an electrode.

An operative method is similar as JOG Operation.

2.9 Tool Holder Setting

The tool holder installation can be done by two methods depending on the machine type.

● Tool with 3 axes

The steps 1 to 4 for tool holder unloading and steps 5~6 for tool holder loading. If tool holder in the machine head will be unloading to loading state, steps 1 to 4 are omitted.

- 1) Press the [T CPOINT] key. This causes the U axis moves to the change work piece position and then stops there.
- 2) The lamp of the [T. CPOINT] key is ON. This indicates that the machine is ready for tool holder unloading.
- 3) Hold the tool holder in machine head with hands securely.
- 4) With the tool holder held with one hand, press the [Unclamp] key with the other hand at same time. This causes the tool holder to be detached from the machine head.

NOTE! Take full caution against drop of the holder.

- 5) Take the tool holder to be attached on the machine head and push the holder to its loading position by hand.
- 6) Press the [Clamp] key, the tool holder will be attached to the machine head.

● Toll without 3 axes

Following steps 1 to 2 for unloading tool holder, and steps 3 to 4 for loading tool holder. If tool holder in the machine head will be unloading to loading state, the steps omitted.

- 1) Hold the tool holder on the machine head with one hand securely.
- 2) Press the [ACK] key while pressing the [Unclamp] key. The tool holder will detached from the machine head.
- 3) Take the tool holder to be attached on machine head and push the holder to its loading position by hand
- 4) Press the [Clamp] key. The tool holder will be attached on the machine head.

PART 2 PREPARATION FOR OPERATION

In general, some steps such as coordinate setting movement will be done before using the Z cutting. This part will introduce the operation methods of some steps.

This part consists of following chapters:

Chapter 3 Coordinate Set

Chapter 4 Move

Chapter 5 Alignment

Chapter 3 Coordinate Set

Coordinate setting refers to the operation of set the current coordinate system and its coordinate value. This chapter introduces the operation methods of coordinate setting. They are:

- Coordinate system change
- Coordinate Value Set

3.1 Coordinate System Change

The CNC supply unit system provider 6 coordinate systems to satisfy various types cutting (multiple cutting ,program cutting according to user-created NC program).pressing the mode button [COORDINATE SET]in the screen, the screen will display coordinate setting scene as Fig 3.1.1.

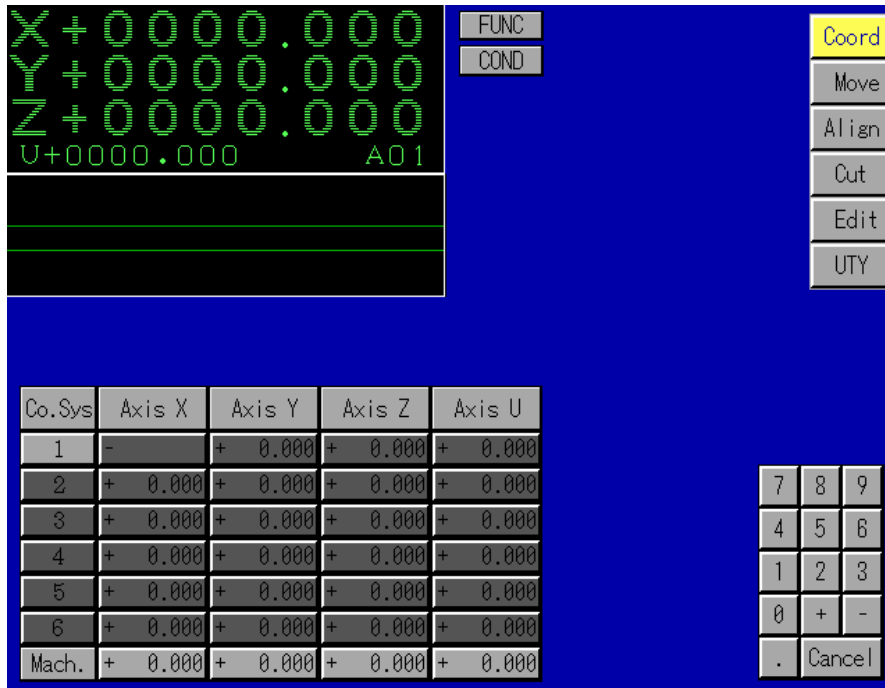


Fig 3.1.1

Press the numbers form[1] to [6] under the [Coordinate System] button in the scene correspond the 6 coordinate systems .Select a coordinate system for current cutting form buttons[1] to [6] by pressing one of them ,the coordinate system will be change. The pressed coordinate systems current coordinate system. Besides the coordinate system change , the coordinate value set can also be done in coordinate set screen.

3.2 Coordinate Value Set

Coordinate value setting refer to the operation of changing the coordinate values of each axis in current coordinate system to the specified values. The Coordinate value setting scene is as Fig 3.2.1

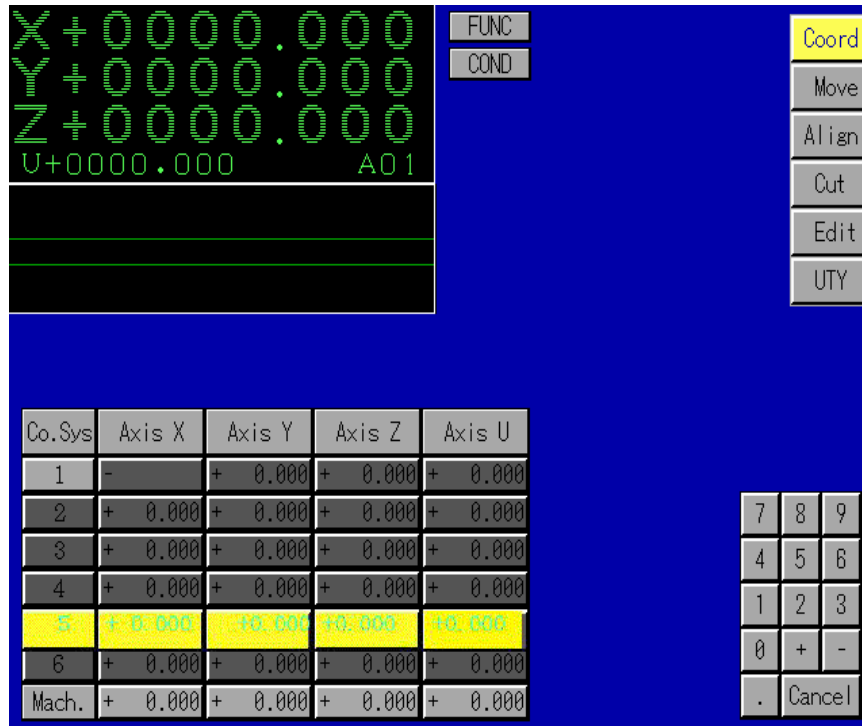


Fig 3.2.1

There are two coordinate value input methods as follows:

- Input the X, Y, Z and U axes coordinates of the specified coordinate system at same time.
Select a coordinate system from coordinate systems [1] to [6] by pressing the button, and then input the coordinate values. At this time, the coordinates of four axes change to specified values at same time.
- Input a coordinate of specified axis of a specified coordinate system.
Press the digit input buttons in which the horizontal direction represents the designate coordinate system and the vertical direction represents the specified axis, and then input the data. Press the [CANCEL] button in the scene can cancel the error data when error occurs.

NOTE! The [Machine] line in the scene displays the value of current machine coordinate system. This line can be input.

[Example of coordinate set operation]

Set the X, Y, Z, U coordinate values of coordinate system 3 to 10.500, and the Y coordinate value of coordinate system 5 to 5.500.

Steps:

- 1) Press the [3] key under the [Co. Sys], at this time, all button in this line of coordinate [3] are in “concave” state ,then press digit buttons [1],[0],[.],[5],[0],[0].

2) Press buttons of [5] line and [Axis Y] column, (At this time, the corresponding buttons are in "concave" state) then press digit buttons [5], [.), [5], [0], [0]. The system completes set automatically after pressing [ENT] key.

3.3 [FUNC] & [COND]

There are two sub mode buttons named [FUNC] and [COND] respectively as shown in Fig.3.3.2. User can also find them in other mode screen.

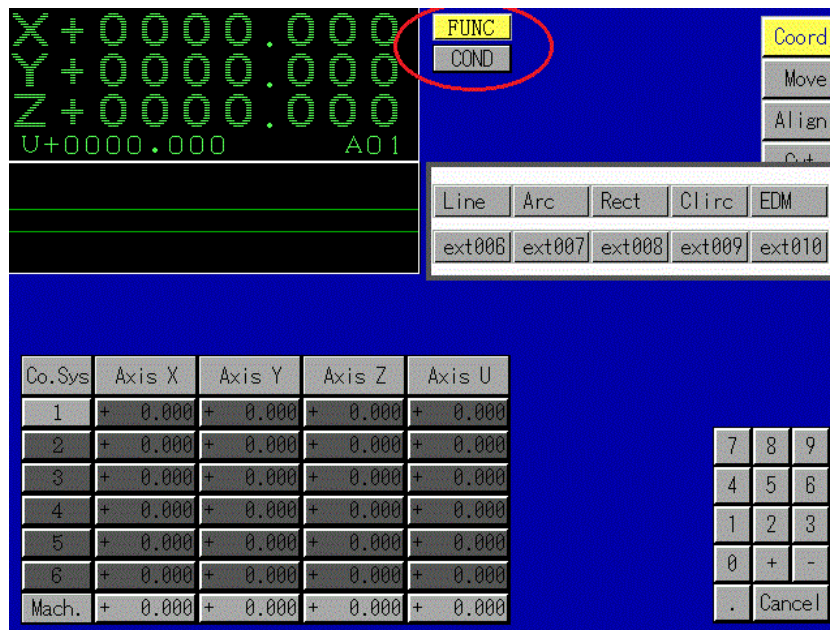


Fig.3.3.1

- [FUNC]

When [FUNC] button is pressed, a sub window which contains 10 buttons will be displayed, as shown in Fig.3.3.2. Each one of the buttons is a shortcut key for NC operation. User can set the shortcut buttons in [UTY]->[SETTING]. When the [UTY], [SETTING] and [Chg.PW] are pressed orderly, mode screen like Fig.3.3.2 will be displayed.

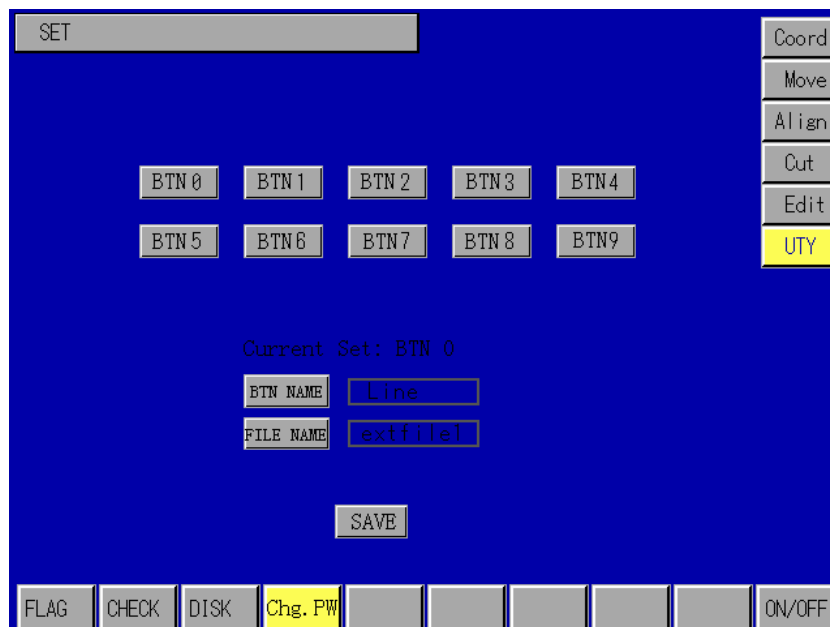


Fig.3.3.2

There are 10 buttons in [Chg.PW] mode screen. User can select any one of them by pressing it, then write or rewrite its “BTN NAME” and specify the NC file which is related to it. After set, the shortcut buttons which are set in [Chg.PW] mode can be seen in [FUNC] mode.

- [COND]

When [COND] mode button is pressed, mode screen like Fig.3.3.3 will be displayed.

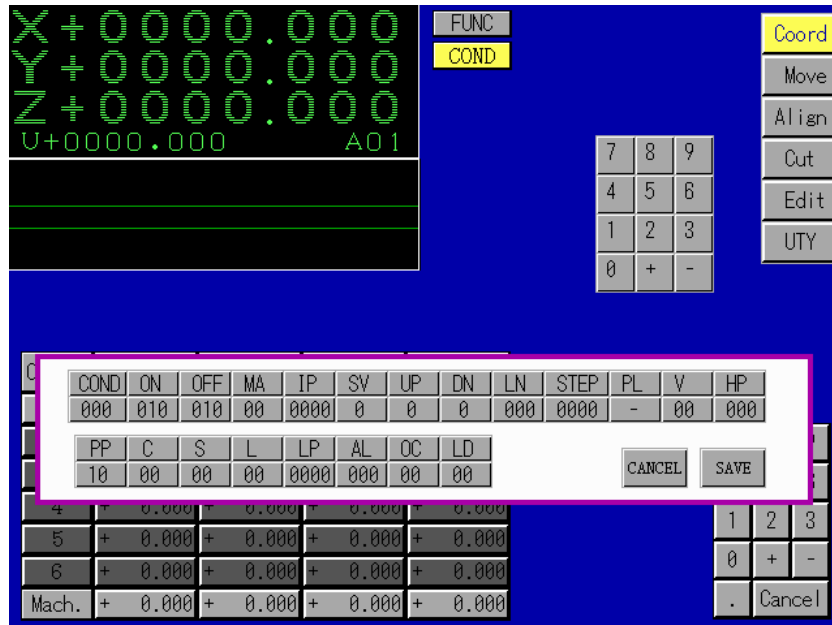


Fig.3.3.3

The sub window in Fig.3.3.3 displays global processing parameter list. The first element named [COND] stands for No. of the list. User can modify No. of list within 000~999. Every element in the list expect [COND] also can be modified by user. After modification, user can save the list or cancel operation.

Chapter 4 Move

"Move" refers to the operation of moving the specified coordinate axis to the specified position, it is also a step in general cutting preparation. In contrast to the [JOG] key operation, the "Move" moves the specified axis to the specified position accurately according to input data. There are three ways for "Move".

- Move
- Limit move
- Halfway move

4.1 Move

Three sub mode buttons [Move], [Half], [Limit] appear after pressing the mode button [Move] in the upper left of the screen. Press [Move] sub mode button, the screen displays the move scene as Fig.4.1.1

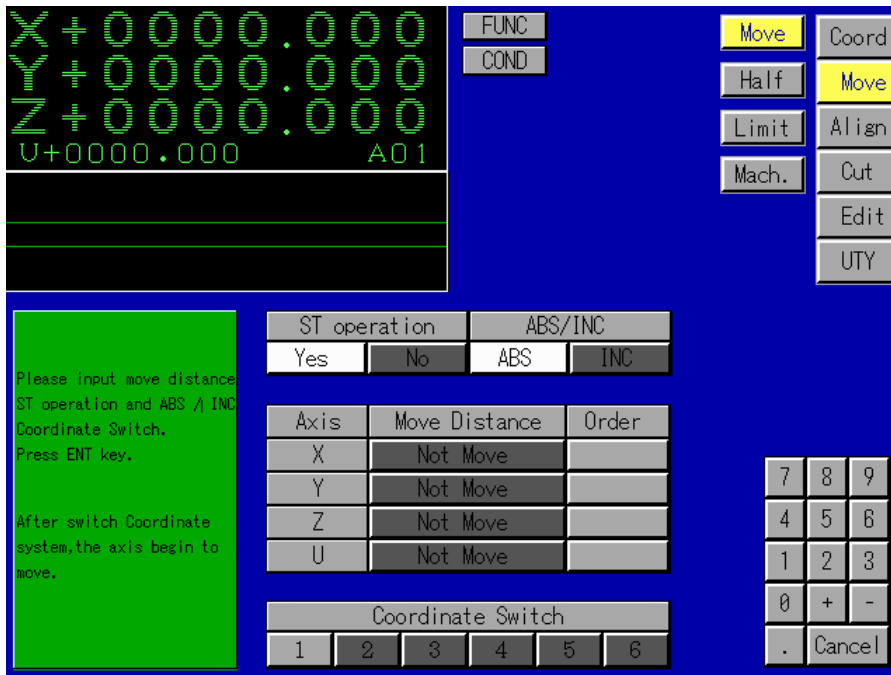


Fig 4.1.1

There are the following input items in this scene:

- [ABS]/[INC] selection
Press [ABS] button represents that the move distance adopts the absolute coordinate value of the current coordinate system. Press [INC] button represents that the move distance adopts the relative coordinate value of the movement of the current position.
- ST operation [Yes]/[No] selection
Press [Yes], the move stops at once if the ST electrode contacts with the work piece during the move process.
- Press [No], the move continues even if the ST electrode contacts with the work piece during the move process.

- Move distance
This parameter is used to input the absolute coordinate value to which to move the axis when the [ABS] is selected and input the distance relative to current position to which to move the axis when the [INC] is selected. The “+”~ “-” represent the move directions. Initial display is “Not move ”

NOTE! If you want to cancel input axis move, you can press [Cancel] button

- Order
The move order of each axis is from "1" to "4". If the move orders of two (or more than two) axes are as same, this two (or more than two) axes move at same time. There is no initial display. The default display order is "1" after inputting the move distance of corresponding axis.
- Coordinate switch
Select the coordinate system of current move position from [1] to [6] coordinate system and press this button.

NOTE! When the ST set is [No], the electrode or machined product, could be destroyed if coordinate axis move continuously after the ST contacts with the axes.

[Example of move operation]

Move from current position X(+1.000),Y(+1.000),Z(+1.000),U(+1.000) to absolute coordinate value X(+10.000), Y(+5.000), Z(-20.000), U(+1.000) position. If the ST contacts with work piece during moving process, the operation stops at once. The order of each axis is that X, Y axes move together the Z axis moves.

Steps:

- 1) Press [ABS] button.
- 2) Press [Yes] button in the [ST operation] item.
- 3) Press the input button corresponding to move distance of X axis (The initial display of this item is "Not Move"), press the digit buttons [1], [0], [.] , [0], [0], [0].
- 4) Press the input button corresponding to move distance of the Y axis, Press digit buttons [5],[.],[0],[0],[0].
- 5) Press input button corresponding to move distance of the Z axis, and press digit buttons[-],[2],[0],[.],[0],[0],[0].
- 6) Press the order button of the [Axis X].(default value is “1”)
- 7) Press the order button of the [Axis Y]. (default value is “1”)
- 8) Press the order button of the [Axis Z].and press the digit button [2].

After pressing the [ENT] key, the electrode moves as Fig. 4.1.2. The scene resumes the initial display after ending the operation.

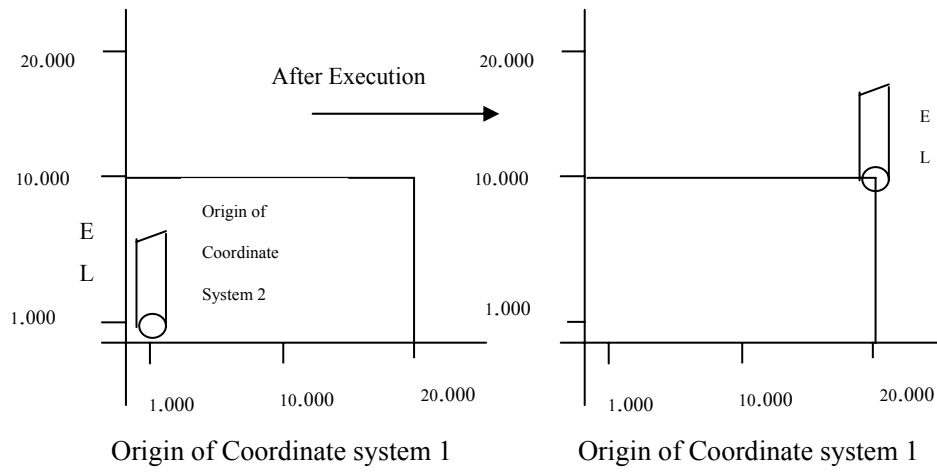


Fig.4.1.2

4.2 Halfway Move

"Halfway Move" refers to the operation of selecting one axis from X, Y, Z and U, and moving the selected axis to the position corresponding to half of its current coordinate value. Press the sub mode button [Half] after pressing the mode button [Move], the screen displays the halfway move as illustrated in Fig.4.2.1

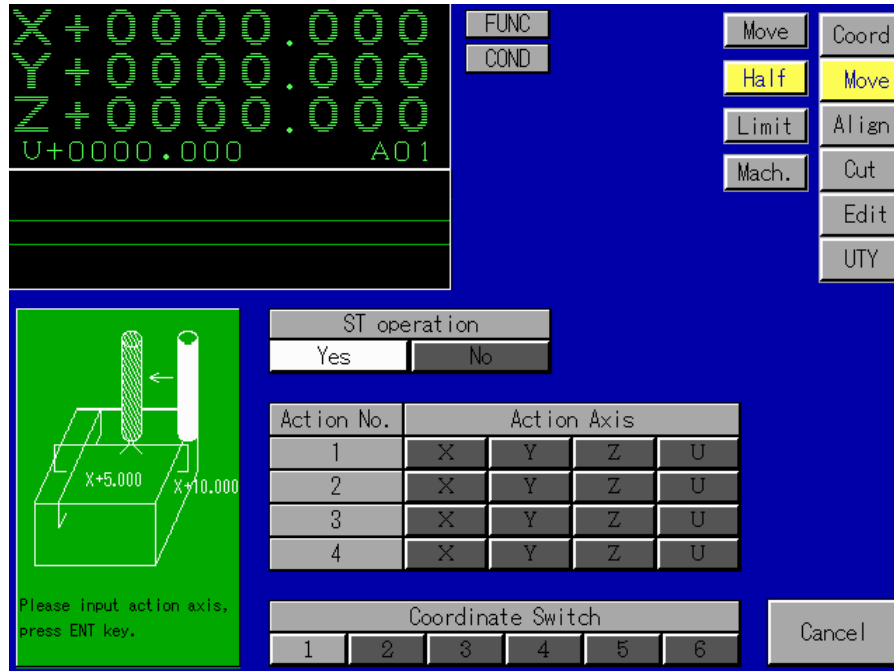


Fig 4.2.1

There are following options in this scene:

- ST operation [Yes]/[No] selection
Press [Yes] button, the move stops if the ST electrode contacts with work piece during the halfway move process.
Press [No] button, the move continues even if the ST electrode contacts with work piece during halfway move process.
- Action
Select action axis from action numbers [1] to [4]. The selectable times of one axis among the X, Y, Z, U axes is only one, and only one axis among the X, Y, Z, U axes can be input with the same number (Press one button of four axes). If one axis is selected move than one times or the several axes are selected as the same numbers, the selections will be omitted.
- Coordinate switch
Select the current coordinate system of halfway move from [1] to [6] coordinate systems .

NOTE! When the ST set is [NO], the electrode or machined product could be destroyed
If the coordinate axis moves continuously after the ST contacts with the work piece.

[Example of halfway move operations]

Do the halfway move operation of the X axis and the Z axis. The current coordinate value is X(+20.000), Z(+5.000).

Steps:

- 1) Press the [Yes] button in the ST items. (If [Yes] is selected, this step can be neglected.)
- 2) Press the [X] button corresponding to the action number [1]
- 3) Press the [Z] button corresponding to the action number [2]

After pressing the [ENT] key, the X axis and Z axis move to X(+10.000) and Z(+2.5000) respectively. The screen resumes the initial display after ending this operation

NOTE! Press the [Cancel] key to cancel the action axis selection.

4.3 Limit Move

"Limit Move" refers to the operation of moving the X, Y, Z axis in specified direction to the machine limit of the CNC supply unit (axis end). Press the sub mode button [Limit] after pressing the mode button [Move], the screen displays the limit move scene as illustrated in Fig.4.3.1

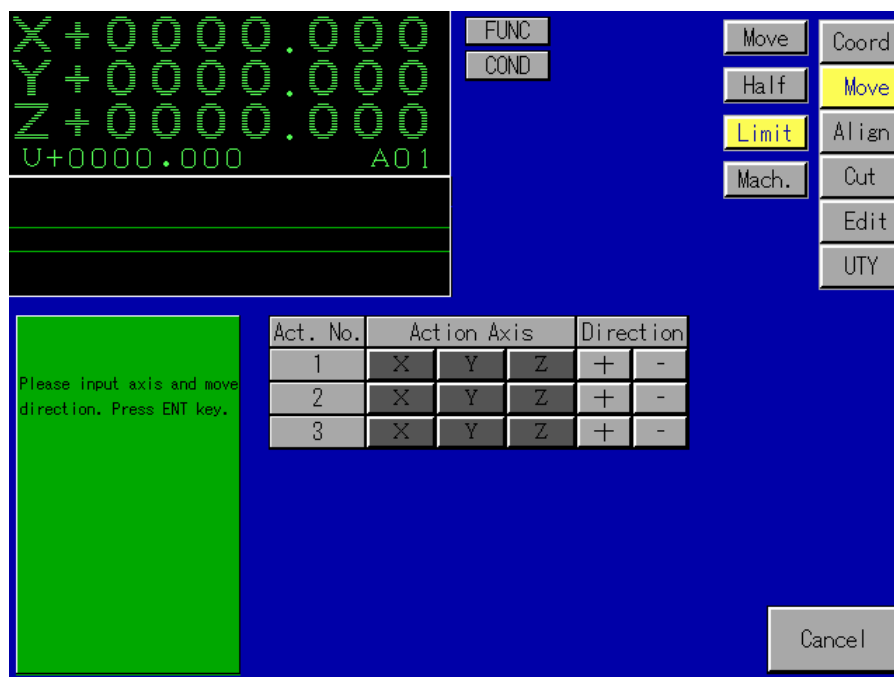


Fig 4.3.1

There are following items to be input in this screen.

- Action
Select action axis from action number [1] to [3]. The selectable times of one axis among the X, Y, Z axes is only one, and only one axis among the X, Y, Z axes can be input with the same number. (Press one of three axes buttons) If one axis is selected more than one time or the several axes are selected as the same number, the selection is the last time input and the previous selections will be omitted.
- Direction
This item determines the limit move direction of selected axis. Corresponding to the action number and the selected action axis, if press [+] button, the axis moves to the limit in positive direction, and the axis moves to the limit in negative direction if press [-] button.

The default direction of limit move for X, Y axis is "+", for Z axis is "-".

NOTE!

- (1)When the ST set is [N], the electrode or machined product could be destroyed if the coordinate axis moves continuously after the ST.
- (2)If the action axis is not selected, the corresponding direction [+], [-] are invalid.

[Example of limit move operation]

Move the X axis to the limit in the "-" direction after moving the Y axis to the limit in the "+" direction.

Steps:

- 1) Press the [Y] button in the [Action] items corresponding to the action number [1], then press the [+] button in the [Direction] select items
- 2) Press the [X] button in the [Action] items corresponding to the action number [2], then press the [-] button in the [Direction] select items

After pressing the [ENT] key, the Y axis and X axis move to their limit position respectively. The screen resumes the initial display after ending the operation

NOTE! Press the [Cancel] button cancel the selected contents.

Chapter 5 Alignment

The alignment is the most important cutting preparation step. The CNC supply unit provides following

five alignment methods:

- Approach face alignment
- Column center alignment
- Comer alignment
- Hole center alignment
- Other alignment

In the Z system, the five alignment sub modes to be select by pressing the [Align] mode button. Press the sub mode button will enter to respectively screen.

5.1 Approach Face Alignment

This sub mode is used to bring the electrode into contact with the work piece from arbitrary direction for detection approach face. There is the detail "Approach face alignment scene for move detail setting of this kind alignment.

5.1.1 Approach face alignment scene

Press the [Face] button in the alignment sub mode selection causes the scene to display as Fig.5.1.1.1

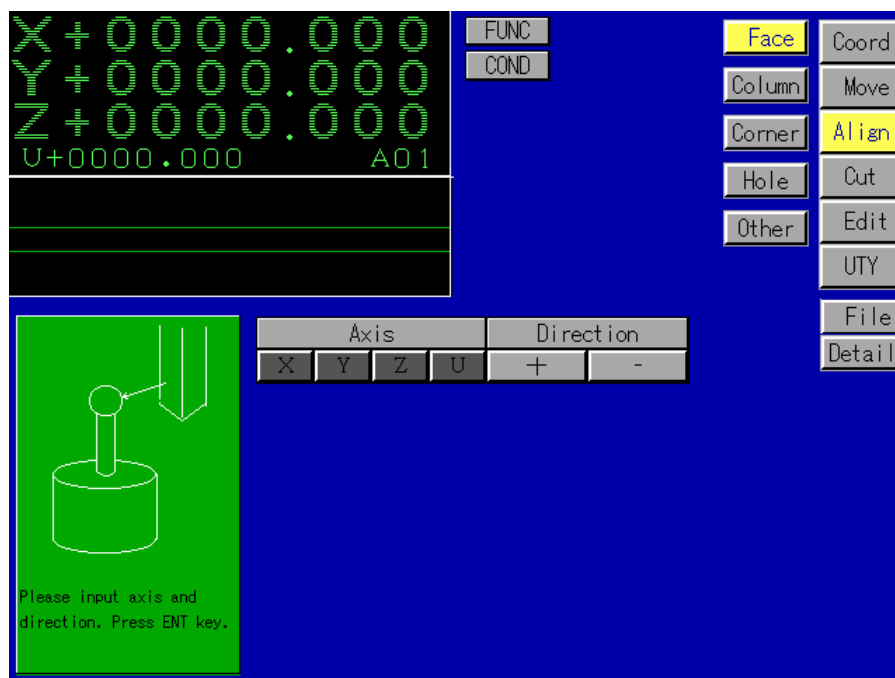


Fig 5.1.1.1

There are following input items in this scene:

- Axis
Select the "Approach face" alignment axis from [X], [Y], [Z], [U] and press the axis button.

- Direction

This item specifies the axis moving direction in "Approach face" alignment. The [+] indicates positive direction and the [-] indicates the negative direction.

5.1.2. Approach face detail alignment scene

Press the [Detail] button for move detail set, the screen displays the detail scene of "Approach face" alignment set as Fig.5.1.2.1.

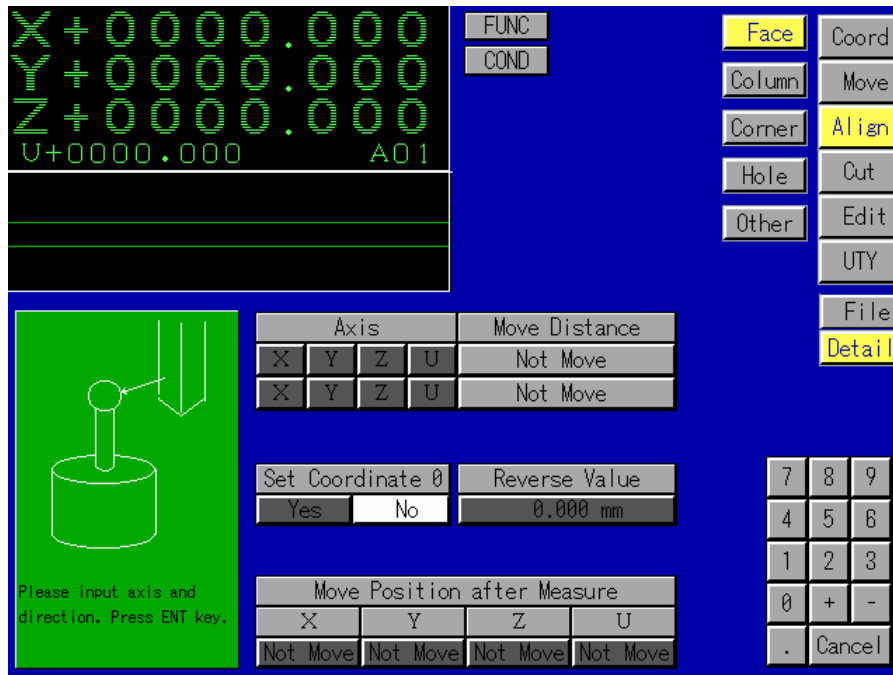


Fig 5.1.2.1

Following input items are added in "Approach face" screen:

- Reverse value

This parameter is used to input the distance which to move the electrode apart from the work piece after the former comes into contact with the latter during the alignment operation.

- Move distance

It is the distance of the electrode moving to its destination. After the electrode moves to its destination, the move operation is ended even if the electrode does not contact with the work piece during "Approach face" alignment process. The initial display is "Not Move".

- Coordinate "0" set [Yes],[No] selection

After pressing the [Yes] for indicating the approach face position being detected, coordinate of the set position id "0". After press the [NO] for indicating the approach face position being detected, the coordinate of the set position id not "0".

- Move position after measure

This parameter is used to input the coordinate of position to which the [X], [Y], [Z], [U] axes move after measure.

NOTE! Be care of using the sign when inputting the move distance. The "+" indicates the positive direction of the axis and the "-" indicates the reverse direction of the axis.

[Example of "Approach face" alignment operation]

Move the electrode on the Z axis in its "-" direction for "Approach face" alignment, and set the measured approach face at "0" and move the electrode to X(+100.000), Y(+ 100.000), Z(+ 100.000), U(+ 100.000) finally.

Analysis: Because the "0" set and move the position after detection can not be set in "Approach face" alignment screen (not detail), this work need to be set in detail "Approach face" alignment.

Steps:

- 1) Move the electrode to just above of the work piece.
- 2) press the [Detail] button in "Face" alignment scene, the screen change to detail 'Face' alignment.
- 3) Press the button in the [Reverse Value] input items, (The default value is 0.000) and then press [-], [1], [0], [0], [0], buttons.
- 4) press the [Z] button in the [Axis] input item.
- 5) Press the button in the [Move distance] input item, (The default value is 0.000), and then press the [-], [1], [0], [0], [0], [0] buttons.
- 6) press the [Yes] button of '0" coordinate set item after measure. (If the "Yes" has been selected ,this step can be neglected.)
- 7) Press the input item button of [X] in the [Move position after measure], and then press the digit buttons, [1], [0], [0], [0], [0], [0], [0].
- 8) Press the input item button of [Y], and then press the digit buttons, [1], [0], [0], [0], [0], [0].
- 9) Press the input item button of [Z], and then press the digit buttons, [1], [0], [0], [0], [0], [0].

After pressing the [ENT] button, the system will perform automatically the "Approach face" alignment.

5.2 Column Center Alignment

Column center alignment refers to the alignment method of detecting the front, back, left and right width of the work piece or the reference sphere and then detecting the center position of the work piece or reference sphere based on the width.

There is the detail screen for detail column center alignment.

5.2.1 Column center alignment screen

Press the [Column] button in the alignment sub mode selections, the screen displays the column center alignment scene as Fig. 5.2.2.1

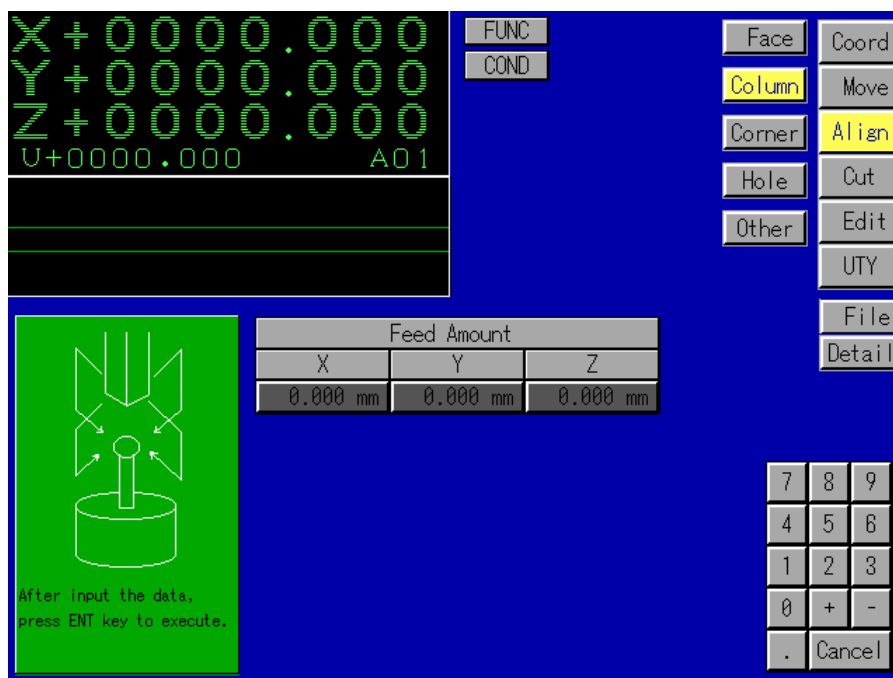


Fig 5.2.2.1

There are following input items in this scene:

- Feed amount X
This parameter is used to set the quickly move amount by which to move the electrode in the X axis away from the contact position with the work piece after the ST operation of the Z axis.
- Feed amount Y
This parameter is used to set the quickly move amount by which to move the electrode in the Y axis away from the contact position with the work piece after the ST operation of the Z axis.
- Feed amount Z
This parameter is used to set the quickly move amount by which to move down the electrode to the contact position with work piece prior to the ST operation in the Z axis or the Y axis.

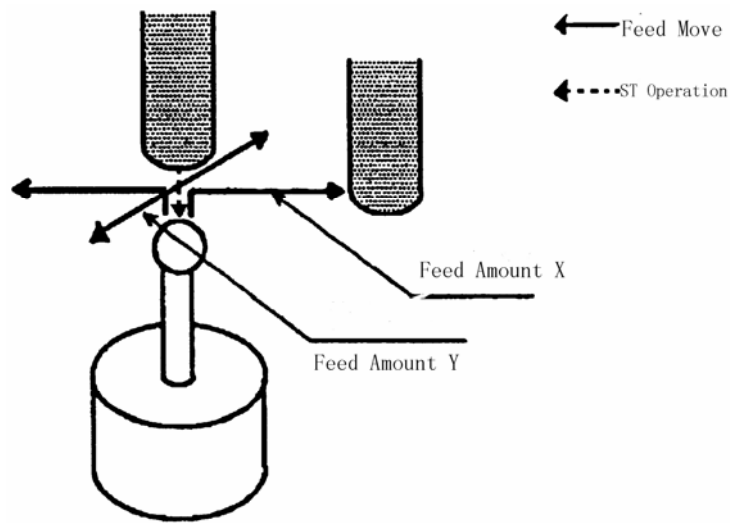


Fig 5.2.1.2

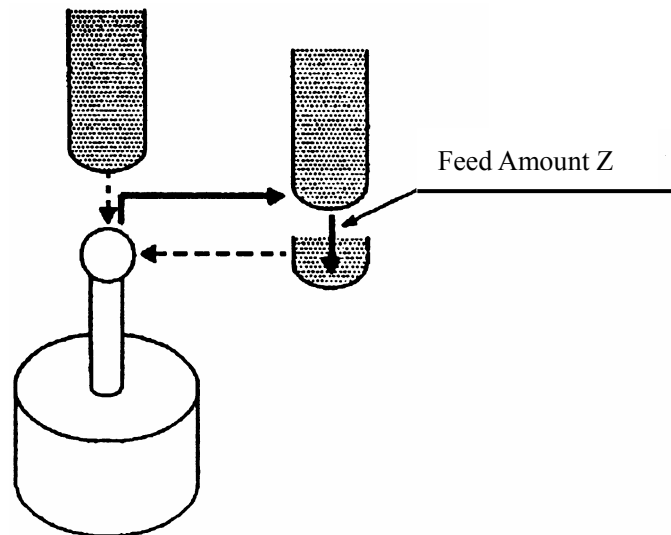


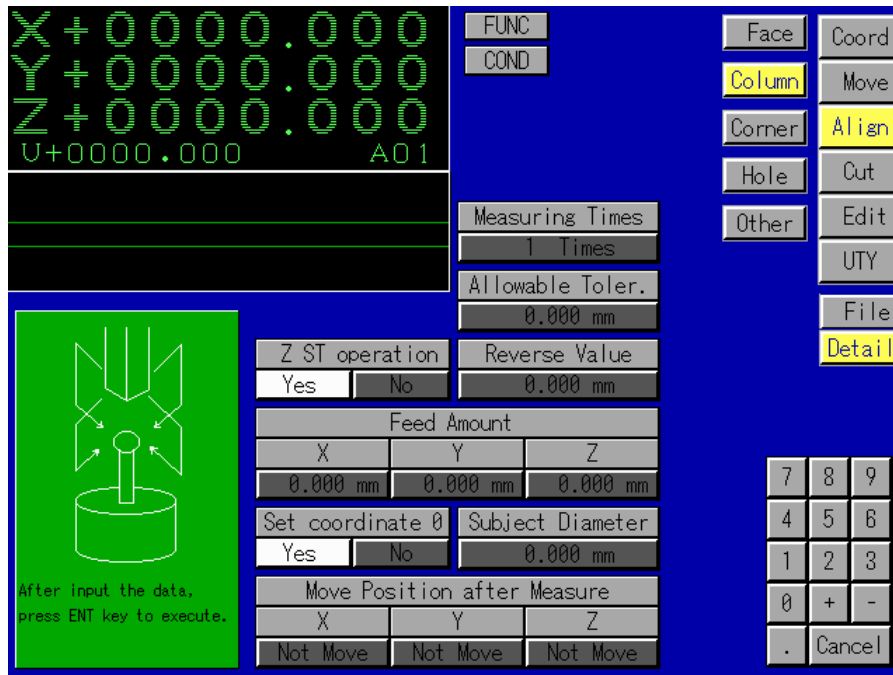
Fig 5.2.1.3

Press the [Detail] button for move detail set, you will enter to the detail column center alignment screen.

NOTE! If the input feed amount of X, Y are too small, the electrode may come into contact with the work piece resulting in shutdown of the alignment operation before the Z axis starts its ST operation.

5.2.2 Column center detail alignment

Press the [Detail] button in the column center alignment scene, the screen displays the detail column center alignment scene as Fig. 5.2.2.1.



There are following input items to be added for detail alignment in this scene:

1. Measuring times

This parameter is used to input the number of times which to measure the work piece for the column center alignment. The number of measuring are from 1 to 3.

2. Allowable tolerance

This parameter is used to set the tolerance with which to execute the measurement so that if the measurement is executed more than one time with the error between the measurements and the first time exceeding the set tolerance after more than two times measuring, an error comment to indicate this is displayed on the screen.

3. ST operation [Yes]/[No] selection of the Z axis

Press the [Yes] button, this indicates that the move operation stops at once if the ST electrode comes into contact with the work piece during the alignment process. Press the [No] button, this indicated that the move operation continues even if the ST electrode comes into contact with the work piece during the alignment process.

4. Reverse value after ST operation

This parameter is used to input the distance between the electrode and the work piece after the electrode contacts with the work piece.

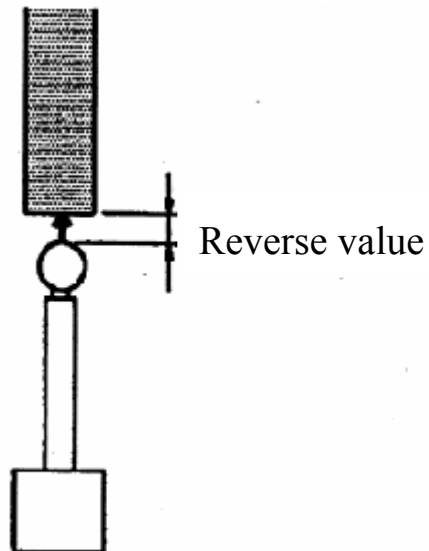


Fig 5.2.2.4

5. Set coordinate "0" [Yes]/[No] selection after measuring

After the column center position. being detected by selecting the [Y], the ".0' coordinate of the column center is set. After the column center being detected by selecting the [N] The "0" coordinate of the column center will not be set.

6. Move position after measuring

This parameter is used to input the coordinate of the [FX], [Y], [Z] axes move position after measuring.

7. Diameter of subject tool

It is used to input the diameter of the reference sphere (or other tools) to be used measuring.

[Example of column center alignment operation]

Use a reference sphere with a diameter of 5.0mm and an electrode of cylindrical shape with diameter. The feed amounts are X (10.000mm), Y(10.000mm), Z (5.000mm).

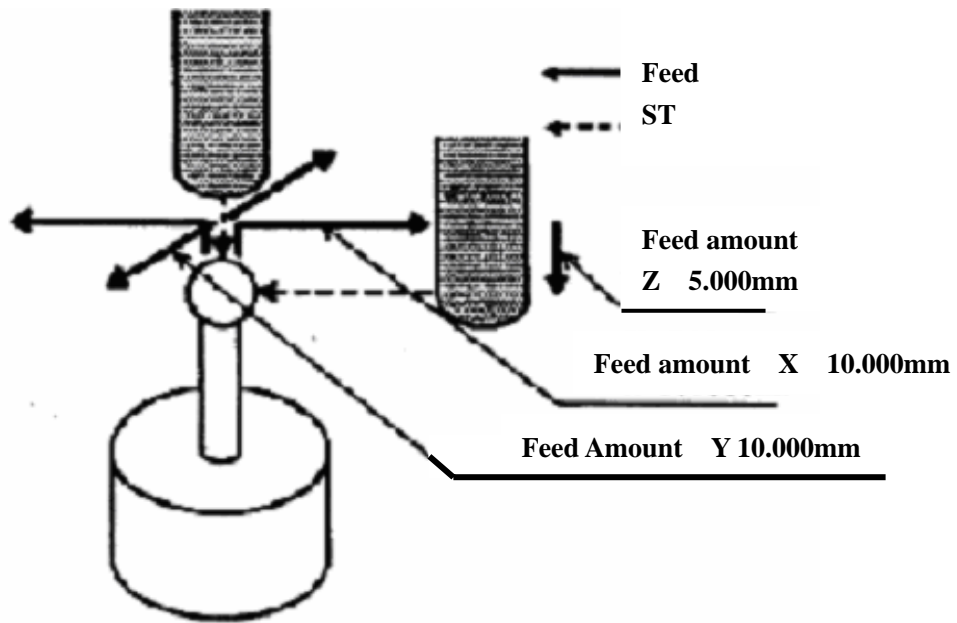


Fig 5.2.2.5

Analysis: The set feed amount can be input in the column center alignment screen. The other input items such as diameter of measurement tool need to be input in the detail column center alignment scene.

Steps:

- 1) Move the electrode to the just above of the reference sphere.
- 2) Press the feed amount [X] button in column center alignment screen and then press the digit buttons [1],[0],[.],[0],[0],[0].
- 3) Press the feed amount [Y] button, then press the digit buttons [1], [0], [0], [0],[0],[0].
- 4) Press the feed amount [Z] button, then press the digit buttons [5], [0], [0], [0].
- 5) Press the [Detail] button and enter to the detail scene (At this time, the feed amounts mentioned above have been displayed in the detail scene.)
- 6) Press the [Yes] button of [ST operation Z]
- 7) Press the button of input items in [Reverse value] ,(The default value is 0.000),and than press [1],[0],[.],[0],[0],[0].
- 8) Press the [Yes] button of the [Set coordinate "0"]
- 9) Press the input button of [Subject diameter], and then press the digit buttons [5], [0], [0],[0],[0].

After pressing the [ENT] key, the Z system will perform automatically the column center alignment operation.

5.3 Corner Alignment

This sub mode is used to measure the two sides of work piece and detect its edge. There is the detail comer alignment screen for detail alignment.

5.3.1 Comer alignment scene

Press the [Comer] button in the alignment sub mode selection, the screen displays the comer alignment scene as Fig .5.3.1.1

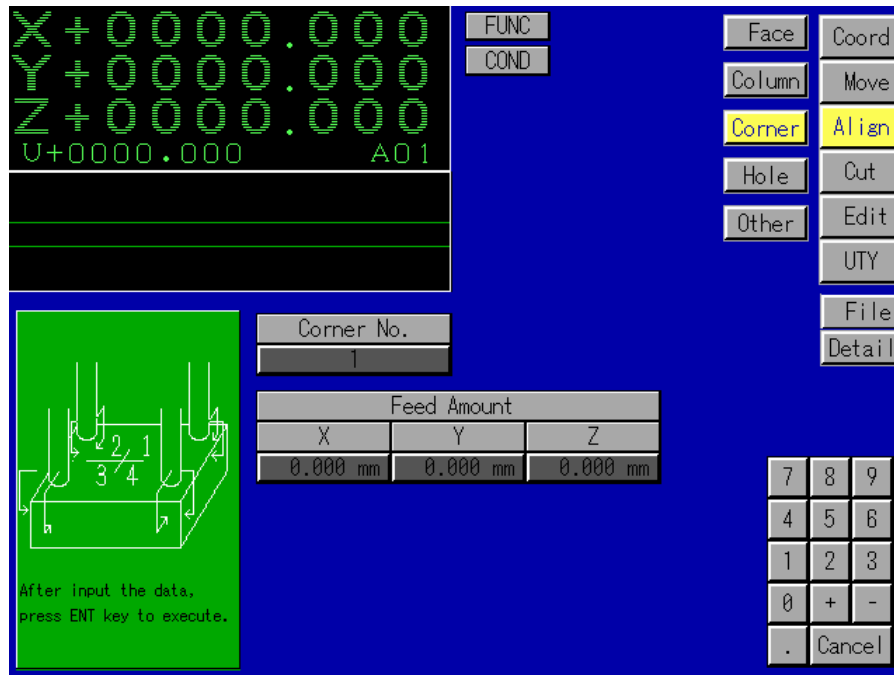


Fig 5.3.1.1

There are following input items of the comer alignment scene:

- Comer No.
This parameter is used to input the No. of the comer to be selected.
- Feed amount X
This parameter is used to set the quick move amount to move the X axis to the position where the electrode could not contact with the work piece after the ST operation of the Z axis.
- Feed amount Y
This parameter is used to set the quick move amount to move the Y axis to the position where the electrode could not contact with the work piece after the ST operation of the Z axis.
- Feed amount Z
This parameter is used to set the quick move amount to move the Z axis to the position where the electrode could contact with the work piece after the ST operation of the X and Y axes.

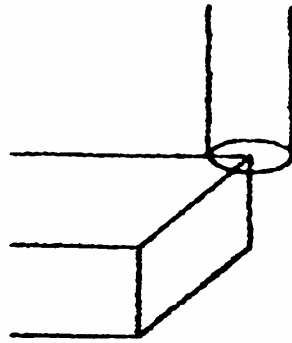


Fig 5. 3. 1. 2

5.3.2 Comer detail alignment scene

The screen displays the comer detail alignment scene as Fig.5.3.2.1 after pressing the [Detail] button in comer alignment scene.

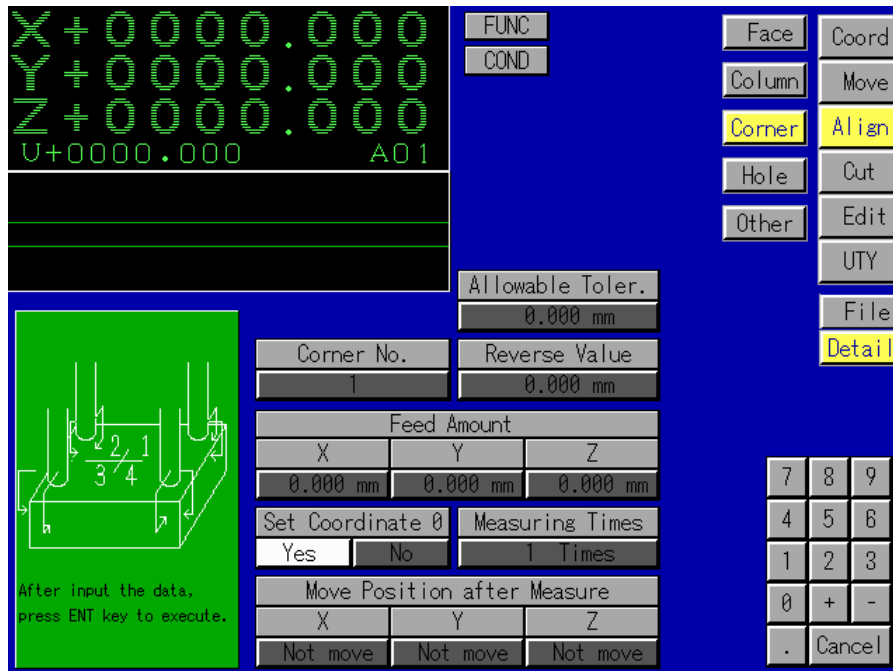


Fig 5.3.2.1

There are following input items to be added in detail scene'

- Measuring times
This parameter is used to set the number of times of comer alignment operation which maybe "1" to "3".
- Allowable tolerance
This parameter is used to set the tolerance with which to execute the that if the measurement is executed more than one time with the error between the measurements and first time exceeding the set tolerance after more than two times measuring, an error comment to indicate this is displayed on the screen.

- Reverse value after ST
This parameter is used to input the distance between the electrode and the work piece after the electrode contacts with the work piece.
- Set coordinate "0" [Yes, No] selection after measuring
After the corner alignment position being detected by selecting the [Yes], the set coordinate of the position being detected by selecting the [No], the set coordinate of the position is not "0".
- Move position after measure
This parameter is used to input the coordinate of the [X], [Y], [Z] axes move position after measuring.

[Example of corner alignment operation]

Use an electrode of cylindrical shape to measure Corner 3 for corner alignment and move the electrode center to the corner of the corner alignment.

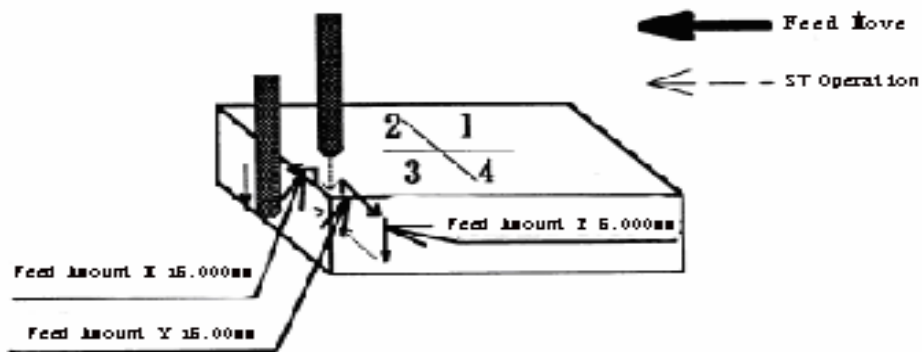


Fig 5.3.2.2

Analysis: The No. of the corner and quick feed amount can be input in the corner alignment scene the set in because the detail corner alignment scene is not needed in this time set.

Steps:

- 1) Move the electrode to the just above of Corner 3.
- 2) Press the input button [Corner No.] in the corner alignment scene, then press the digit button [3].
- 3) Press the feed amount input button [X], and then press the digit buttons [1], [5], [.] , [0],[0],[0].
- 4) Press the feed amount input button [Z], and then press the digit buttons [1], [5], [.] , [0],[0],[0].
- 5) Press the feed amount input button [Z], and then press the digit buttons [5], [.] , [0], [0],[0].

After pressing the [ENT] key, the Z system will perform automatically corner alignment.

5.4 Hole Center Alignment

Hole center alignment refers to the operation of testing the front, back, left and right width of hole of the work piece and then testing the hole center based on the width.

There is the detail hole center alignment scene for move detail alignment set.

5.4.1 Hole center alignment scene

Press the [Hole] in selection button of the alignment sub mode, the screen displays the hole center alignment scene as Fig 5.4.1.1.

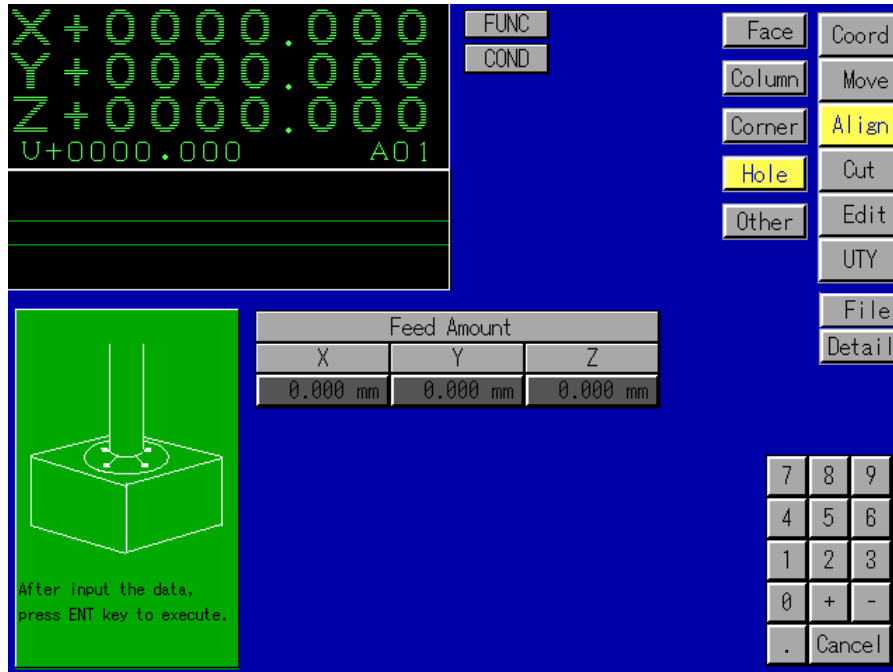
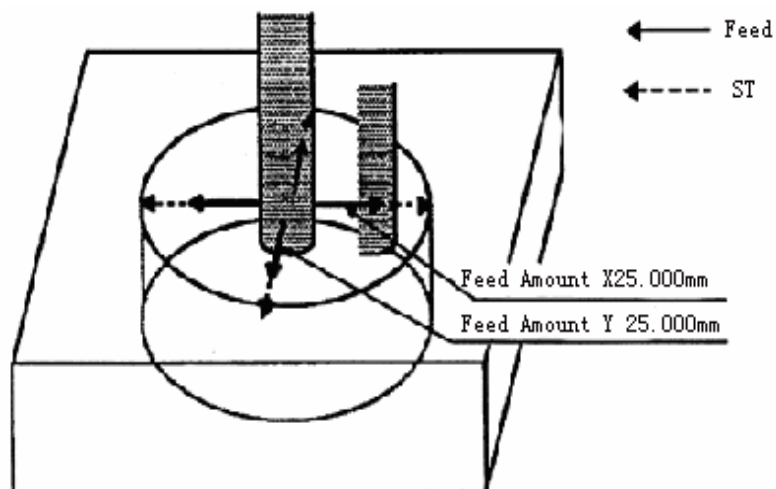


Fig 5.4.1.1

There are following, input items in the hole center alignment screen:

- Feed amount X

This parameter is used to input the quick move amount which to move the electrode from its current position to a position near the work piece before the X axis starts its ST operation. The amount is the sum of positive and negative values.5.4.1.2.



- Feed amount Y
This parameter is used to input the quick move amount which to move the electrode from its current position to a position near the work piece before the Y axis starts its ST
- Feed amount Z
This parameter is used to input the quick move amount which to move the Z-axis the position where the electrode could contact with the work piece before the X and Y axes.

NOTE! Please input this parameter at a slightly smaller value to prevent the electrode contacts with the work piece.

5.4.2 Hole center detail alignment scene

Press the [Detail] button in the hole center alignment scene, the hole center detail scene will be shown in the screen as Fig.5.4.2.1.

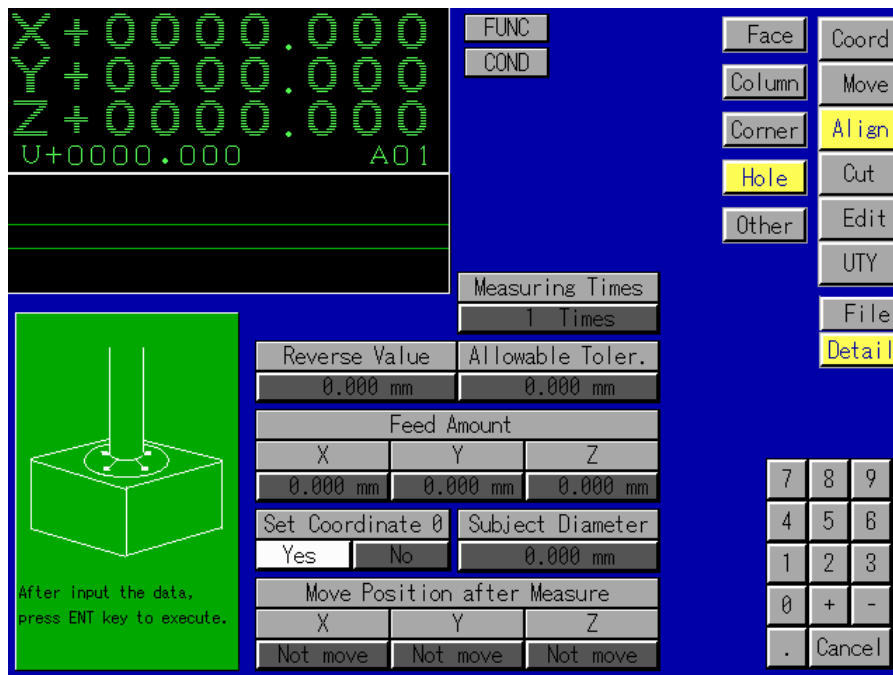


Fig.5.4.2.1

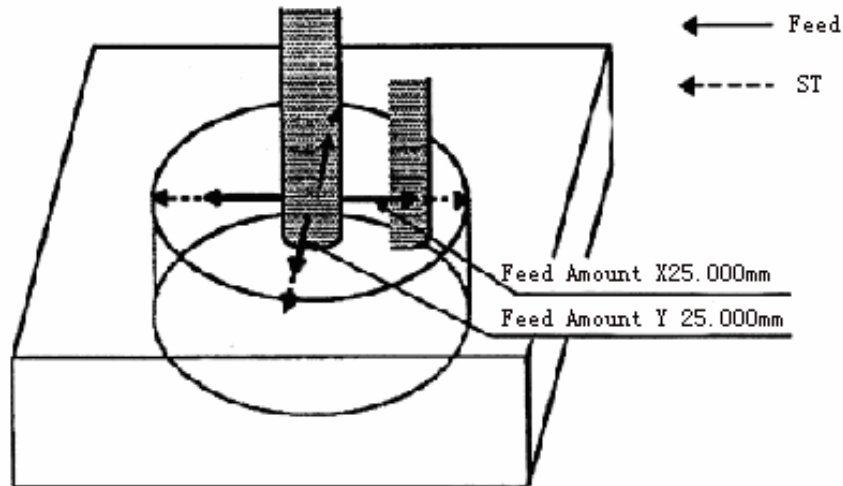
There are following input items to be added in the hole center alignment screen:

- Measuring times
This parameter is used to input the operation numbers of the hole center alignment. Its range is from 1 to 3
- Allowable tolerance
This parameter is used to set the tolerance with which to execute the measurement so that if the measurement is executed more than one time with the error between the measurements and the first time exceeding the set tolerance after the more than two times measuring, an error comment to indicate this is displayed on the screen.
- Reverse value after ST
This parameter is used to input the distance between the electrode and the work piece after the former contacts with the latter.
- 4. Set coordinate "0" [Yes]/[No] selection after measuring
After the hole center position being detected by selecting the [Yes], the set coordinate of the position is "0". After the hole center position being detected by selecting the [No], the set coordinate of the position is not "0".
- Diameter of subject tool
This parameter is used to input the diameter of measuring tool which is used to measure the hole diameter in the work piece.
- Move position after measure
This parameter is used to input the position coordinate to which the [X], [Y], [Z] axes move after measuring.

[Example of hole center alignment operation]

Used an electrode of cylindrical shape with a diameter of 10.0mm to measure a hole drilled in the work piece with a diameter of 50.0mm for "Hole center" alignment.

Analysis :The data need to be input in the hole center alignment screen and in the hole center detail alignment screen.



Steps:

- 1) Move the electrode to near the center of the drilled hole in the work piece.
- 2) Press the feed amount input button [X] in the hole center alignment scene, and then press the digit buttons [2], [5], [.] , [0], [0], [0].5)
- 3) Press the feed amount input button [Y], then press the digit buttons [2], [5], [.] , [0], [0],[0].
- 4) Press the [Detail] button for entering to the hole center detail alignment scene. (The feed amounts input will be displayed on this screen.)
- 5) Press the [Subject Diameter] button, and then press the digit buttons [5], [0], [.] , [0], [0], [0].for inputting the diameter of the hole in the work piece.

After press the [ENT] key, the system will perform automatically the hole center alignment.

5.5 Other Alignment

The other alignments are the two alignment methods other than the four alignment methods mentioned above Random 3 points alignment and the Discharge alignment.

In the Z screen, after pressing the [Align] mode button, the five alignment sub mode buttons can be selected. The screen and input methods after pressing the [Face], [Column], [Comer], [Hole], buttons have been introduced above. There are two new sub mode. Discharge alignment and Random 3 points alignment to be selected after pressing the [Other] button.

5.5.1 Random 3 points alignment

Random 3 points alignment refers to the method of measuring the center of special shape work piece which could not be measured by other methods such as the column center alignment and the hole center alignment.

5.5.1.1 Random 3 points manual alignment

This mode allows the JOG key to be used to feed the X and Y axes to bring the electrode ST at three randomly selected points of the work piece, thereby calculating the center of these three points.

Select the [3 points] button, the screen displays the random 3 points scene (Manual is the default state)as Fig .5.5.1.1.1.

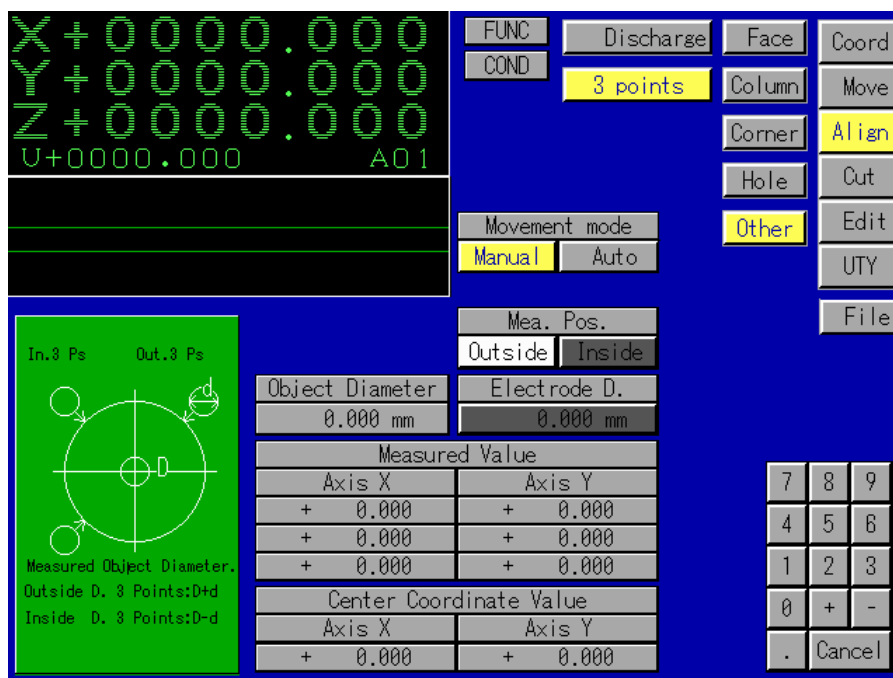


Fig .5.5.1.1.1s

There are following input items in the random 3 points manual alignment screen:

- [Manual]/[Auto] Selection
[Manual] is the default state. The scene changes to the random 3 points automatic alignment scene by pressing [Auto].

REFERENCE : 5.5.1.2 Random 3 points "Auto" alignment

- [Outside]/[Inside] selection

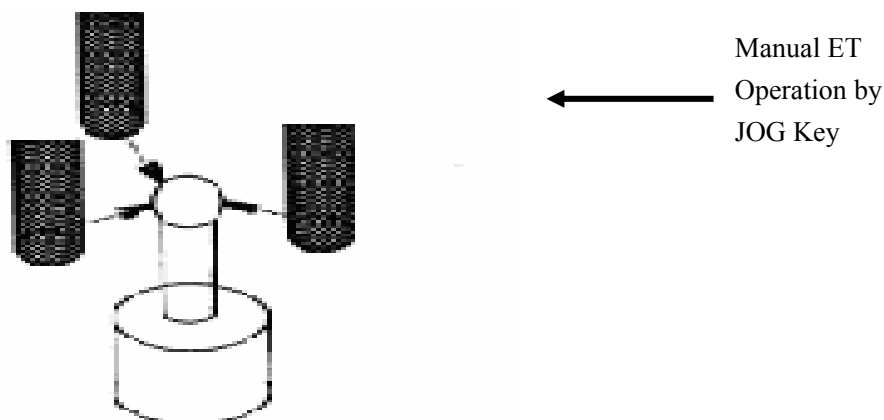
This parameter is used to select whether to measure the inside or outside diameter for the alignment. [Outside] is the default selection. At this time, the ST 3 points are on outside circle. If press the Inside] button, the ST 3 point are on inside circle.

- Subject diameter

This parameter id used to input the measuring electrode diameter for alignment.

[Example of Random 3 points manual operation]

Used an electrode with a diameter of 10.000mm to execute the "Random 3 point outside diameter alignment.



Steps:

- 1) Press the [Manual] button.
- 2) Press the [Outside] button.
- 3) Press the input button of the [Subject Diameter], and then press the digit buttons [1].
- 4) Used the [JOG] key to move the electrode to a position where the electrode contacts with the three randomly selected points on the outside diameter of the work piece. The coordinate values of the 3 points, the calculated center coordinate and measuring tool diameter will be displayed respectively in the [Measured value], [Center coordinate value] and [Object Diameter] items.

5.5.1.2 Random 3 points "Auto" alignment

This mode refers to the operation of calculating the center of the 3 points and the diameter of the work piece after the electrode executes automatically the ST with the three randomly selected points of the work piece.

Press the [Auto] button on the Random 3 points "Manual" alignment screen, the screen will change to the random 3 points "Auto" alignment scene as Fig.5.5.1.2.1.

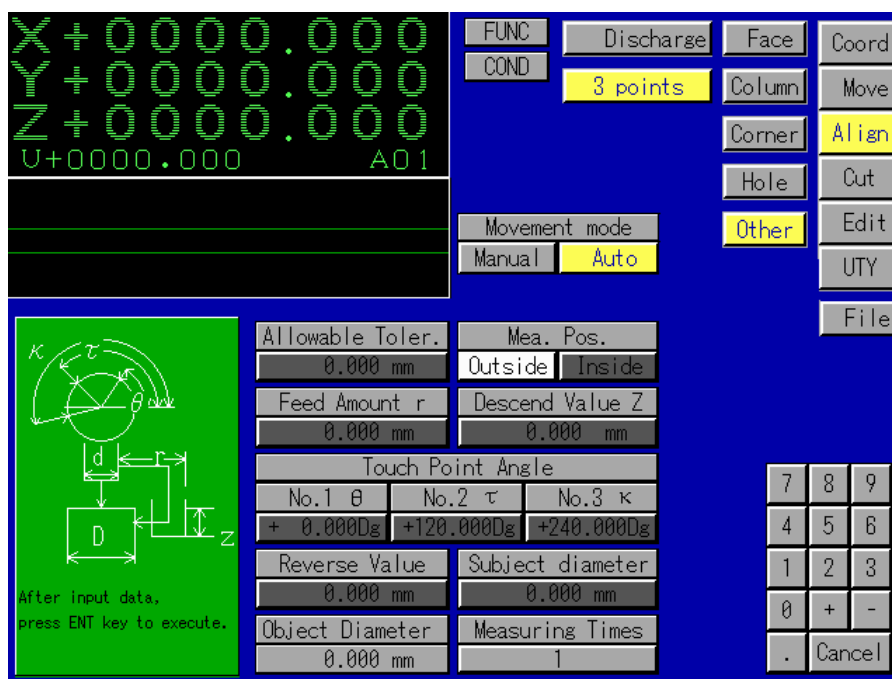


Fig.5.5.1.2.1.

There are following input items in this screen:

- [Outside]/[Inside] selection
This parameter is used to select whether to measure the inside or outside diameter for alignment. If the [Outside] is selected, the ST 3 points are on outside diameter points. If the [Inside] is selected, the ST 3 points are on inside diameter points.
- Measuring times
This parameter is used to input the operation times of the random 3 points "Auto" alignment. Its range is from 1 to 3.
- Allowable tolerance
This parameter is used to set the tolerance with which to execute the measurement so that if the measurement is executed more than one time with the error between the measurements and the first time exceeding the set tolerance after the more than two times measuring, an error comment to indicate this is displayed on the screen.
- Feed amount r
It is the move amount of the electrode moving to the position where the electrode does not contact with the work piece.
- Descend value
It is the move amount of the electrode moving down to the position where the electrode could contact with the work piece before the ST operation starts.
- Touch angle, No. 1θ, No. 2τ, No. 3 κ.
They are angles of 3 touching points for ST operation.

It is used to input the distance between the electrode and the work piece after the electrode contacts with the work piece.

- Subject diameter

It is used to input the diameter of the measuring tool.

[Example of Random 3 point "Auto" alignment operation]

Use a reference sphere with a diameter of 5.0mm and an electrode of cylindrical shape with a diameter of 10.0mm to execute the random 3 points (Auto. Outside) alignment.

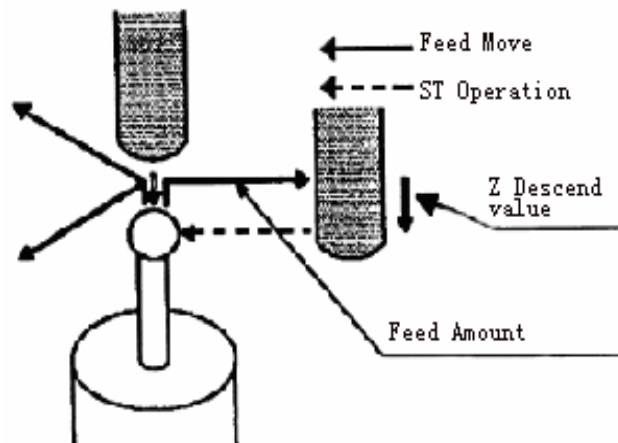


Fig 5.5.1.2.2

Steps:

- 1) Move the electrode to near the position just above the reference sphere.
- 2) Press the [Outside] button.
- 3) Press the input button of the [Feed amount], and then press the digit buttons [2],[0],[.],[0],[0],[0].
- 4) Press the input button of [Descend value z], and then press the digit, buttons [5],[.],[0],[0],[0].
- 5) Input [01, [.], [01, [01, [0] in the input item [No. 1 01, [1], [2], [0], [.], [0], [0], [0] in the input item [No.2 ~] and [21, [4], [0], [.], [0], [0], [0] in the input item [No.3 κ].
- 6) Press the input button of [Subject Diameter], and then press the digit buttons [1], [0], [.],[0],[0],[0].

After pressing the [ENT] key, the Z system will perform automatically the random 3 points "Auto" "Outside" alignment. The displayed value of [Object Diameter] is 5.000mm after measuring.

5.5.2 Discharge alignment

Discharge alignment refers to the operation of moving the axis (slight modify) by pressing the [JOG] key to execute the electrode alignment.

Press the [Other] button in [Alignment] mode, and then select [Discharge alignment], the screen displays the discharge alignment scene as Fig 5.5.2.1

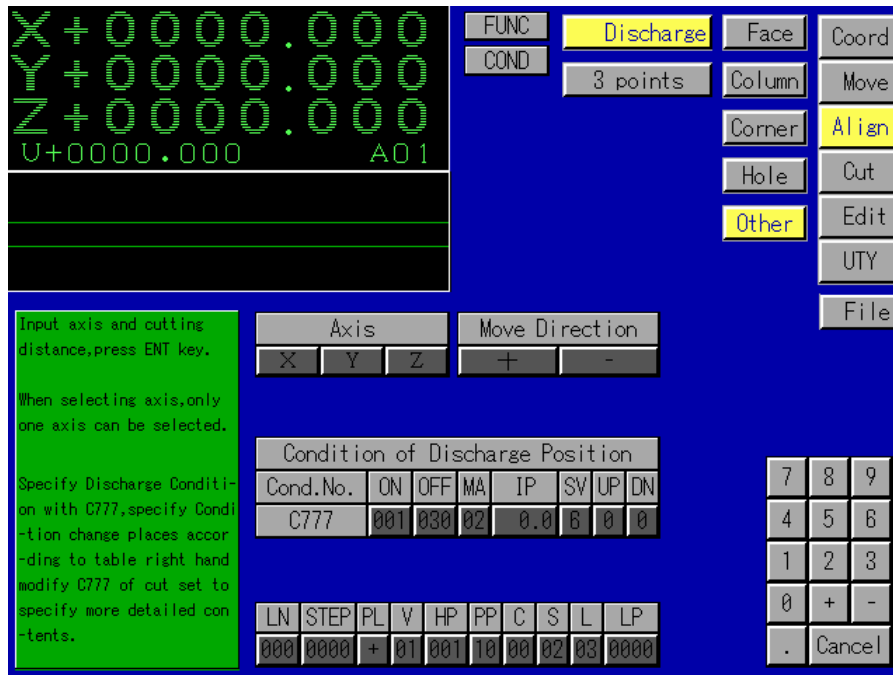


Fig 5.5.2.1

There are following input items in this screen:

- Axis [X]/[Y]/[Z] selection
It is used to input the axis for the discharge alignment.
- Move direction [+] / [-] selection
It is used to input the move direction of the axis.
- Condition of the discharge position
It is used to change the data of c777 items of the discharge alignment. Press the input button of change condition and input the new values.

PART3 CUT

Having completed the cutting preparation such as coordinate set, move and alignment, you will enter to the cutting mode for the final target "Cut". This part will introduce the different cutting methods of the Z system.

This part consists of following chapters:

Chapter 6 Assist cutting

Chapter 7 Manual cutting

Chapter 8 Multiple cutting

Chapter 9 User cutting

Chaper 10 Edit

Chapter 6 ASSIST CUTTING

In this chapter, the system determines automatically the cutting conditions, and completes automatically the NC program for this cutting to do cutting process finally(shape, electrode wear, roughness and so on). The assist cutting includes two cutting methods.

1. Assist cutting
2. Single cutting

The set and operation method of the two cutting will be introduced as follows.

6.1 Assist Cutting

Press the [Cut] button from the mode select buttons, at this time, press the [Assist] mode button from the four cutting selections appear in the screen, and then select the [Assist] button from the [Assist] and the [Single] sub mode button, the screen displays the assist cutting scene as Fig 6.1.1

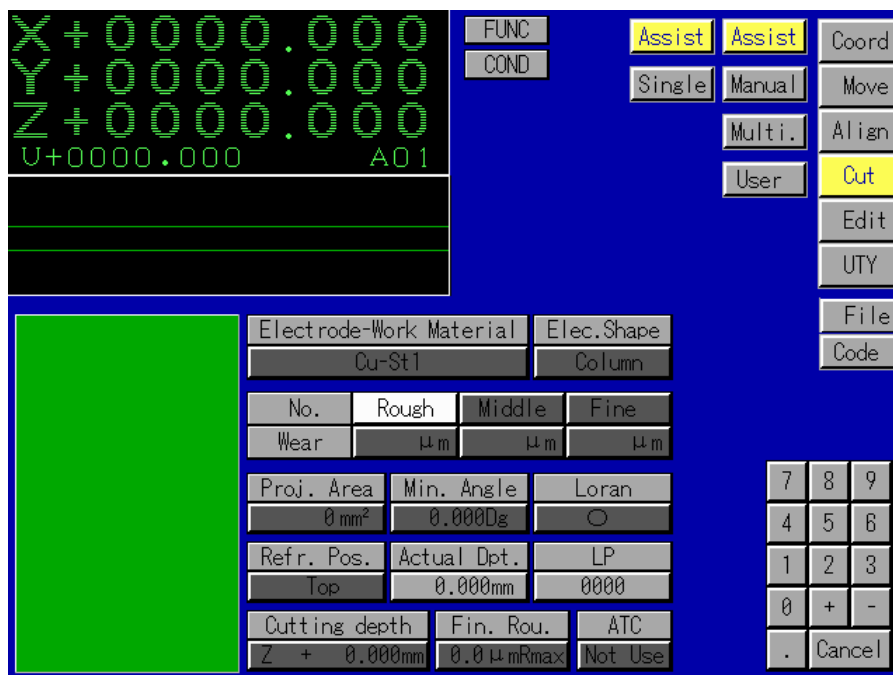


Fig 6.1.1

In the assist cutting scene, the cutting condition and the necessary data must be input and calculated automatically respectively. Then the system calculates the cutting condition from rough cutting to final cutting according to the input data and cutting data, and develops automatically the cutting process according to the input data(electrode materials, electrode).

There are following input items in this scene:

1. ATC

Press [ATC] input button, the screen displays the help scene at the left bottom as Fig 6.1.2

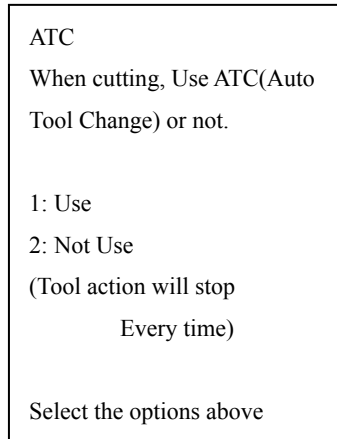


Fig.6.1.2

The selectable state of ATC includes:

- (1) Use
- (2) Not Use

Press the digit key to select the No..

2 .Electrode materials

This parameter is used to select the electrode materials. When pressing the [Elec. Mtr.] input button, the help scene will appear in left bottom of screen as Fig.6.1.3.

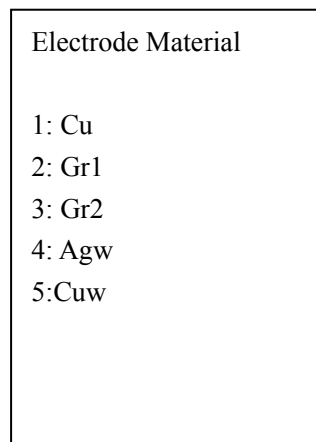


Fig.6.1.3

The selectable electrode materials are as follows:

- (1) Cu: copper electrode
- (2) Gr1: graphite (ED3, ISQ-63) electrode
- (3) Gr2: graphite (POCP, EDM-3) electrode
- (4) AgW: Silver and tungsten electrode
- (5) CuW: Copper and tungsten electrode

The No. of the electrode materials are input by pressing the digit buttons.

3. Work piece materials

This parameter is used to select the work piece materials. Press the [Work Mtr.] input button ,the help scene will appear in the left button of this screen as Fig 6.1.4

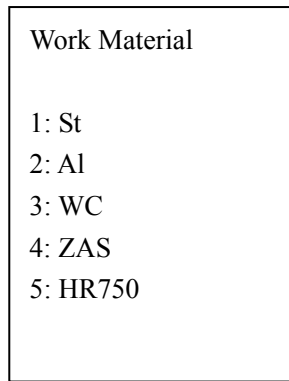


Fig.6.1.4

The selectable work piece materials are:

- (1)St: steel(SKD61,NAK80,STAVAX,PD555)
- (2)Al: aluminum alloy
- (3)WC: tungsten and carbon
- (4)ZAS: sub-aluminum alloy
- (5)HR750:copper alloy

The No. of the work piece materials are input by pressing the digit button.

NOTE! Some work piece materials could not be selected according to the different electrode materials.

This parameter is used to select the electrode shape. After pressing the [Elec. Shape] button, the help scene will be displayed at the left bottom of the screen as Fig.6.1.5

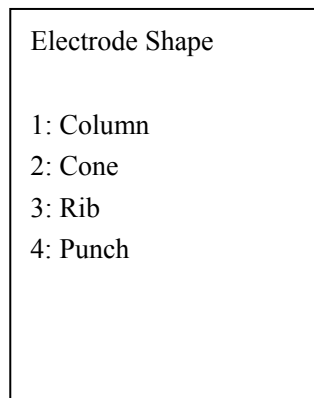


Fig.6.1.5

There are four electrode shapes as follows

- (1) Column shape (Square and circular columns are included in this type.)
- (2) Rib shape (This type is selected for an electrode, the projected area of which is fiat in terms of its aspect ratio.)
- (3) Cone shape (Square and circular cones are included in this type. This type is selected for an electrode .In this case, the projected area will change curing the cutting process).

(4) Punch (This type of electrode is selected producing punch refereed to as "Reverse Discharge"
In this case, a work piece being produced is installed on the machine head while an electrode
is placed on a cutting table.)

The No. of the electrode shapes are input by pressing the digit buttons.

6.2.1 Assist Code

There are two sub mode buttons named [File] and [Code] respectively on the right of [Assist] mode screen. If [Code] button is pressed, a NC file which contains automatic processing technological parameters list will be created. As shown in Fig. 6.1.6, after pressing [Code] button, user can name the NC file within 8 characters and save it or cancel operation.

User can also find [Code] button in other mode screen, such as [Cut]->[Manual]->[Line] and [Cut]->[Manual]->[Arc].

The function of [File] button will be introduced in Chapter 13.

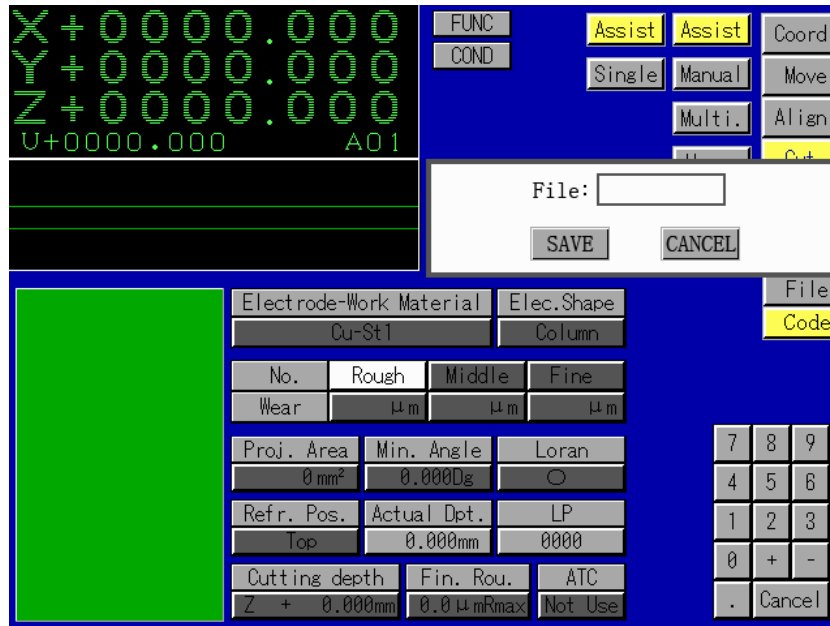
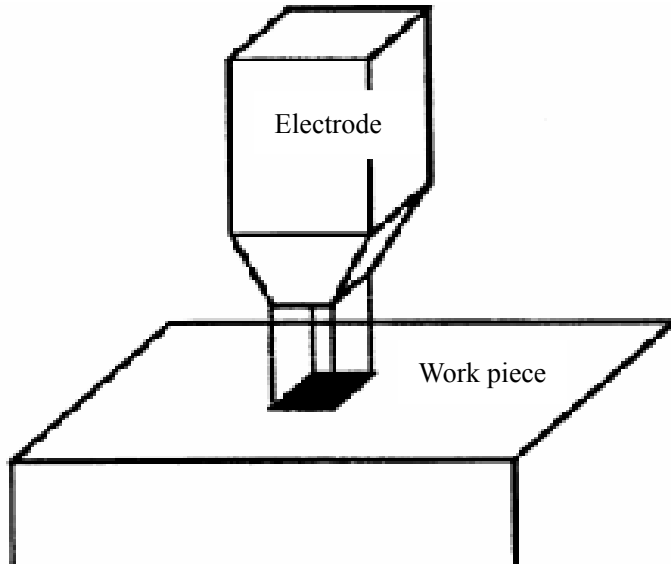


Fig. 6.1.6

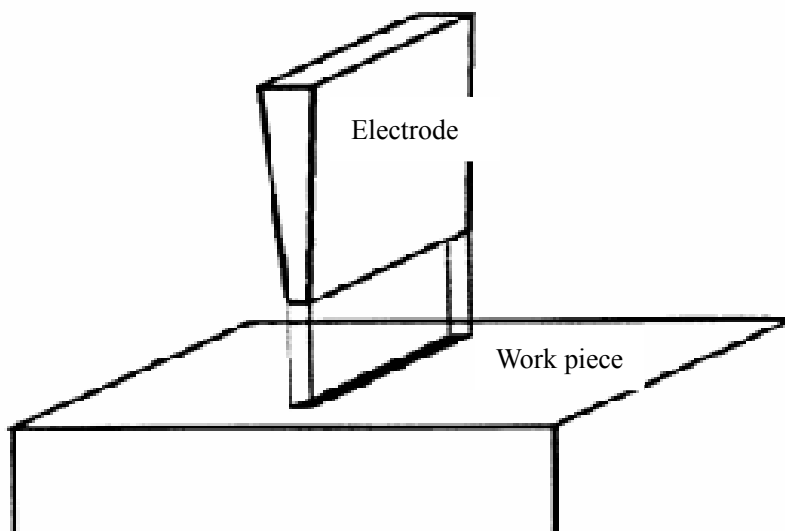
INFORMATION ABOUT THE ELECTRODE SHAPE

1) Column shape

This type is selected when the electrode bottom is sharp regardless of the sectional shape of the electrode (such as round or square).

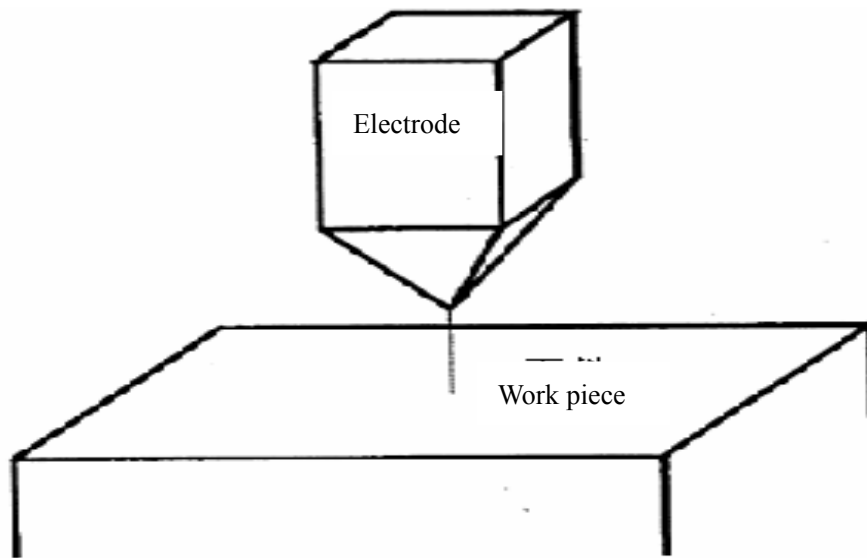


This type is selected when the projected area of the electrode is flat in terms of its aspect ratio as in the case of slitting.

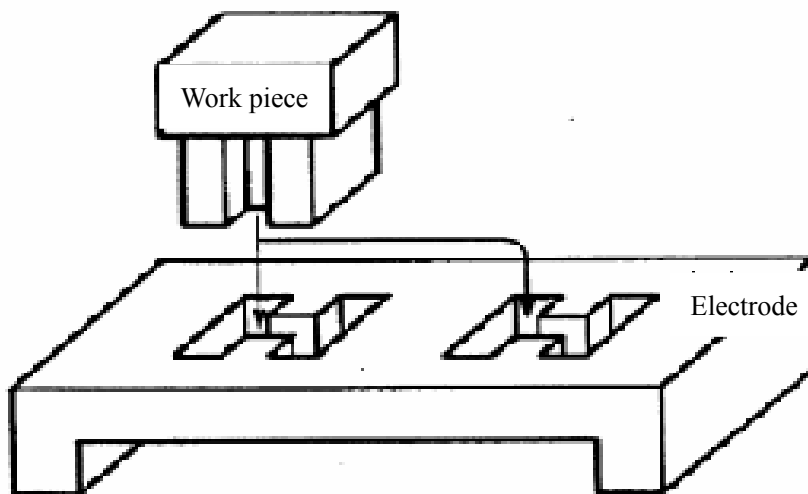


3) Cone shape

This type is selected when the electrode bottom is sharp regardless of the sectional shape of the electrode (such as round or square).



4) This type is selected when as electrode with hold formed in it is used to machine a work piece into a columnar shape.



This method is called generally as "Reverse Discharge" and is used for cutting complex shape punch. The hole may be formed the electrode, for example, by wire-cut EDM. In this case, the work piece need to be fixed on the machine head and the electrode should be fixed on-the work piece (plane) table. The selection of "Punch" causes the cutting to be performed with the polarity set opposite to that in the case of ordinary EDM cutting

(1) Tool No.

The numbers electrode of used in assist cutting are 1 electrode, 2 electrodes, 3 electrodes. When the ATC selection is "Use", input the tool No. corresponding to the electrode to be used.

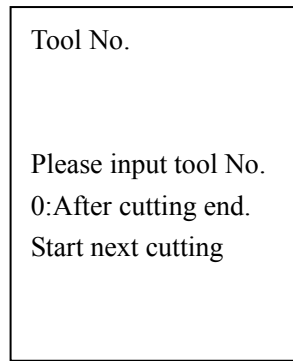
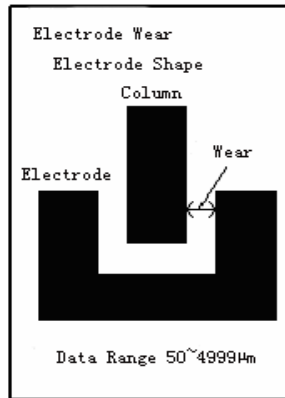


Fig 6.1.6

(2) Electrode wear

The input item corresponding to the No .is the electrode wear, when the initial display of the electrode wear is as “Not Use”, the electrode wear value is input corresponding to the electrode wear of the electrode to be used.

Fig 6.1.7



(3) Projected area

If the [Proj, area] item is selected, the screen displays the help scene at the left bottom as Fig 6.1.8.

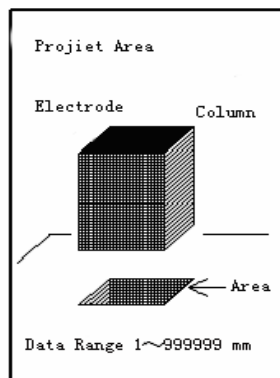


Fig 6.1.8

Input the projected area.

The "Projected area" refers to here as follows:

- (1) Bottom area of the electrode if the electrode is of column shape.
- (2) Projected area of the electrode (actually involved in the cutting) viewed from its cutting direction if the electrode is of rib or cone shape.
- (3) The discharge area of EDM cutting process (in terms of projection in the X,Y plane), if the electrode is intended for punching.

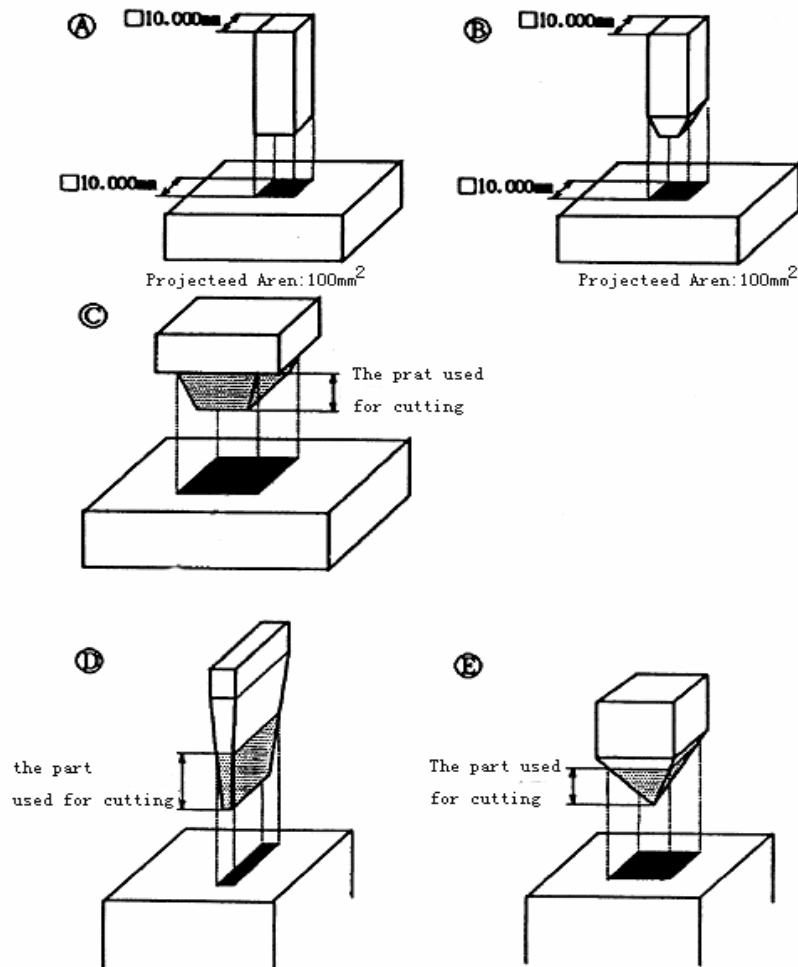
NOTE! The bottom hole area should be subtracted while calculating the projected area
If there is a bottom hole in the electrode.

INFORMATION ABOUT THE PROJECTED AREA

The energy available from EDM has a close relation with the projected area of an electrode used in the machining. Therefore this area should be calculated (or measured) as accurately as possible for input in the parameter.

“Projected area”

The “Projected area” represents the area of the electrode projected onto the work piece.



Cases (A) and (B) are equal in their projected areas. This means that the two cases allow the projected area of the electrode to be calculated with the shape of its tip ignored. Cases (C), (D) and (E) require the projected area of the electrode to be calculated only for its portion involved in the machining.

- Min. gradient angle

Press the [Min. Angle] button, the screen will display the scene at the left bottom, as Fig 6.1.9

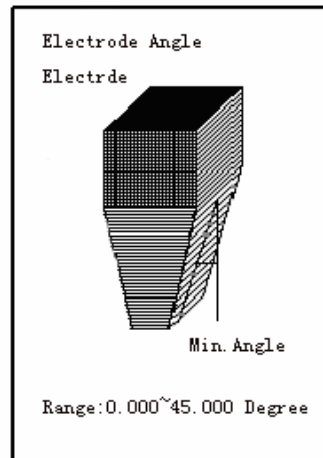


Fig 6.1.9

The “Angle” refers to here as the gradient of the electrode side surface, There are different means of the gradient according to the different electrode shapes:

- (1) Min. gradient

It represents the gradient of the electrode of the column shape.

- (2) Rib gradient

It represents the gradient of the electrode of the rib shape.

- (3) Cone gradient

It represents the gradient of the electrode of the cone shape.

If the side surface of the electrode is parallel to the Z axis, the parameter is set at “0”. If any angle is formed between the electrode side surface and the Z axis, input the angle

- LORAN

Press the [LORAN] input button, the screen displays the help scene at the left bottom as Fig 6.1.10

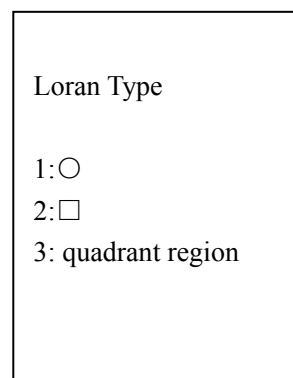


Fig 6.1.10

LORAN refers to here as the relative orbiting movement of the electrode (in the X-Y Plane) during the cutting process, There are three types of LORAN to be selected as follows:

- (1):○(The electrode performs its LORAN operation on the X-Y plane in a circular type.)
- (2):□(The electrode performs its LORAN operation on the X-Y plane in a circular type.)
- (3): Quadrant region(The electrode performs its LORAN operation each quadrant region)

The No. of LORAN type can be input by pressing the digit button.

3. Reference position

Press the [Refr. Pos.] input button, the screen displays the help scene at the left bottom as Fig 6.1.11

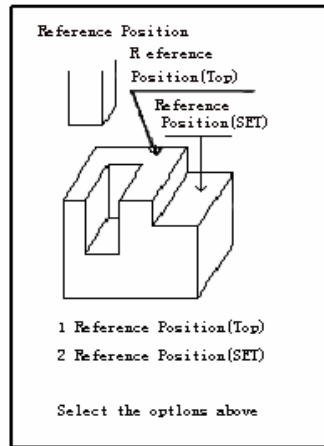


Fig 6.1.11

This parameter is used to select whether to set the reference position on the of the work piece or any other selected position. The selectable items are as follows:

- (1) Reference position (Top)

Reference position is set on the work piece top

- (2) Reference position (SET)

Reference position is set on the arbitrary position.

4. Cutting depth Z and Actual depth

Select the [Cutting Depth] button ,the screen displays the help scene as Fig.6.1.12

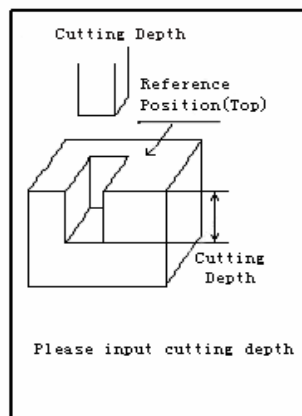


Fig 6.1.12

Input the cutting depth data as a coordinate value of the Z axis in the absolute coordinate system. Input the data with(+)or(-) assed to it depending on whether the cutting end position is above or

below the reference position ($Z=0$), respectively. When the reference position is set in “SEL” position the data represents the depth from the reference position, while the “Depth” corresponds to the depth from the cutting start position.

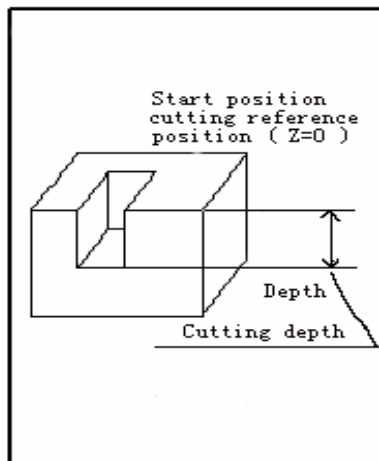
NOTE When the reference position is set on the “Top” side. No “Actual depth” can be Input, (The “cutting depth Z” is equal to the “Actual depth”) When the reference Position is set on the “SEL” side, input the parameter “Actual depth”.

INFORMATION ABOUT THE REFERENCE POSITION

◆ Reference position

The “Reference position” refers to position on the Z axis set as reference for machining (or the position set to $Z = 0$). In the case of the series, the top of a work piece is generally used as a reference position for its machining ($Z=0$), which, however, can be set at any selected point (height).

- Parameter setting when the “Reference position” is set at “Top”



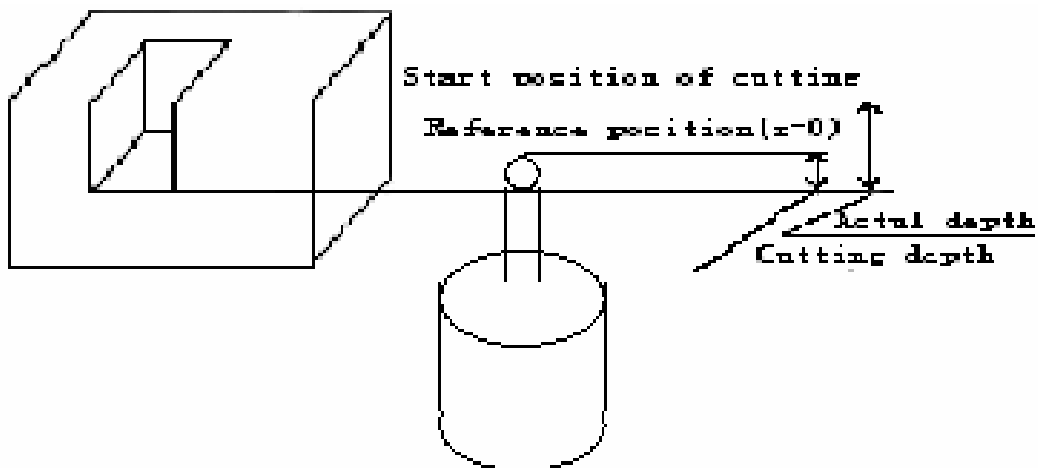
Cutting depth: This represents the distance from the reference position to the cutting end position. Set the parameter by inputting the Z coordinate value of the cutting end position in the “Absolute”.

Actual depth : this represents the distance from the work piece top to the cutting end position. Set the parameter by inputting.

The absolute value of the depth (distance).

When the “Reference position” is set at “Top”, the data input in the parameter “Cutting depth” becomes equal to that input in the parameter “Actual depth”, (The Input of data in the parameter “Cutting depth” makes it impossible to input any data in the parameter “Actual depth”.)

- Parameter setting when the “Reference position” is set at “SEL”



As one of examples in which the “Reference position” is set at any position of the work piece other than its top, there is a case where a reference sphere is used to set the reference position for its cutting as shown in the above drawing .In this case, set the parameter

“Reference position” on “SEL” ,side and input proper data in each of the parameter “cutting Depth” and “Actual depth”.

- Finish roughness

After pressing the [Fin. Rou.]the screen display the help scene at the left bottom as Fig.6.1.13

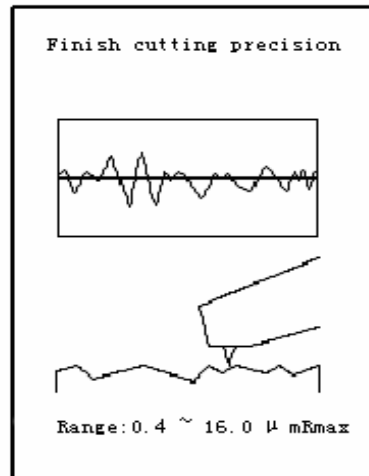


Fig 6.1.13

The range of the finish roughness value to be input is different according to the different input data of the electrode materials ,the work piece materials and the projected area. Please input the demand finish roughness according to the input range displayed in this help scene.

[Example of assist cutting operation]

Use such an electrode and the work piece as illustrated in Fig.6.1.14 for assist cutting of the work piece in 1-electrode mode.

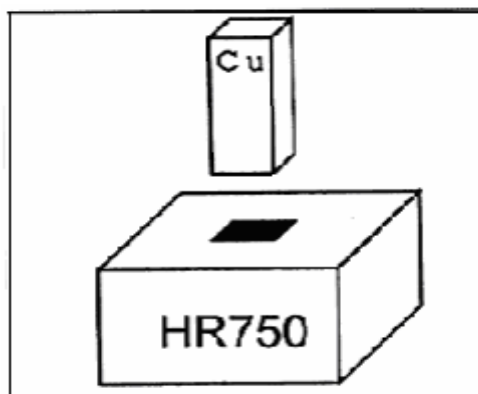


Fig 6.1.14

The condition and demands:

- Use the ATC and the tool No. is 5.
- The shape of the electrode is square column. The cutting is to be performed form rough cutting to finish cutting with only one electrode.
- The bottom area of the electrode is of 100mm²
- The electrode wear is to be set at 300μm.

- The copper is selected as the electrode material
- The work piece material is HR750.
- The reference position is set the top of the work piece .The cutting dept Z is equal to -3.00mm.
- The final finish roughness is set as 10 μ m Rmax.

Steps:

- 1) Move the electrode to the upper of the work piece.
- 2) Set the top of work piece as zero in the approach face alignment mode.
- 3) Press the [ATC] input button, then press the digit button [1] (1:use)
- 4) Press the [Elec. Mtr.] input button ,and then press the digit button[1] (1:Cu)
- 5) Press the [Work Mtr.] input button ,and then press the digit button[5] (1:HR750)
- 6) Press the [Elec.Shape.] input button ,and then press the digit button[1] (1:column shape)
- 7) Press the first button of [Tool No.],then press the digit [5].(The tool No. is 5)
- 8) Press the [Wear] input button corresponding to the electrode number [1],then press [3],[0],[0] (The electrode [2],[3] is not used)
- 9) Press the [Proj. Area] input button, then select the “Top”.
- 10) Press the [Refr. Area] input button, then select the “Top”.
- 11) Press the [Cutting Depth] input button, then press the digit keys [-],[3],[.],[0],[0],[0].
- 12) Press the [Fin. Rou.]button, then press the digit keys [1],[0],[.],[0].

After pressing the [ENT] key, the system starts automatically to the assist cutting.

6.2 Single cutting

The single cutting refers to the operation of determining automatically the finish cutting conditions according to the specified conditions (electrode material, electrode shape, electrode wear, roughness) and performing the cutting process only according to the finish cutting condition.

Select the [Assist] mode from the cutting types, and the select the [Single] sub mode button.

The screen change to the single scene as Fig 6.2.1

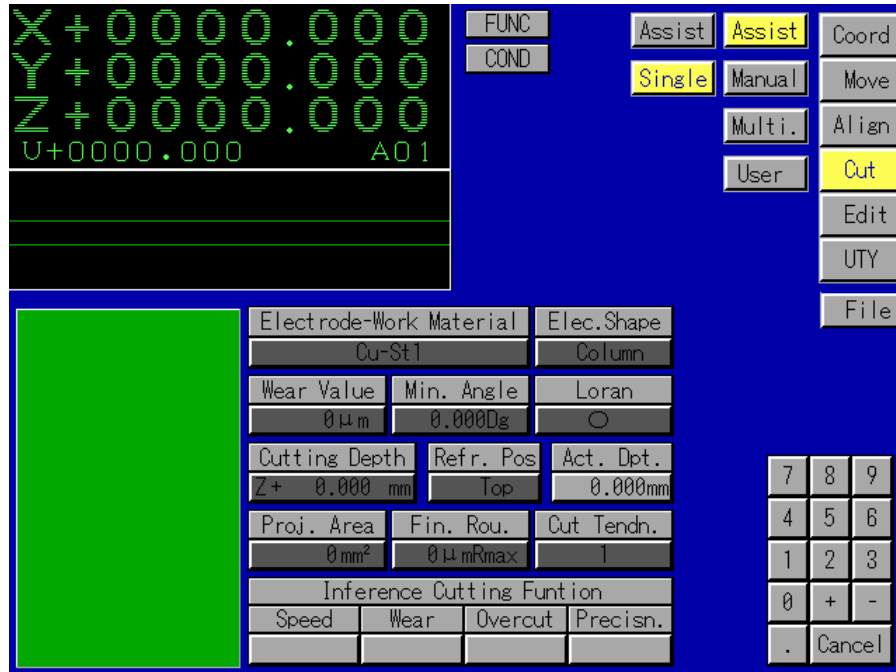


Fig 6.2.1

There are following input items in the single cutting screen:

- Electrode materials

This parameter is used to input the electrode materials. After pressing the [Elec. Mtr.] input item button, the screen display the help scene at the left bottom as Fig 6.2.2

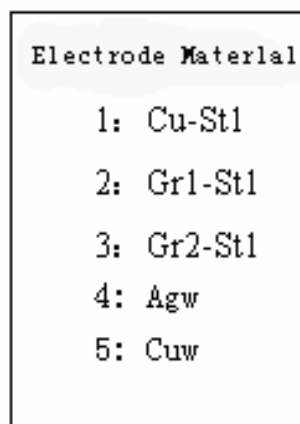


Fig 6.2.2

The selectable electrode materials are as follows:

- (1) CU: copper electrode
- (2) Gr1: graphite (ED3, ISQ-63) electrode
- (3) Gr2: graphite (POCP,EDM-3) electrode
- (4) AgW: silver and tungsten electrode
- (5)CuW: copper and tungsten electrode

The No. of the electrode materials are input by pressing the digit buttons.

- Work piece material

This parameter is used to select the work piece materials, After pressing the [Work Mtr] Input item button, the screen displays the help scene at the left bottom as Fig 6.2.3

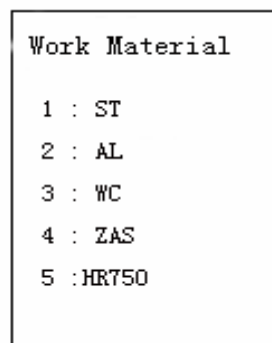


Fig.6.2.3

The selectable work piece materials are as follows:

- (1) St: Steel (SKD61, NAK80, STAVAX, PD555)
- (2) Al: Aluminum alloy
- (3) Wc: Tungsten and carbon steel
- (4) ZAS: Sub-aluminum alloy
- (5) HR750: Copper alloy

The No. of the work piece materials are input by pressing the digit buttons.

Some work piece materials could not be selected according to the different electrode materials.

NOTE

- Electrode shape

This parameter is used to select the electrode shape. After pressing the [Elec.Shape] button, the help scene will be displayed at left bottom of the screen as Fig.6.2.4

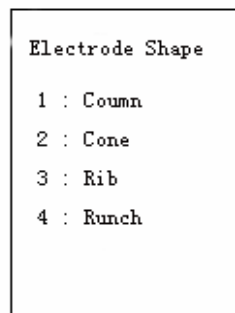


Fig.6.2.4

There are four electrode shapes as follows:

- (1) Column shape (Square and circular columns are included in the type)
- (2) Rib shape (This type is selected for an electrode, the projected area of which is flat in Terms of its aspect ratio)
- (3) Cone shape (Square and circular cones are included in this type, in this case, the Projected area will change during the cutting process.)
- (4) Punch (This type of electrode is selected for producing a punch referred to as “Reverse Discharge”,. In this case, a work piece being produced is installed on the Machine head while an electrode is placed in a cutting table.)

The No. of electrode shapes are input by pressing the digit buttons.

“Information about the electrode shape” in the first section of Chapter 6

REFERENC [Assist cutting].

- Electrode wear

The No. of electrode using for the single cutting is 1.

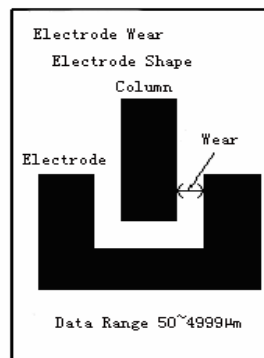


Fig.6.2.5

Input the electrode wear value in the electrode wear input item.

- Min. gradient angle

Press the [Min. Angle] input button, the screen display the help scene at the left bottom.

As Fig.6.2.6

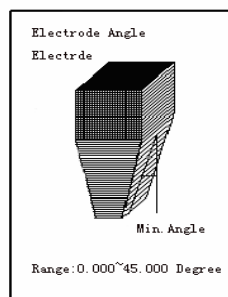


Fig.6.2.6

The “Angle” refers to here as the gradient of the electrode side surface. There are different means of the gradient according to the different electrode shapes.

- (1) Min. gradient

It represents the gradient of the electrode of the column shape.

(2) Rib gradient

It represents the gradient of the electrode of the rib shape.

(3) Cone gradient

It represents the gradient of the electrode of the cone shape.

If the side surface of the electrode is parallel to the Z axis, The parallel is set at "0". If Any angle is formed between the electrode side surface and the Z axis, input the angle.

● LORAN

Press the [LORAN]input button, the screen displays the help scene at the left bottom as Fig.6.2.7

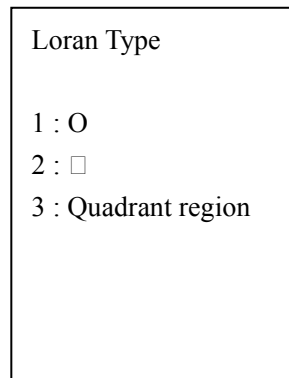


Fig.6.2.7

LORAN refers to here as the relative orbiting movement of the electrode (in the X-Y Plane) during the cutting process, There are three types of LORAN to be selected as follows:

- (1) O (The electrode performs its LORAN operation on the X-Y plane in a circle type.)
- (2) □ (The electrode performs its LORAN operation on the X-Y plane in a square type.)
- (3) Quadrant region (The electrode performs its LORAN operation in each quadrant region.)

The No. of LORAN type can be input by pressing the digit button.

● Reference position

Press the [Refr. Pos.]input button, the screen displays the help scene at the left bottom as Fig 6.2.8

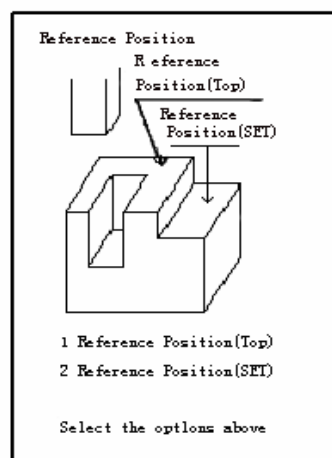


Fig.6.2.8

This parameter is used to selected whether to the reference position on the top of the work piece or any other selected position .The selectable items are as follows:

(1) Reference position (Top)

Reference position is set on the work piece top,

(2) Reference position (SEL).

- Cutting depth Z and Actual depth

Select the [Cutting depth] input button, the screen displays the help scene at the left bottom as Fig.6.2.9

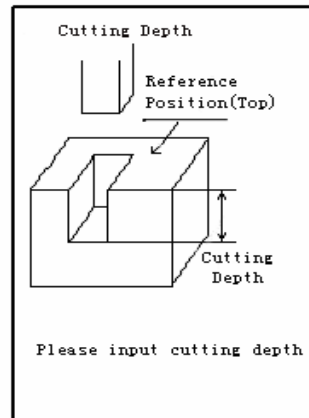


Fig.6.2.9

Input the cutting depth as a coordinate value of the Z axis in the absolute coordinate system. Input the data with (+) or (-) added to it depending on whether the cutting end position is above or below the reference position ($Z = 0$) respectively. when the reference position is set in “SEL” position, the “Cutting depth Z” data represents the depth from the reference position, while the “Actual depth” data corresponds to the depth from the cutting start position.

NOTE! When the reference position is set on the “Top” side, no”Actual depth” can be input.(The Cutting depth Z” is equal to the “Actual depth”).when the reference position is set on the “SEL” side ,input the parameter “Actual depth”.

REFERENCE “Information About the reference position” on the first section of Chapter 6
[Assist cutting]

- Projected area

If the [Proj. Area] input item has been selected, the screen displays the help scene at the Left bottom as Fig.6.2.10

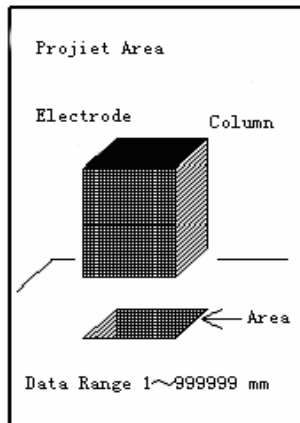


Fig.6.2.10

Input the projected area.

The means of the projected area mentioned here is that as follows:

- (1) It is the bottom area when the electrode is in the column shape:
- (2) It is the projected area in the electrode cutting direction when the electrode is in the rib shape or in the cone shape.(However, the electrode is the cutting part).
- (3) It is the EDM discharge area (the area of in X-Y plane) when the electrode is the punch.

NOTE! The bottom hole area should be subtracted from the projected area when there is a bottom hole in the cutting electrode.

“information about the projected area” in the first section of Chapter 6

REFERENCE [Assist cutting]

- Finish roughness

After pressing the [Fin. Rou.] input button, the screen displays the help scene at the left bottom as Fig.6.2.11

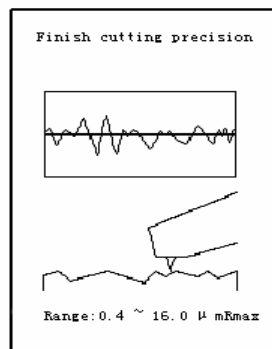


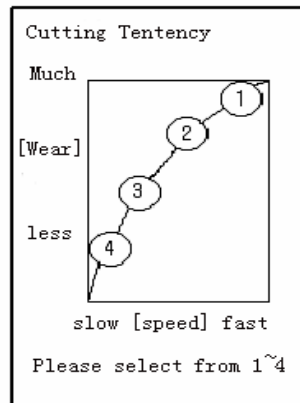
Fig.6.2.11

The range of the finish roughness value to be input is different according to the different Input data of the electrode materials, the work piece materials and the projected area.

Please input the demand finish roughness according to the input range displayed in the help scene.

- Cutting tendency

Press the [Cut Tendon.] button, the screen displays the help scene at left bottom as 6.2.12



6.2.12

Input the No. of selection according to the single cutting tendency inference prompted in this scene.

After inputting the cutting tendency. the values of the [Cut speed],the [Wear],the [Over cut] and the [Roughness] will be displayed in the [Inference cutting property].

- ATC

After pressing the [ATC] input item, the screen displays the help scene at the left bottom. as Fig6.2.13

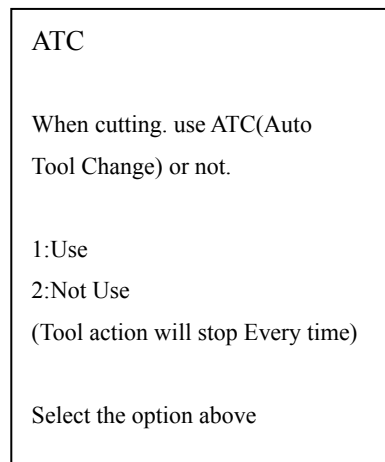


Fig 6.2.13

The select state of ATC is as follows.

(1) Use

(2) Not Use

Press digit key to select the No.

- Tool No.

The 1 electrode is used in the single cutting. When the ATC item is selected as “Use”, input the tool No. corresponding to the electrode to be used.

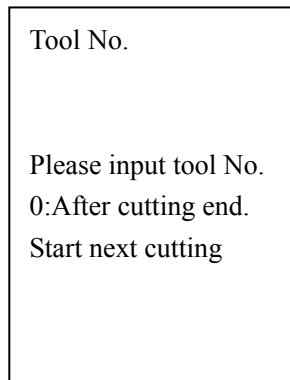


Fig.6.2.14

[Example of single cutting operation]

Use such an electrode and the work piece as illustrated in Fig 6.2.15 for single cutting

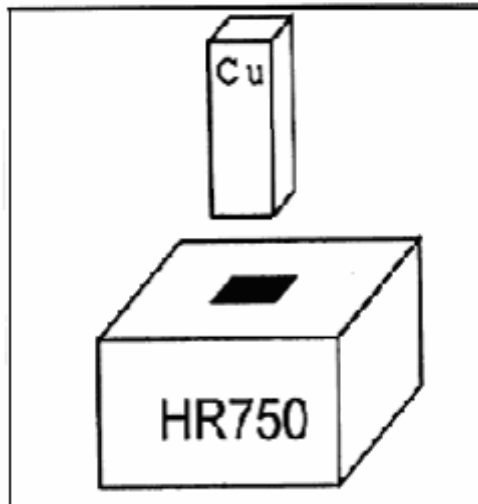


Fig.6.2.15

The conditions and demands:

- The shape of the electrode is square column (The electrode side surface is parallel to the cutting direction).
- The bottom area of the electrode is of 100mm²
- The electrode is set in the machine head.
- The electrode wear is to be set at 300μm.
- The copper is selected as the electrode material.
- The work piece material is HR750.
- The reference position is set on the top of the work piece, the cutting depth Z is equal to -3.000mm.
- The final finish roughness is set at as 10μmRmax.
- Use the ATC and the tool No. is 5.

Steps:1): Move the electrode to the upper of the work piece.

2): Set the top of work piece as zero in approach face alignment mode.

3): Press the [ATC] input button, then press the digit key [1].(1:use)

4): Press the [Tool No.] input button, then press the digit key [5] (The tool No. is 5)

5): Press [Elec. Mtr.] input button, then press the digit button [1] (1:Cu)

6): press the [Work Mtr.] input button then press the digit button [5] (5:HR750).

- 7): Press the [Elec. Shape] input button, then press the digit button [1](1:column shape).
- 8): Press the [Elec. Wear] input button, then press [3],[0],[0].
- 9): Press the [Min. Angle] input button, then press the digit buttons [0],[.],[0],[0],[0].
- 10): Press the [LORAN] input button, then press the button [1].
- 11): Press the [Cutting depth] input button, then press the [-],key and the digit keys [3].
[.],[0],[0],[0].
- 12): Press the [Refr Pos.] input button, then select the “top”.
- 13): Press the [Proj. Area] input button, then press [1]0],[0].
- 14): Press the [Fin. Rou] button, then press the digit keys [1],[0],[.],[0].
- 15): Press the [Cut Tend.] button, then press the digit key [1].

After press the [ENT] key, the system will perform automatically the single cutting.

Chapter7 MANUALLCUTTING

The manual cutting refers to the cutting method of determining all of the parameters such as cutting condition, No. of cutting by operators.

The manual cutting of the Z system includes following four cutting sub modes.

- Line cutting
- Arc cutting
- Thread cutting
- Spiral cutting
- PIKA 10 cutting

Press the [Manual] button from the cutting mode selection keys in the [Cut] mode, the screen will change to the manual cutting, and there are five different cutting sub modes being selectable. They are [Line],[Arc],[Thread],[Spiral]and [PIKA10]. There are detail scenes corresponding to the [Line],[Arc]and [PIKA10] to perform the more detail setting of there cutting.

7.1 Line Cutting

“Line Cutting” refers to the manual cutting of the cutting, conditions and the No. of cutting of the line cutting are input by hands.

There is the detail scene corresponding to the line cutting to do more detail setting of this Cutting..

7.1.1 Line cutting scene

Press the [Line]button from the five manual cutting sub modes, the screen displays the Manual line cutting scene as Fig.7.1.1.1

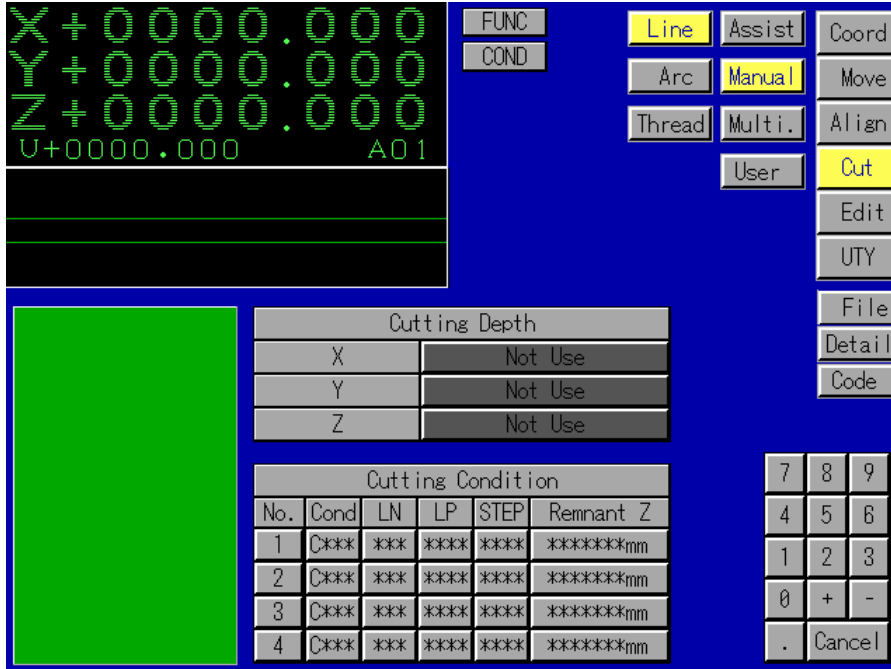


Fig.7.1.1.1

There are following input items in the screen:

- Cutting depth

This parameter is used to select the axis on which to move the electrode for the line cutting in X, Y, Z direction and input the cutting depth by pressing the corresponding input button of the three axes. The initial display of this item is "Not Use"

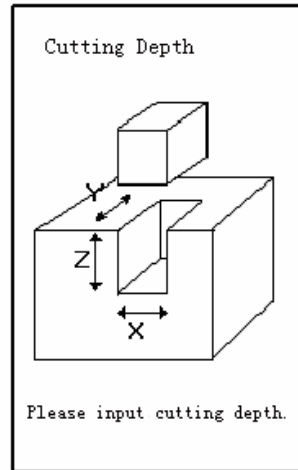


Fig.7.1.1.2

- Specify the cutting condition o No. of cutting

The No. of cutting is the display item. The maximum No. of cutting in the line manual cutting (not detail) screen is 4.

NOTE! The actual No. of cutting is determined according to the input No. of cutting condition.

- Specify the cutting condition o Condition

Press the [Cond] button, the screen displays the help scene at the left bottom as Fig.7.1.1.3

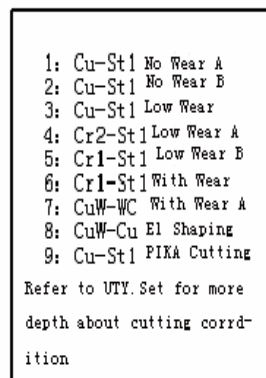


Fig.7.1.1.3

Input the No. of cutting condition directly.

- Specify the cutting condition o LN (LOP, AN type)

Press the [LN] button, the screen displays the help scene at the left bottom as Fig.7.1.1.4

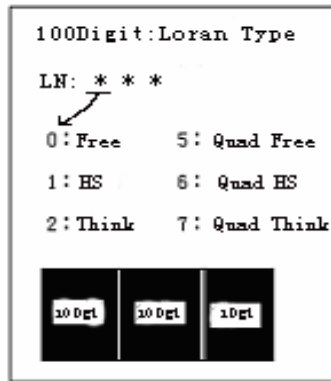


Fig7.1.1.4

If you want to know the means of the each LN type, you can press the [100Dgt], [10Dgt], [1Dgt] buttons in the help screen.

Input the No. of LN type by digit key.

- Specify the cutting condition of LP (the selection of quadrant region LN types)
After pressing the [LP] button, the screen displays the help scene at the left bottom as Fig7.1.1.5

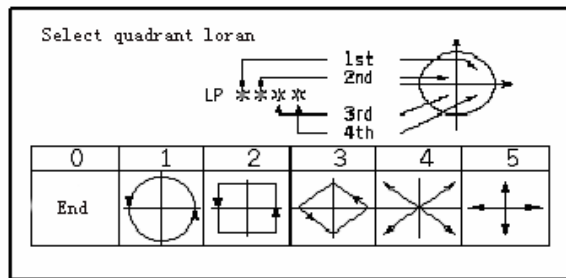


Fig7.1.1.5

The LP parameter is used to divide the plane of LN operation into four parts and use the four types LN shape. Input the No. of LP reference to the help scene.

- Specify the cutting condition of STEP (scope of step)
After pressing the [STEP], the screen displays the help scene at the left bottom as Fig7.1.1.6

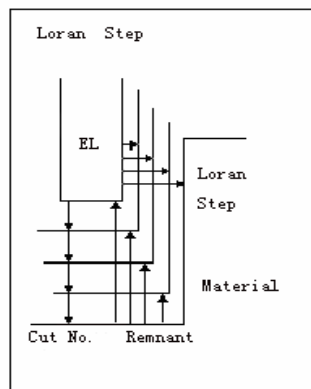


Fig7.1.1.6

The parameter "STEP" is used to set the scope of the plane on which to perform the LOPAN operation. When calculating the scope of STEP the selected over cut of cutting condition should be considered completely, and then input the scope of the step.

- Specify the cutting condition o Bottom Remnant of Z axis

After pressing [Remnant Z] button, the screen displays the help scene at the left bottom as Fig.7.1.1.7

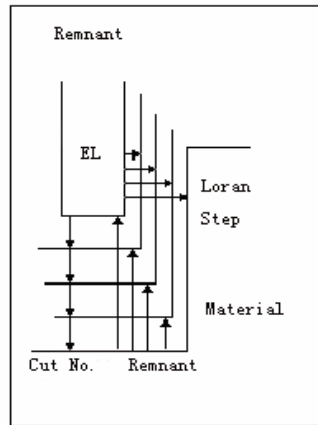


Fig.7.1.1.7

The "Bottom Remnant Z" is used to set the difference between the specified cutting depth and the cutting depth by electrode cutting. Please consider completely the selected over cut, and then calculate and input the bottom remnant.

If you want to set more detail line cutting, press the [Detail] key to enter to the line cutting detail scene.

7.1.2 Line cutting detail scene

Press the [Detail] button from the line cutting screen, the line cutting detail scene as Fig.7.1.2.1.

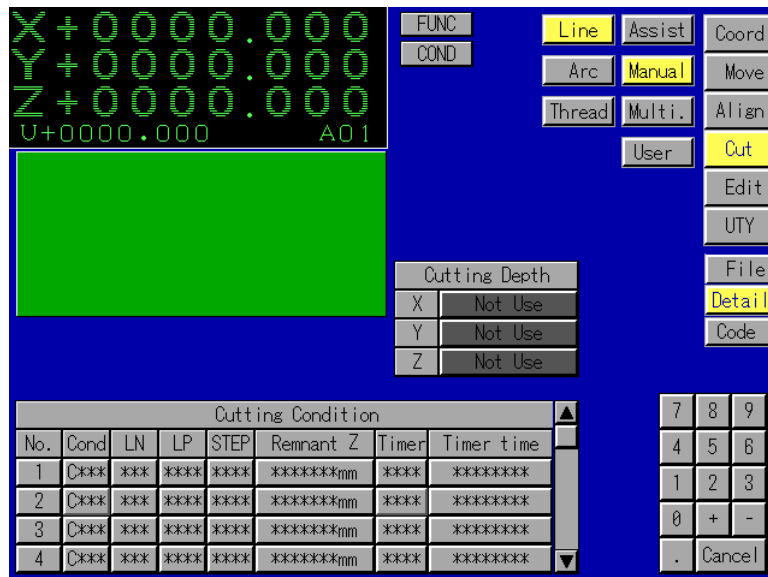


Fig7.1.2.1

There are following input items to be added or modified in the line cutting detail scene:

- ATC

Press the [ATC] input item, the screen displays the help scene at the left bottom as Fig7.1.2.2

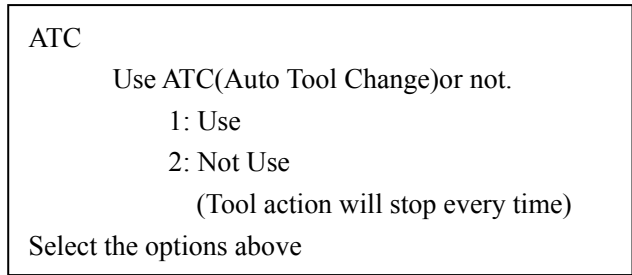


Fig.7.1.2.2

- The select state of ATC is as follows.

(1)Use

(2) Not Use

Press the digit keys to select the No.

- Tool No

When the ATC is selected as "Use", input the tool No. corresponding to the electrode to be used.

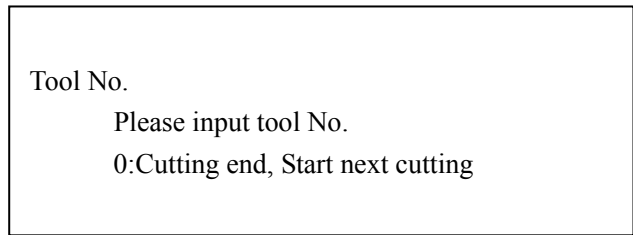


Fig7.1.2.3

- Specify the cutting condition No. of cutting

The No. of cutting is still the display item. However, the maximum No. of cutting increases to 8 comparing with common case. (The No. of cutting in the screen is 4, it be added in use rolling bar.)

NOTE! The No. of cutting should be determined according to the input No. of cutting condition.

- Specify the cutting condition of Timer

Press the [Timer] button, the screen displays the help scene at the left bottom as Fig7.1.2.4

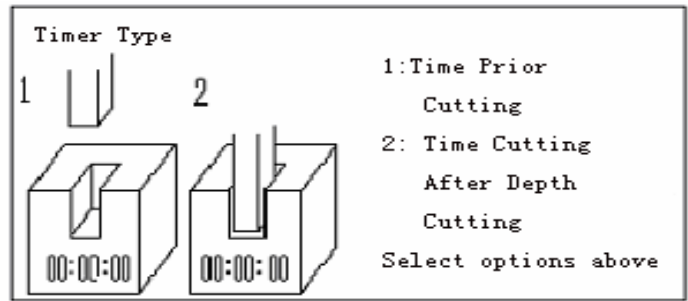


Fig7.1.2.3

Time cutting includes:

(1) Time priority cutting

In this case, the Time cutting is performed for the time duration set by the parameter "Time (Timer cutting time)" from the time when the cutting is started. If the cutting operation is executed to the set depth before the set time is reached, it is ended at this point.

(2) Time cutting after depth cutting

After the cutting operation is executed to the set depth, the time cutting is performed in the time duration set by the parameter "TIME (Time cutting time)". Please select and input the No. of type of "TIME".

● Timer cutting time

After pressing the [Timer Time], the screen displays the help scene as

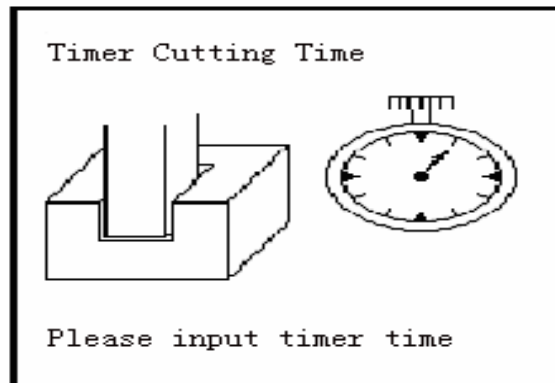
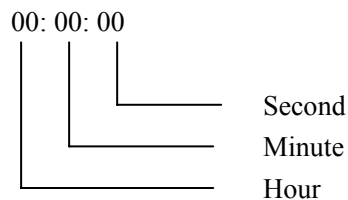


Fig7.1.2.5

Then input the time of Timer cutting.

[Example of time input operation]



Input the cutting time by pressing the digit key, If the cutting time is one minute and twenty seconds, press the digit keys [1], [2], [0].

7.1.3 Line cutting Code

If [Code] button is pressed, a NC file which contains automatic processing technological parameters list of line cutting will be created. As shown in Fig.7.1.3.1, after pressing [Code] button, user can name the NC file within 8 characters and save it or cancel operation.

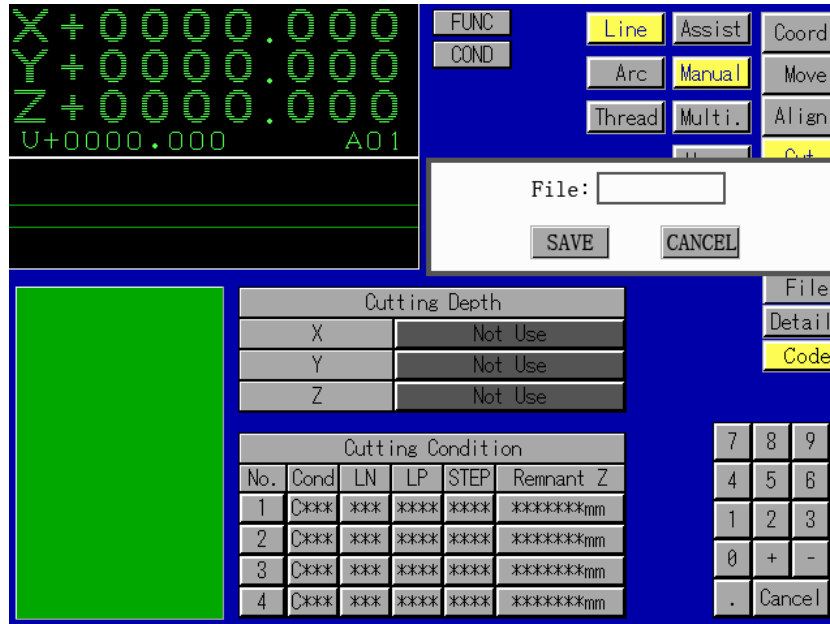


Fig.7.1.3.1

7.2 Arc Cutting

“Arc Cutting” refers to the manual cutting of the cutting path along a arc. All of the cutting conditions and the No. of cutting of the arc cutting are input by hands.

7.2.1 Arc cutting scene

Press the [Arc] button from the five manual cutting sub modes, the screen displays the **manual arc cutting scene** as Fig.7.2.1.1.

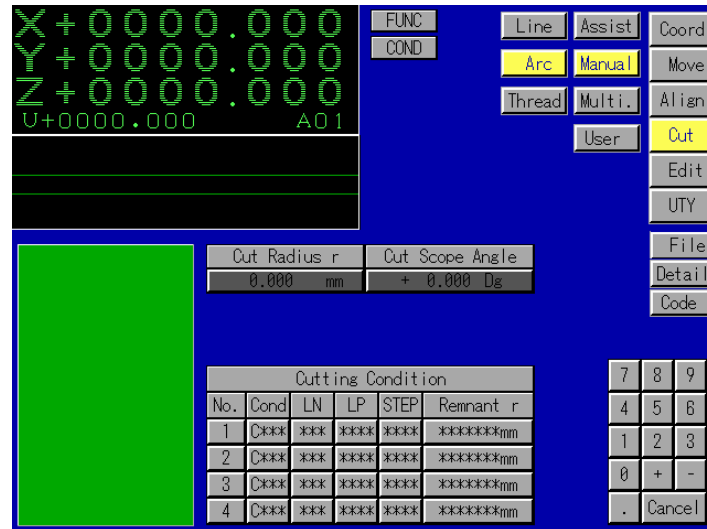


Fig. 7.2.1.1

There are following input items in this screen:

- Cutting scope angle

It is the angle range of the arc manual cutting. Its unit is degree.

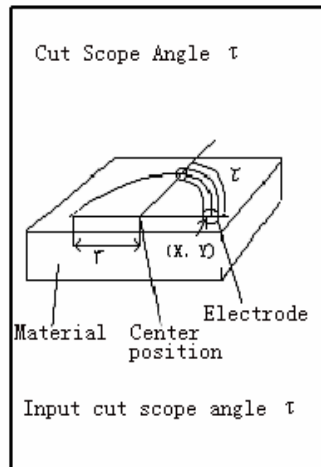


Fig7.2.1.2

- Cutting radius r

It is the cut arc radius.

- Specify the cutting condition o No. of cutting

The No. of cutting is the display item. The maximum No. of cutting is 4 in the arc manual cutting (not detail) scene. Set the corresponding cutting condition near by No. of cutting.

NOTE! The actual No. of cutting is determined according to the input No. of cutting condition.

- Specify the cutting condition o Condition

Press the [Cond] button, the screen displays the help scene at the left bottom as Fig7.2.1.3.

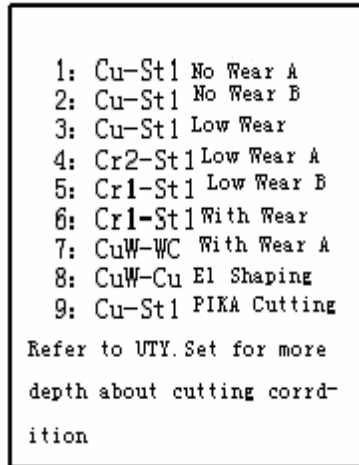


Fig7.2.1.3

Input the No. of cutting condition directly.

- Specify the cutting condition o LN (LORAN type)

Press the [LN] button, the screen displays the help scene at the left bottom as Fig7.2.1.4

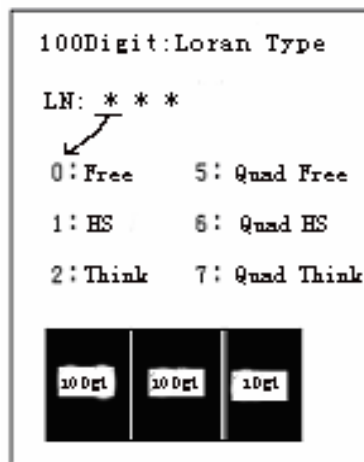


Fig7.2.1.4

If you want to know the means of the each LN, you can press the [100Dgt], [10Dgt], [1Dgt] buttons.

Input the No. of LN by digit key.

- Specify the cutting condition o LP (the selection of quadrant region LN)

After pressing the [LP] button, the screen displays the help scene at the left bottom Fig.7.2.1.5

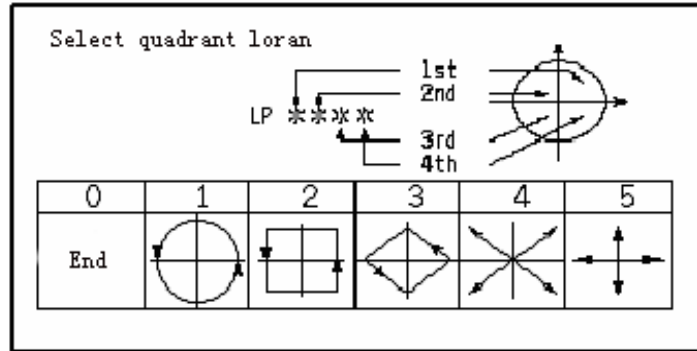


Fig7.2.1.5

If you want to Know the means of the each LN, you can press the [100Dgt], [10Dgt], [1Dgt] buttons.

Input the No. of LN by digit key.

- Specify the cutting condition o STEP (scope of step)

After pressing the [STEP], the screen displays the help scene at the left bottom as Fig7.2.1.6

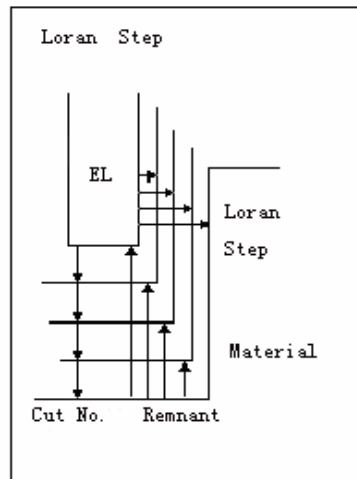


Fig7.2.1.6

The parameter "STEP" is used to set the scope (size)of the plane on which to perform the LOP AN operation. When calculating the scope of STEP, the selected over cut of cutting condition should be considered completely, and then input the scope of the step.

- Specify the cutting condition o Remnant of radius

After pressing the [Remnant τ] button, the screen displays the help scene at the left bottom as Fig.7.2.1.7.

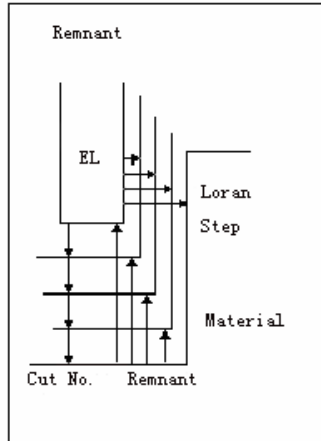


Fig7.2.1.7

The “Remnant of radius” is used to set the difference between the specified radius and the cutting radius. Please consider completely the selected over cut, and then calculate and input the remnant of radius.

If you want to set more detail arc cutting, press the [Detail] key to enter to the arc cutting detail scene.

7.2.2 Arc cutting detail scene

Press the [Detail] button from the arc cutting screen, the screen displays the arc cutting detail scene as Fig7.2.2.1

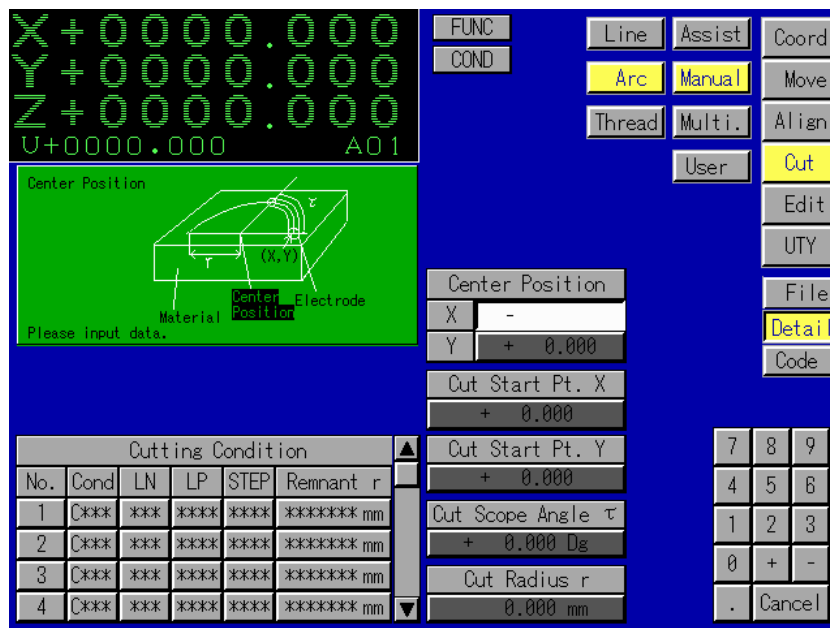


Fig7.2.2.1

There are following input items to be added or modified in the arc cutting detail screen:

- ATC

Press the [ATC] input item, the screen displays the help scene at the left bottom Fig7.2.2.2

ATC

Use ATC(Auto Tool Change)or not.

1: Yes

2: No (EL action will stop every time)

Select the options above

Fig.7.2.2.2

- The select state of ATC is as follows.
 - (1) Use
 - (2) Not Use
 Press the digit keys to select the No.
- Tool No

When the ATC is selected as "Use", input the tool No. corresponding to the electrode to be used.

Tool No.

Please input tool No.

0: Cutting end, Start next cutting

Fig7.2.2.3

- Center position X,Y

It is the center X, Y coordinates of cut arc.
- Cutting start point X

It is the X coordinate of start point of cut arc
- Cutting start point Y

It is the Y coordinate of start point of cut
- Specify the cutting condition o No. of cutting

The No. of cutting is still the display item. However, the maximum No. of cutting increases to 8 comparing with common case. (The No. of cutting in the screen is 4, it be added by using rolling bar.)

NOTE! The No. of cutting should be determined according to the input times of cutting condition

7.2.3 Arc cutting Code

If [Code] button is pressed, a NC file which contains automatic processing technological parameters list of arc cutting will be created. As shown in Fig.7.2.3.1, after pressing [Code] button, user can name the NC file within 8 characters and save it or cancel operation.

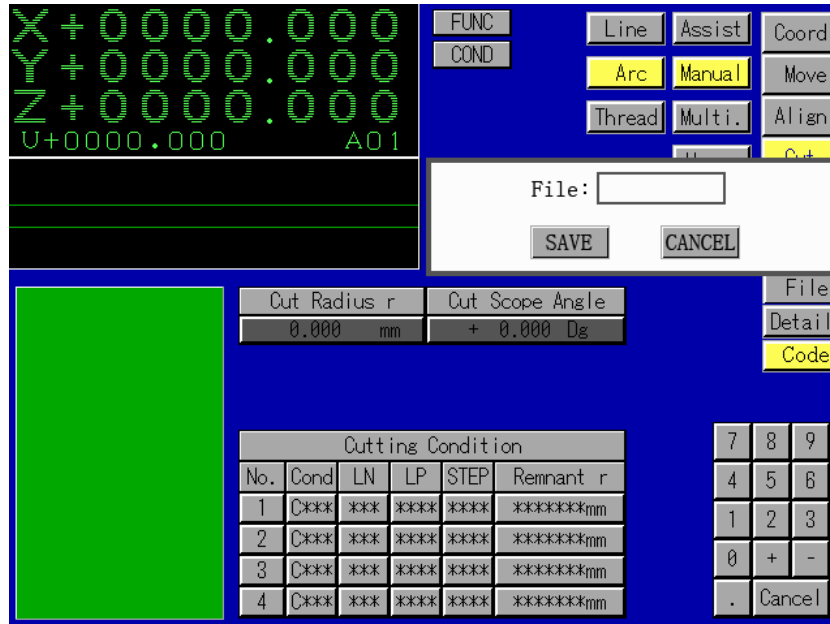


Fig.7.2.3.1

7.3 Thread Cutting (Z, U axis)

The "Thread Cutting" refers to the cutting mode provided for thread cutting with the which has its machine head installed a U axis (the rotation axis with R head or property of angle calculation), This cutting mode cannot be used for any machine without a U axis on its machine head.

Press the [Thread] button from the five sub mode buttons of the manual cutting, the screen displays the thread cutting scene as Fig.7.3.1.

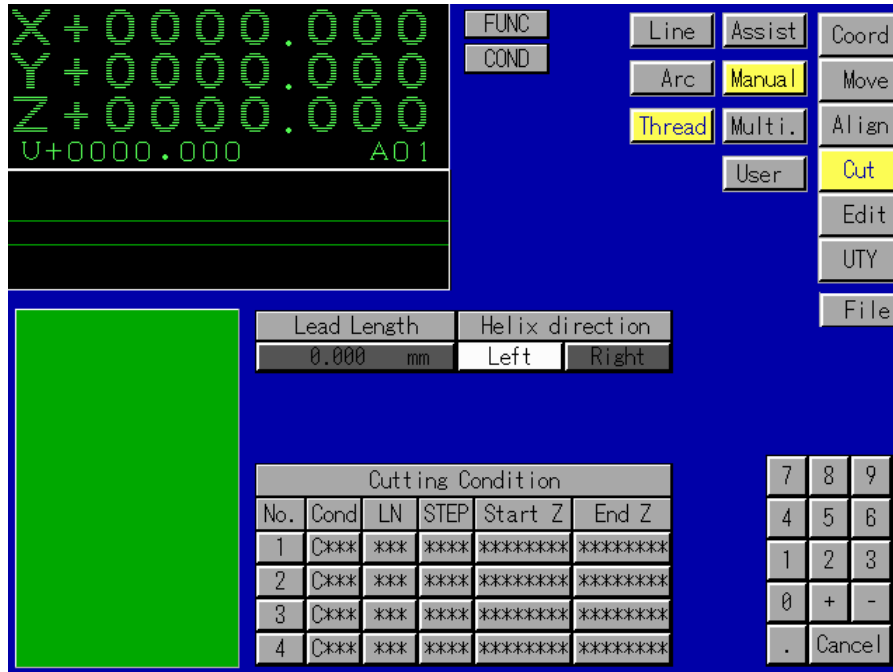


Fig.7.3.1.

There are following input items in this scene:

- Helix direction

There are two types helix direction.

(1) Right

In this case, the thread cutting is performed with the U axis rotating around the Z axis the clockwise direction when the Z axis is viewed from above. (The electrode moves the U axis in the "-" direction for the cutting.)

(2) Left

in the case, the thread cutting is performed with the U axis rotating about the Z axis the counterclockwise direction when the Z axis is viewed form above. (The electrode moves on the U axis in the "+" direction for the cutting.

Please select the helix direction reference to the help scene as Fig.7.3.2. Select the [Left] and [Right].

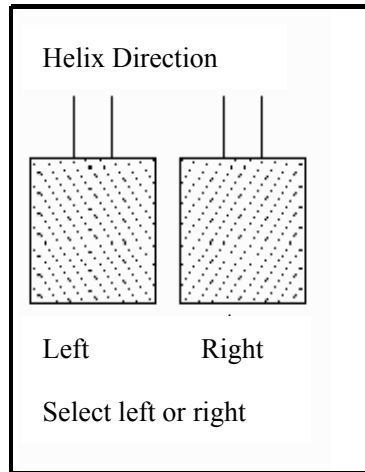


Fig7.3.2

- Lead length

Press the [Lead length] input item, the screen displays the help scene as Fig7.3.3

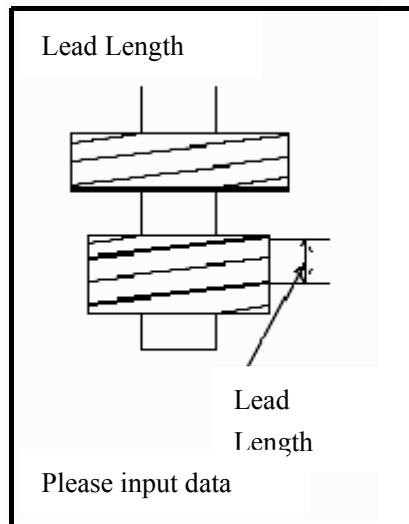


Fig7.3.3

Input the lead length (the cutting length in Z axis for each rotation of the U axis)

- Specify the cutting condition o No. of cutting

It is the display item. The maximum No. of cutting is 4 in the thread cutting scene. Set the corresponding cutting condition near by No. of cutting.

NOTE! The actual No. of cutting is determined according to the input No. of cutting condition.

- Specify the cutting condition o Condition

Press the [Cond] button the screen displays the help scene at the left bottom as Fig7.3.4

```

1: Cu-St1 No Wear A
2: Cu-St1 No Wear B
3: Cu-St1 Low Wear
4: Cr2-St1 Low Wear A
5: Cr1-St1 Low Wear B
6: Cr1-St1 With Wear
7: CuW-WC With Wear A
8: CuW-Cu El Shaping
9: Cu-St1 PIKA Cutting

Refer to UTY. Set for more
depth about cutting corrd-
ition

```

Input the No. of cutting condition directly.

- Specify the cutting condition of LN (LORAN type)

Press the [LN] button, the screen displays the help scene at the left bottom as Fig 7.3.5

Loran Type	
LN***	
100Dgt	0 Free Loran
	1 HS Loran
	2 Z Loran
10Dgt	0 SERVO END
	3 Trk Ret To SRV
	6 Cnt Ret To SRV
1Dgt	0 Loran END
	1 Circle Loran

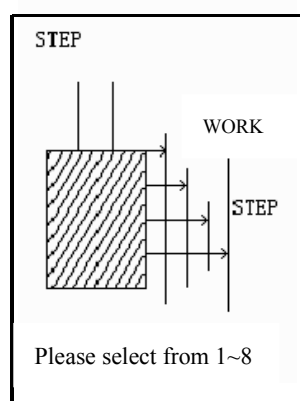
Please input data

Fig7.3.5

Input the No. of LN type by digit key.

- Specify the cutting condition of STEP (scope of step)

After pressing the [STEP], the screen displays the help scene at the left bottom as Fig.7.3.6



The parameter "STEP" is used to set the scope (size) of the plane on which to perform the LORAN operation. When calculating the scope of STEP, the selected over cut of cutting condition should be considered completely, and then input the scope of the STEP

- Specify the cutting condition of Cutting Condition Start Z
Press the [Start Z] input button, the screen displays the help scene at the left bottom as Fig 7.3.7

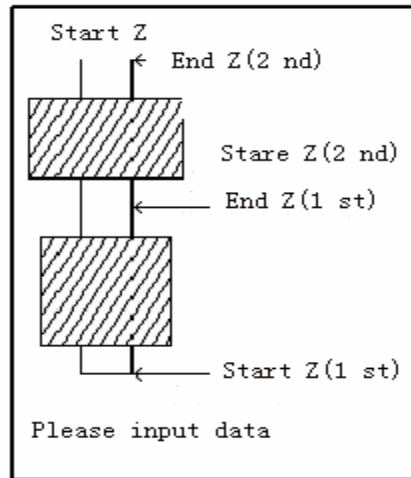


Fig.7.3.7

Input the position of Z axis when cutting starts.

- Specify the cutting condition of End Z
Press the [End Z] input item, the screen displays the help scene at the left bottom as Fig.7.3.8

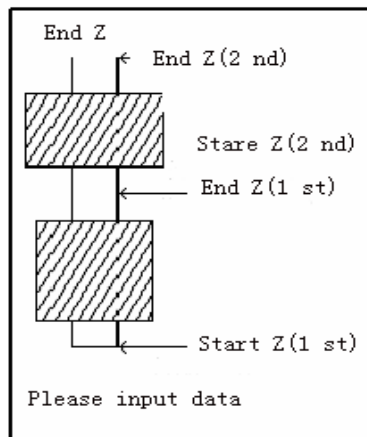


Fig.7.3.8

Input the position of Z axis when cutting ends.

Chapter 8 MULTIPLE CUTTING

The "Multiple Cutting" refers to the cutting mode which allows more than executed according to one cutting operation (one NC program). There are four types cutting sub modes in the "Multiple Cutting" mode:

- Random
- Lattice 1
- Lattice 2
- Circle

The cutting methods and alignment of each multiple cutting sub mode will be introduced follows

8.1 Random Multiple cutting

In this sub mode the multiple cutting can be performed at user-selected position. Select the [Multi.] mode in the [Cut] mode, then press the [Random] sub mode button from four cutting buttons of the [Multi.], the screen displays the random position multiple cutting scene as Fig.8.1.1

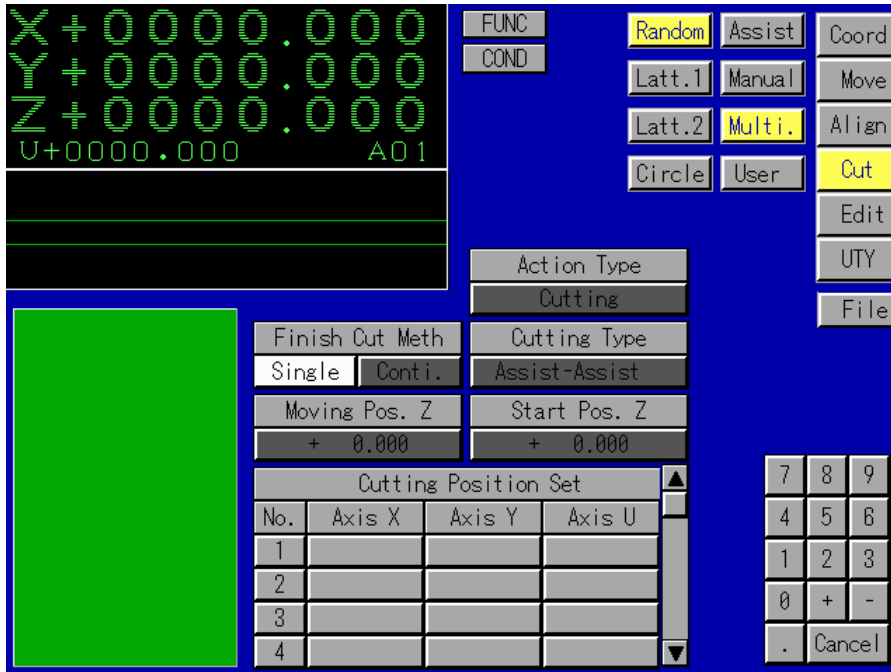


Fig.8.1.1

There are following input items on this scene.

- Cutting type

Press the [Cutting type] button, the screen displays the help scene at the left bottom as Fig.8.1.2

Cutting Type
 1: Ast.-Assist
 2: Ast.-Single
 3: Man.-Line
 4: Man.-Arc
 5: Man.-Thread
 6: Man.-Spiral
 7: User
 Select the option above

Fig.8.1.2

There are seven types of cutting can be used form the multiple cutting. Input the type No. of the cutting by the digit key.

- Action type

Press the [Action type] button, the screen display the help scene at the left bottom as Fig.8.1.3

Action Mode
 1: Cutting....Execute
 Cutting
 2: Dry.....Execute Dry
 Cutting
 3: Single Step.....Check
 Position
 Select the options above

Fig.8.1.3

There are three type of this cutting, Input the type No. of the cutting by the digit keys.

- Finish cutting method

Press the [Finish cut meth] button, the screen displays the help scene at the left bottom as Fig.8.1.4

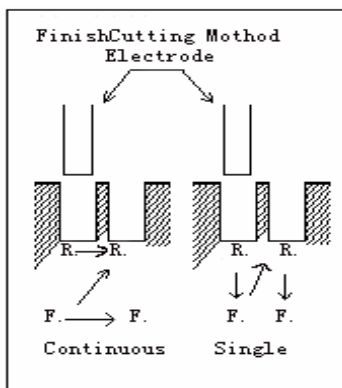


Fig.8.1.4

There two types of the finish cutting methods:

(1) Continuous

This cutting operation is proceeded from the rough cutting to the finish cutting for all holes.

(2) Single

This is hole cutting operation from the rough cutting to the finish cutting on one-by-one hole

Select the one type of the cutting from two types, then input its No..

- Moving position Z

This parameter is used to input the position of Z axis after it moving from current position to the next cutting position

- Start position Z

It is the position of the Z axis when the cutting starts.

- Cutting position set No. of holes

The No. of holes is not input item. The maximum No. of holes is 8 in the Z system.(The screen displays the 4 holes. It can be added by using rolling bar.)

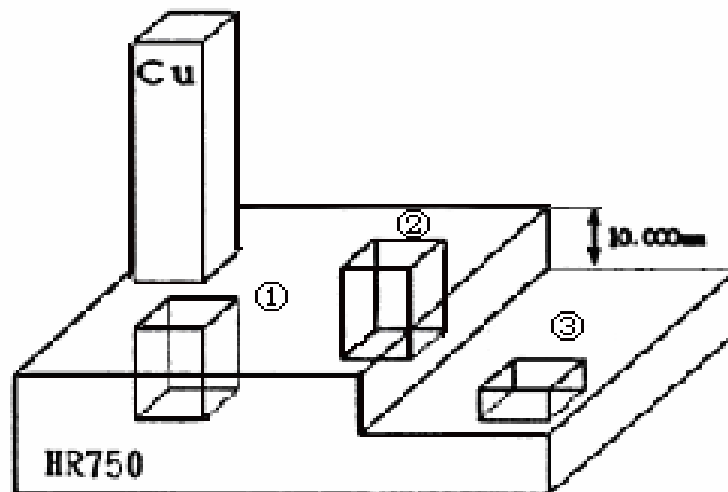
- Cutting position set X, Y, Z axis

It is the absolute coordinate to be input of X, Y, Z axes corresponding to the holes. Press the [ENT] key after inputting all items mentioned above, the system will perform automatically the selected position multiple cutting.

[Example of the random multiple cutting operation]

With cutting position ① as its origin point, perform the multiple cutting in (assist-assist cutting with 1 electrode) mode at the following three cutting positions. The cutting diagram is shown as

Fig.8.1.5



The cutting conditions and demands are as follows

- Cutting position:

① (X=0.000, Y=0.000, Z=0.000)

② (X=20.000, Y=20.000, Z=0.000)

③ (X=45.000, Y=0.000, Z=-9.000)

- Use the continuous finish cutting method
- The electrode to be used for the cutting is a square pole shape (with its side para the cutting direction)
- With the top of the work piece as the reference position for the cutting, the work piece cut to the final cutting point (cutting depth) set at -3.000(mm) as its Z coordinate value the current coordinate system
- The material of the electrode is copper.
- The material of the work piece is HR750.
- The electrode wear is set at 300pm
- The electrode has a bottom area of 100mm²
- The work piece is to be finished with a final surface roughness of 10pm Rmax

Step:

- 1) Move the electrode to the start cutting position
- 2) Select the alignment (Approach face) mode to set the work piece top at "0"
- 3) Press the [Multi.] mode button and the [Random] mode button, to change the screen to the "Random multiple cutting" mode.
- 4) Press the [Cutting type] button and input [1]. (1: assist, assist cutting)
- 5) Set the assist cutting condition. Press the mode key [assist] and sub mode button [assist] to change the screen to the assist, assist cutting mode. Then input the following items.

REFERENCE! Chapter 6 [Assist cutting]

- [Electrode material]: copper;
 - [Work piece material]: HR750;
 - [Electrode shape]: square pole
 - [Electrode wear]: 300pm;
 - [Projected area]: 100mm²
 - [Min. gradient angle]: 0;
 - [LOR AN type]: □
 - [Reference position]: top;
 - [Cutting depth]: -3.000m m;
 - [Finish cutting roughness]: 10pm Rmax;
- 6) Press the [Multiple] and [Selected] buttons return the selected position multiple cutting screen.
 - 7) Press the [Action type] button, input [1]. (1: cutting)
 - 8) Press the [Conti.] button of the [Finish cut meth] item
 - 9) Corresponding to the [No. of holes]/[1],[2],[3] of the [Cutting position set] mode, press the input keys of [Axis X],[Axis Z],and then input the [0] to [9],[.],[+],[-] and the absolute coordinate values of three holes by using the digit keys.

After pressing the [ENT] key, the system will complete automatically the cutting for three Holes according to the order.

8.2 Lattice 1 Multiple Cutting

This sub mode performs the multiple cutting at the Lattice (1) position Press the [Latt.1] button from the four types of cutting buttons of the [Multi], the screen displays the Lattice 1 multiple cutting scene as Fig.8.2.1

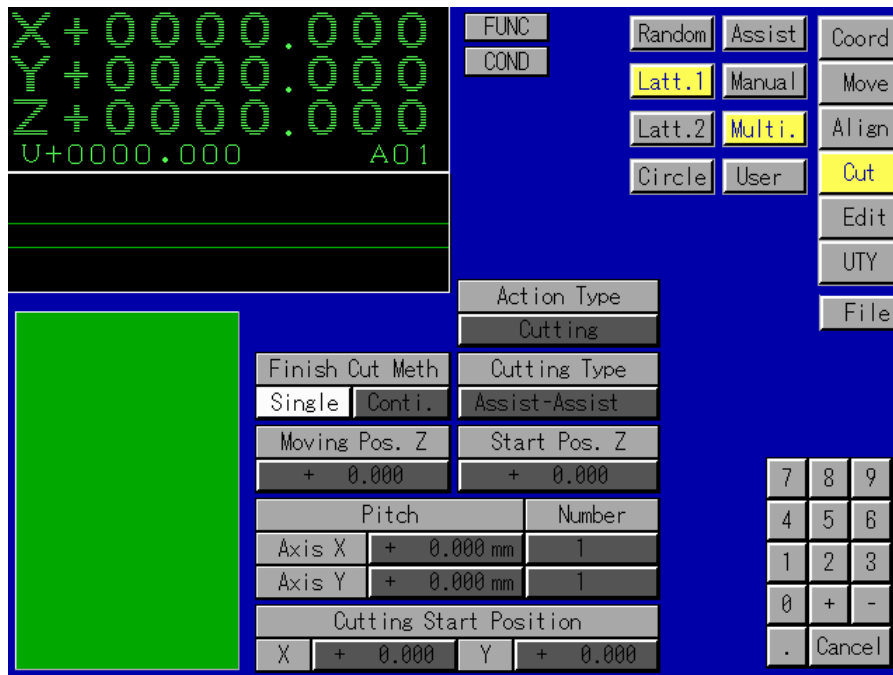


Fig.8.2.1

Three arc following input items in this scene.

- Cutting type
Press the [Cutting type] button, the screen displays the help scene at the left bottom as Fig.8.2.2.

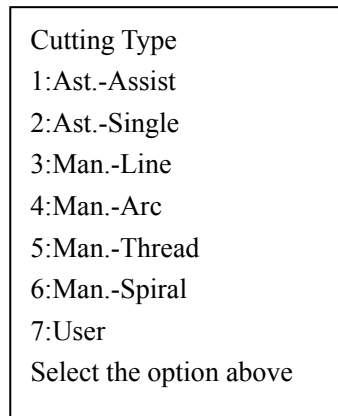


Fig.8.2.2

There are seven types of multiple cutting. Input the No. of cutting types by using the digit keys.

- Action type
Press the [Action type] button, the screen displays the help scene at the left bottom as Fig.8.2.3

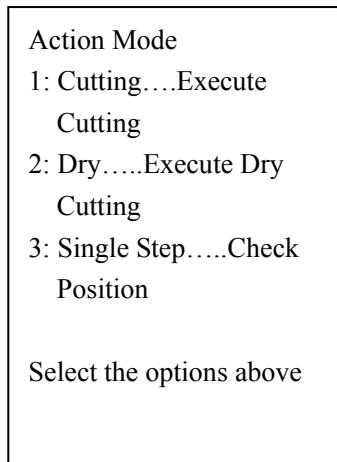
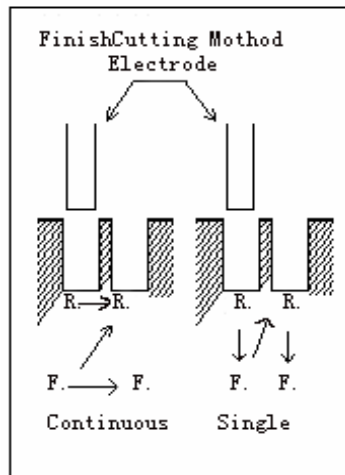


Fig. 8.2.3

There are three types of cutting. Input the No. of action types by using the digit keys

- Press the [Finish cut meth] button, the screen displays the help scene as Fig.8.2.4.



There are two types of the finish cutting methods.

(1) Continuous

This cutting operation is proceeded from the rough cutting to the finish cutting for all holes.

(2) Single

This is the hole cutting operation from the rough cutting to the finish cutting on one-by-one hole.

Select one type from the two types of cutting, then input

- Moving position Z
This parameter is used to input the position of Z axis after it moving from current cutting position to the next cutting position
- Start position Z
It is position of the Z axis when the cutting start
- Pitch and Number

This parameter is used to input the cutting position pitch in the X/Y axis direction and the number of hole to be cut. The total number of holes to be cut can be determined by (the No. of holes in X axis direction) * (the No. of hole in Y axis direction).The diagram is shown as Fig. 8.2.5.

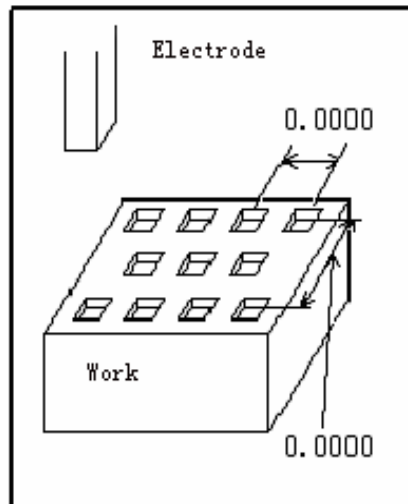


Fig.8.2.5

- Cutting start position X/Y

Input the coordinates values of X axis and Y axis position when the cutting starts. Press the [ENT] key after inputting all items mentioned above, the system will perform automatically the selected Lattice 1 multiple cutting.

[Example of the Lattice 1 multiple cutting operation]

With cutting position (Lattice 1), perform the multiple cutting in (assist, assist cutting with 1 electrode) mode. The cutting diagram is shown as Fig.8.2.6

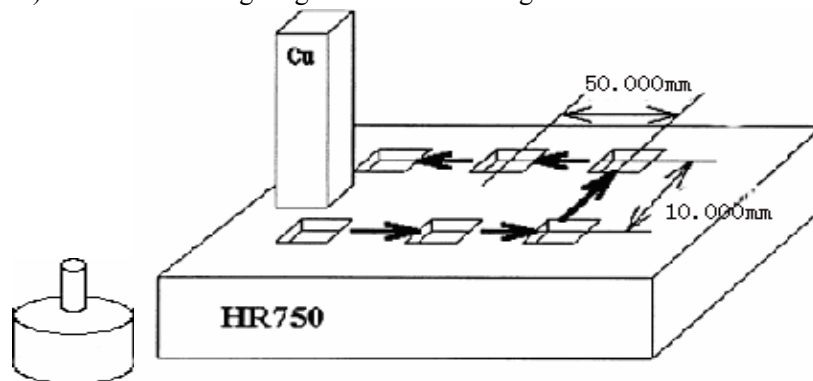


Fig.8.2.6

The cutting conditions and demands are as follows

- Use the continuous finish cutting method
- The electrode to be used for the cutting as a square pole shape (with its side parallel to the cutting direction)

- With the top of the work piece as the reference position for the cutting, the work piece cut to the final cutting point (cutting depth) set at -3.000mm as its Z coordinate value the current coordinate system.
- The material of the electrode is copper.
- The material of the work piece is HR750
- The electrode wear is set at 300p. m
- The electrode has a bottom area is 100mm²
- The workpiece is to be finished with a finish cutting roughness of 10pmRmax

Steps:

- 1) Move the electrode to the start cutting position.
- 2) Select the alignment (Approach face) mode to set the workpiece top at "0"
- 3) Press the [Multi.] mode button and the [Latt.1] mode button, to change the screen to the "Lattice 1 position multiple cutting" mode.
- 4) Press the [Cutting type] button and input [1]. (1: assist=assist cutting)
- 5) Set the assist cutting conditions. Press the mode key [Assist] and submode button [Assist], to change the screen to the assist=assist cutting mode. Then input the following items

REFERENCE Chapter 6 [Assist cutting]

- [Electrode material]: copper;
 - [Workpiece material]: HR750;
 - (Electrode shape): square pole
 - [Electrode wear]: 300pm;
 - [Projected area]: 100mm²;
 - [Min. Gradient]: 0;
 - [LORAN type]:
 - [Reference position]: top;-
 - [Cutting depth]:-3.000mm;
 - [Finish cutting roughness]: 10pm Rmax;
- 6) Press the [Multi.] and [Latt.1] buttons to return the Lattice 1
 - 7) Press the [Conti.] button of the [Finish cut meth] item multiple cutting scene.
 - 8) Press the input button of the [Pitch] corresponding to the [X] axis, then input [5],[0], [.] , [0], [0], [0] and press the input button of the [Number] to input [2].
 - 9) Press the input button of the [Pitch] corresponding to the [W-] axis, then input [1], [0], [.] , [0] ,[0] and press the input button of the [Number] to input [3].
 - 10) Input the absolute coordinates of the [Axis X] and [Axis Y] of the [Cutting start position].
- After pressing the [ENT] key, the system complete automatically the Lattice 1 (6 holes) cutting according to the cutting order

8.3 Lattice 2 Multiple Cutting

This sub mode performs multiple cutting at the Lattice (2) position. Press the [Latt.2] button from the four types of cutting buttons of the [Multi.], the screen displays the Lattice 2 multiple cutting scene as Fig.8.3.1.

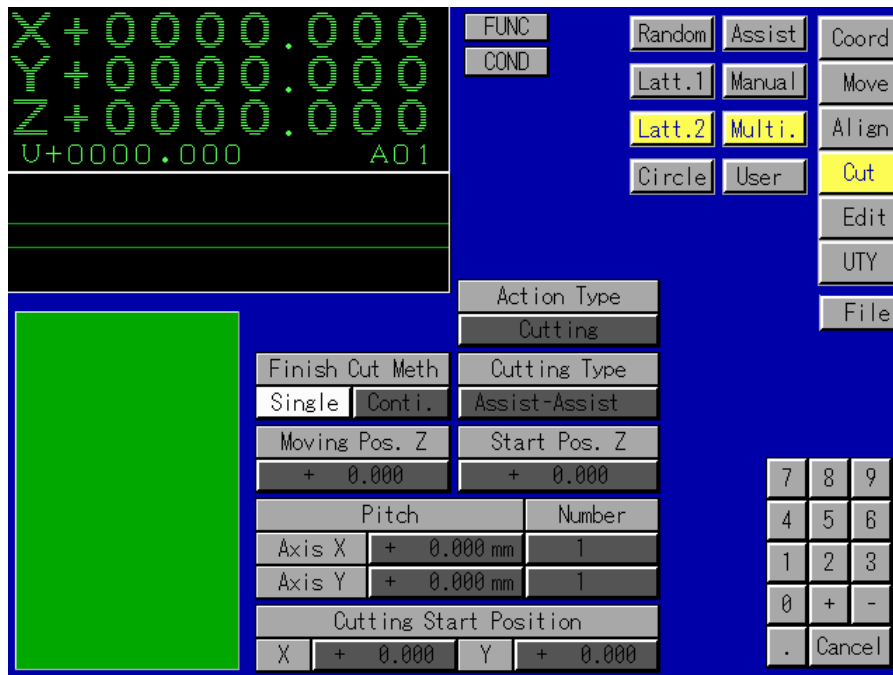


Fig.8.3.1.

There are following input items in this screen-

- Cutting type

Press the [Cutting type] button, the screen displays the help scene at the left bottom as Fig.8.3.2

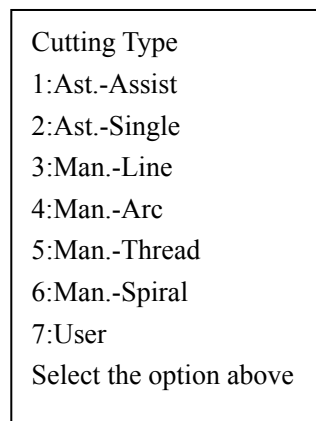


Fig.8.3.2

There are seven types of multiple cutting. Input the No. of cutting types by using the keys.

- Action type

Press the [Action type] button, the screen displays the help screen at the left bottom Fig.8.3.3.

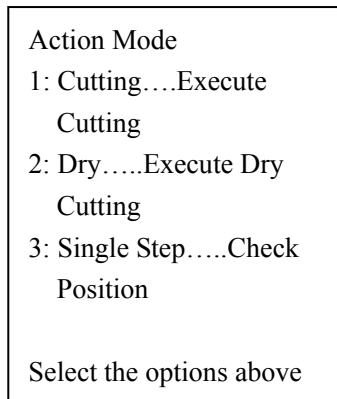


Fig8.3.3

There are three types of cutting .input the No. of action types by using the digit keys.

- Finish cutting method

Press the [Finish cut meth] button, the screen displays the help scene at the left bottom as Fig.8.3.4

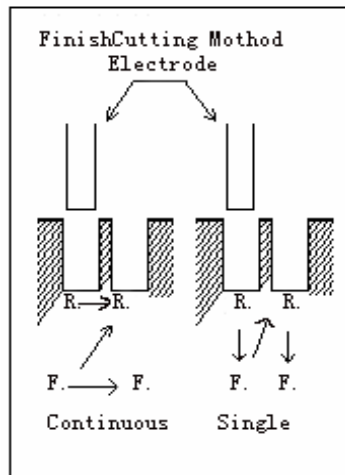


Fig.8.3.4

There are two types of the finish cutting methods

- (1) Continuous

This cutting operations is proceeded from the rough cutting to the finish cutting holes.

- (2) Single

This is hole cutting options form the rough cutting to the finish cutting on one-by-one hole.

Select one type from the two types of the cutting, then input its No

- Moving position Z

This parameter is used to input the position of Z axis after it moving from current cutting position to the next cutting position

- Pitch and Number

This parameter is used to input the cutting position pitch in the X/Y axis direction and the number of holes to be cut. The diagram is shown as Fig.8.3.5

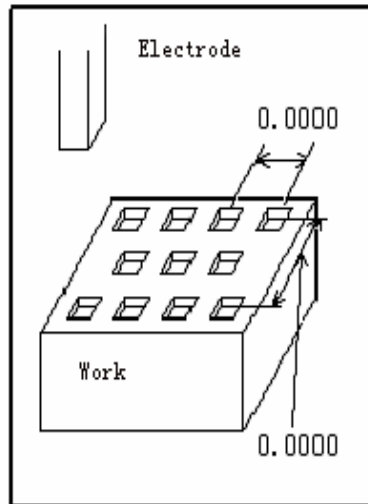


Fig.8.3.5

- Cutting start position X/Y

Input the coordinates values of X-and Y axes position when the cutting starts. Press the [ENT] key after inputting all items mentioned above, the system will perform automatically the selected Lattice 2 multiple cutting.

[Example of the Lattice 2 multiple cutting operation]

With cutting position (Lattice 2), perform the multiple the cutting in (assist, assist cutting with 1 electrode) mode. The cutting diagram is shown as Fig.8.3.6

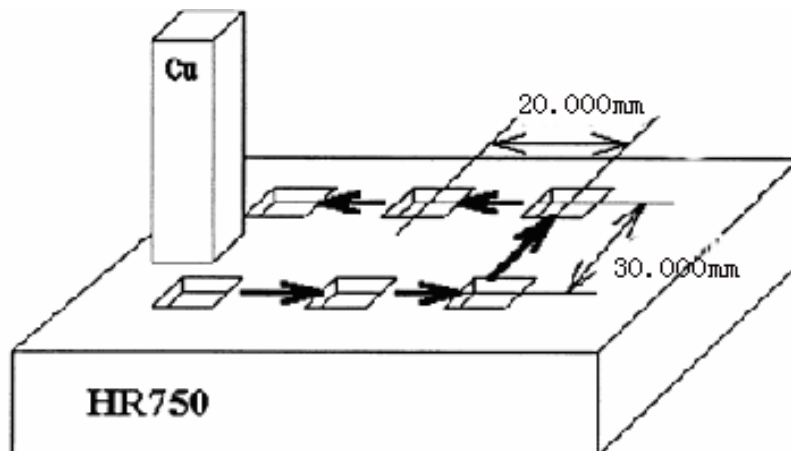


Fig. 8.3.6

The cutting conditions and demands are as follows:

- Use the continuous finish cutting method
- The electrode to be used for the cutting is a square pole shape (with its side parallel to the cutting direction).
- With the top of the work piece as the reference position for the cutting, the work piece is cut to the final cutting point (cutting depth) set at -3.000mm as its Z coordinate value the current coordinate system.

- The material of the electrode is copper. The workpiece material is HR750
- The electrode wear is set at 300pm
- The electrode has a bottom area of 100mm²
- The work piece is to be finished with a finish cutting roughness of 10umRmax

Steps:

- 1) Move the electrode to the start cutting position
- 2) Select the alignment (Approach face) mode to set the workpiece top at “0”
- 3) Press the [Multiple] mode button and the [Lattice 2] mode button, to change the screen to the "Lattice 2 position multiple cutting" mode.
- 4) Press the [Cutting type] button and input [1]. (1: assist, assist cutting).
- 5) Set the assist cutting conditions. Press the mode key [Assist] and submode button [Assist], to change the screen to the assist, assist cutting mode. Then input the following items.

REFERENCE! Chapter 6 [Assist cutting]

- [Electrode material]: copper;
 - [Workpiece material]: HR750
 - [Electrode shape]: square pole;
 - [Electrode wear]: 300pm;
 - [Projected area]: 100mm²;
 - [Min. gradient]: 0;
 - [LOP, AN type]: □
 - [Reference position]: top;
 - [Cutting depth]: -3.000mm
 - [Finish cutting roughness]: 10pm Rmax;
- 6) Press the [Multi.] and [Latt.2] buttons to return the Lattice 2 multiple cutting scene
 - 7) Press the [Action type] button, input [1]. (1. cutting)
 - 8) Press the [Conti.] button of the [Finish cut meth] item.
 - 9) Press the input button of the [Pitch] corresponding to the [X] axis, then [.] , [0], [0], [0] and press the input button of the [Number] to input [3].
 - 10) Press the input button of the [Pitch] corresponding to the [Y] axis, then [.] , [0], [0], [0] and press the input button of the [Number] to input [2].
 - 11) Input the absolute coordinates of the [Axis X] and [Axis Y] of the position.
- After pressing the [ENT] key, the system will complete automatically the Lattice 2 (8 holes) cutting according to the cutting order.

8.4 Circle Multiple Cutting

This sub mode performs the multiple cutting at the circle position.

Select the [Multi.] mode from the [Cut] mode, then press the [Circle] button from the four types of cutting sub modes of the [Multi.] mode. the screen displays the circle multiple cutting scene as Fig.8.4.1.

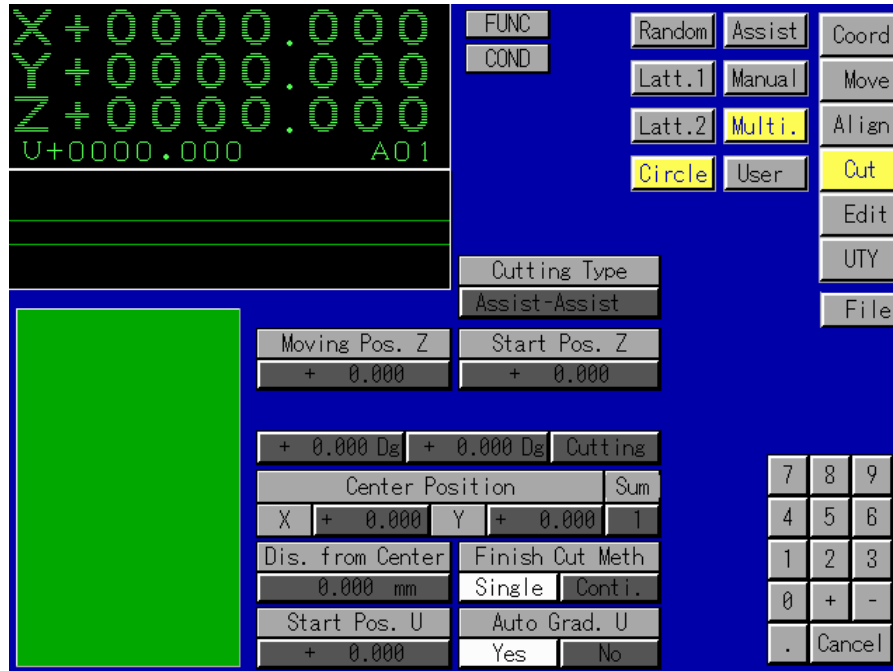


Fig. 8.4.1

There are following input items in this scene.

- Cutting type

Press the [Cutting type] bottom, the screen displays the help scene at the left bottom as Fig.8.4.2

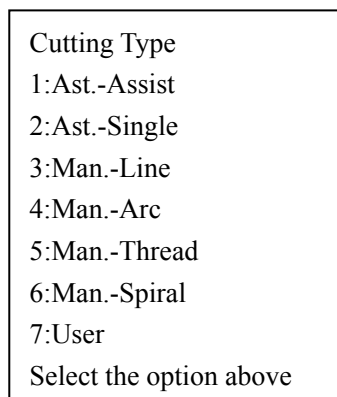


Fig.8.4.2

There are seven types of multiple cutting. Input the No. of cutting types by using the digit Keys.

- Action type

Press the [Action type] button, the screen displays the help screen at the left bottom Fig.8.4.3.

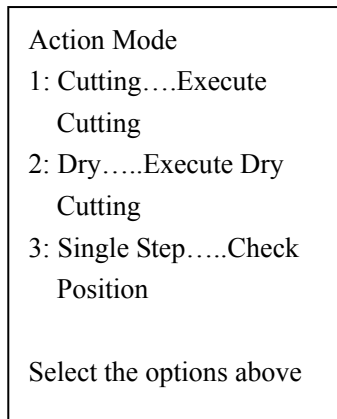


Fig8.4.3

There are three types of cutting .input the No. of action types by using the digit keys.

- Finish cutting method

Press the [Finish cut meth] button, the screen displays the help scene at the left bottom as Fig.8.3.4

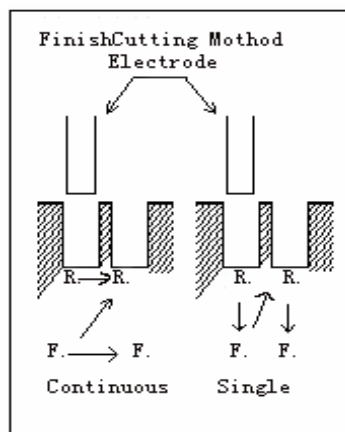


Fig.8.4.4

There are two types of the finish cutting methods

- (1) Continuous

This cutting operations is proceeded from the rough cutting to the finish cutting holes.

- (2) Single

This is hole cutting options form the rough cutting to the finish cutting on one-by-one one hole.

Select one type from the two types of the cutting, then input its No..

- Moving position Z

This parameter is used to input the position of Z axis after it moving from current cutting position to the next cutting position.

- Start position Z

It is the position of the Z axis when the cutting starts.

- Start angle θ

Press [Start angle 0], the screen displays the help scene at the left bottom as Fig.8.4.5.

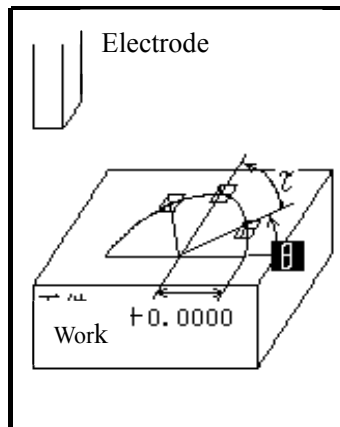


Fig.8.4.5

Input the start cutting angle.

- Cutting pitch angle
It is a pitch angle when the electrode moves to the next cutting position.
- Sum
It is the total number of holes to be cut along the circle.
- Center position
It is a center position coordinates of the circle along which to perform the cutting.
- Distance from center
It is a radius of the circle along which to perform the cutting.
- Start position U
It is the absolute coordinate values of the U axis when the cutting starts.
- Auto Graduation U [Yes] / [No] selection
Press the [Auto grad. U] button, the screen displays the help scene at the left bottom as Fig.8.4.6.

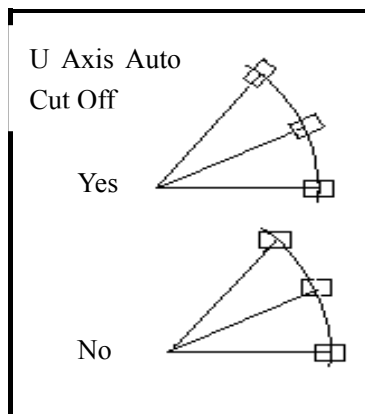


Fig. 8.4.6

Press the [Yes] button, the U axis rotates according to the cutting pitch angle. Press the [No] button, the U axis does not rotate.

After all items mentioned above being input, then press the [ENT] button, the system will perform automatically the selected circle multiple cutting.

[Example of circle multiple cutting]

The multiple cutting (assist, assist cutting with 1 electrode) is executed at the circle cutting positions. The cutting diagram is shown as Fig.8.4.7.

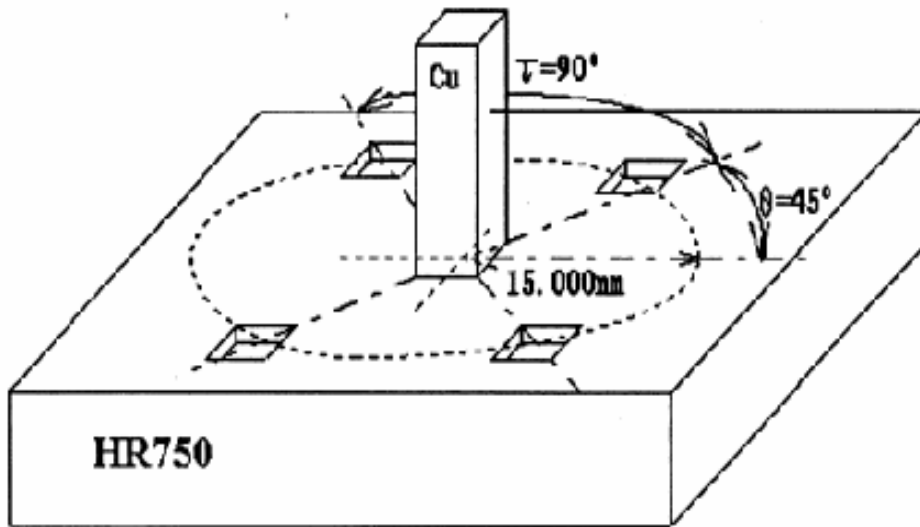


Fig.8.4.7

- The cutting continuous finish cutting method.
- The electrode to be used for the cutting is square pole shape (with its side parallel to the cutting direction).
- With the top of the work piece as the reference position for the cutting, the work piece cut to the final cutting point (Cutting depth) set at -3.000mm as its Z coordinate value in the current coordinate system.
- The material of electrode is copper.
- The work piece material is HR750.
- The electrode wear set at 300pm.
- The electrode has a bottom area of 100mm².
- The work piece is to be finished with a finish cutting roughness of 10μmRmax.

Steps:

- 1) Move the electrode to the start cutting position.
- 2) Select the alignment (Approach face) mode to set the work piece top at "0".
- 3) Press the [Multi.] mode button and the [Circle] mode button, to change the scene to the "Circle position multiple cutting" mode.
- 4) Press the [Cutting type] button and input [1]. (1: assist, assist cutting).
- 5) Set the assist cutting conditions. Press the [Assist] mode button and the [Assist] submode button, to change the screen to the assist=assist cutting mode. Then input the follows items:

- [Electrode material]: copper;
 - [Workpiece material]: HR750;
 - [Electrode shape]: square pole
 - [Electrode wear]: 300pm;
 - [Projected area]: 100mm²
 - [Min. gradient]: 0;
 - [LOP, AN type]:
 - [Reference position]: top;
 - [Cutting depth]:-3.000mm
 - [Finish cutting roughness]: 10pm Rmax;
- 6) Press the [Multi.] and the [Circle] to return the circle multiple cutting scene
 - 7) Press the [Action type] button, input [1]. (1: Cutting).
 - 8) Press the [Conti.] button of the [Finish cut meth].
 - 9) Press the [Start angle 0] button, the start angle shown in Fig.8.4.7 is 45 degree, therefore, input [4], [5], [.), [% [0], [0].
 - 10) Press the [Cutting pitch angle Z] button, the cutting pitch angle shown is Fig 8.4.7 is 90 degree, press the digit keys [9],[0],[.),[0],[0],[0].
 - 11) Press the input button corresponding to the [Sum], and input [4]
 - 12) Input the absolute coordinates of the [X-Axis] and [Y-Axis] of the [Center position] (All of these are "0").
 - 13) Press the input button corresponding the [Dis. from center], input the radius value [1], [5], [.), [0], [0], [0].
 - 14) Input the [Start Pos. U] value [0].
 - 15) Press the [Yes] button of the [Auto Grad. U]

After pressing the [ENT] key, the system will perform automatically the circle (4 holes) cutting.

Chapter 9 USER CUTTING

This chapter describes the NC program created by user cutting method other than the CNC supply unit creates automatically. There are three types, of the user cutting of the NC program being executed on the hard disk, floppy disk and on other external device. The operation methods of the three types of NC program cutting will be introduced as follows.

9.1 HD User Cutting

This mode is used to execute the NC program to be loaded from HD (hard disk).

Select the [User] from the [Cut] mode, the screen displays the [HD], and [FD] sub modes to be selected. Press the [HD] button, the screen displays the HD user cutting scene as Fig.9.1.1

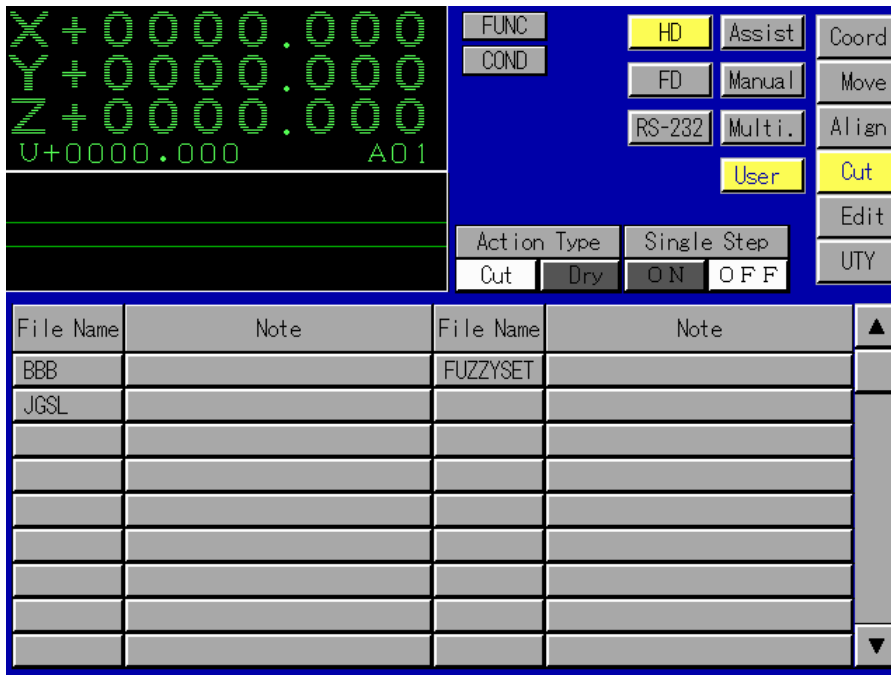


Fig.9.1.1

There are following input items:

- Action type [Cut] / [Dry] selection
Select one of them and then press the button.
- Single step cutting [ON] / [OFF] selection
Select a button from the [ON] and [OFF]. Press the [ON], it indicates the NC program to execute the single step cutting. Press the [OFF], it indicates the NC program not to execute the single step cutting.
- User NC file table
The NC file names and the file notes in the hard disk are listed on the user NC file table. Select the file name need to be executed, then press the file button to change the screen to the NC code contents scene of this file as Fig.9.1.2.

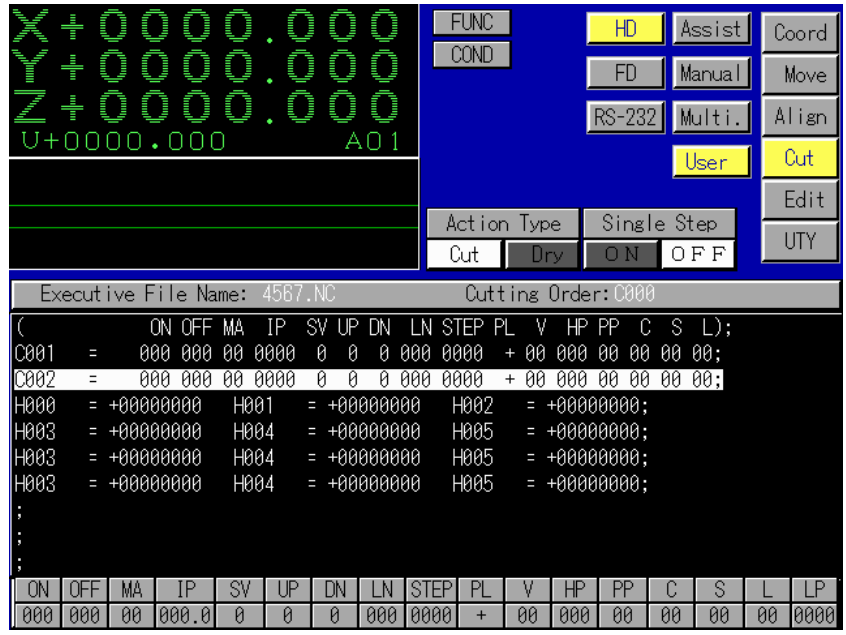


Fig.9.1.2.

After selecting the file, press the [ENT] key, the system executes the cutting according to the code of NC file.

After selecting the file, press the [ENT] key, the system executes the cutting according to the code of NC file.

[Example of user cutting operation]

Execute the NC program "USER NC" in the user field of floppy disk. The dry cutting is selected and the single step cutting is not selected.

Steps:

1) Press the [Cut], [User] and [FD] button in order.

2) Press the [Dry] button of the [Action Type] item.

3) Press the [OFF] button of the [Single Step] item.

4) Find the file name "USER NC" from the file list and press the button of this file name.

After pressing the [ENT] key, the CNC supply unit executes the cutting according to the NC program.

10.2 Edit Function

The function buttons (include [Ins / Rep], [Range set] [Paste] and so on) at the bottom of the edit window are function buttons used to edit the NC files. The operation methods of each function will be introduced as follows:

- [Ins/Rep]

This button is the “alternative” button. Press this button each time, the [Insert] and the [Replace] is selected state ultimately. They are changed to each other. When the [Insert] is selected (This half button is in “concave” state in this time), the character input does not cover the previous character in current cursor. When the [Replace] is selected (The half button is in “concave” state in this time), the input new character will replace the previous character which is in the current cursor.

- [Range set]

Press the [Range set] button, the function button will change as Fig10.1.2.



Fig10.1.2

Move the operation cursor by using the direction keys to select the range of the NC file contents text. The selected range will be displayed in highlight form at this time. The next operation can be executed by pressing the new function keys at this time.

The new function keys are:

- (1) [Del]

Delete the selected text contents(and write this text contents to the clipboard), then return to the last [Insert/Replace] state.

- (2) [Copy]

Copy the selected text contents(and write this text contents to the clipboard), then return to the last [Insert/Replace] state.

- (3) [Top]

Move the cursor to the beginning line of the file, then return to the last [Ins / Rep] state(The contents in the clipboard is not changed at this time).

(4) [End]

Move the cursor to the end line of the file, then return to the last [Insert/Replace] state(The contents in the clipboard is not changed at this time).

NOTE! At this time, the contents in the clipboard will not be changed by pressing any other function keys, then return to the last [Ins / Rep] state.

- [Paste]
Insert the contents in the clipboard to the front of current position where the cursor is in.
- [Find]
Press the [Find] function button, the screen displays the prompt. Input the characters to be found and the searching direction according to the prompt, then press the [Return] key, the system will find the match text contents. After finding, display the match text in highlight form. If you want to find continuously, press the [Return] key. If the match text contents can not be found the prompt message will be displayed.
- [Repla]
Press the [Replace] function button, the screen displays the prompt. Input the old characters and new characters and searching direction according to the prompt, then press the [Return] key. If the match text is found, it is displayed in color highlight. At this time, press the [Return] key again to confirm, system will replace the old characters with the new characters, If you press any other key, the replace will not be done. After this, the system searches next match text till this work is finished.
- [Page Up]
Press this function button, the screen will be turn to the next page, and the screen displays contents of last page.(If the current page is the first page, this function button is invalid.)
- [Page Down]
Press this function button, the screen will be turn to the next page, and the screen displays contents of next page(If the current pae is the last page, this function button is invalid)
- [File]
If you want to do file operation in edit state, you need press the [File] function button. The File operation screen is shown as Fig.10.1.3

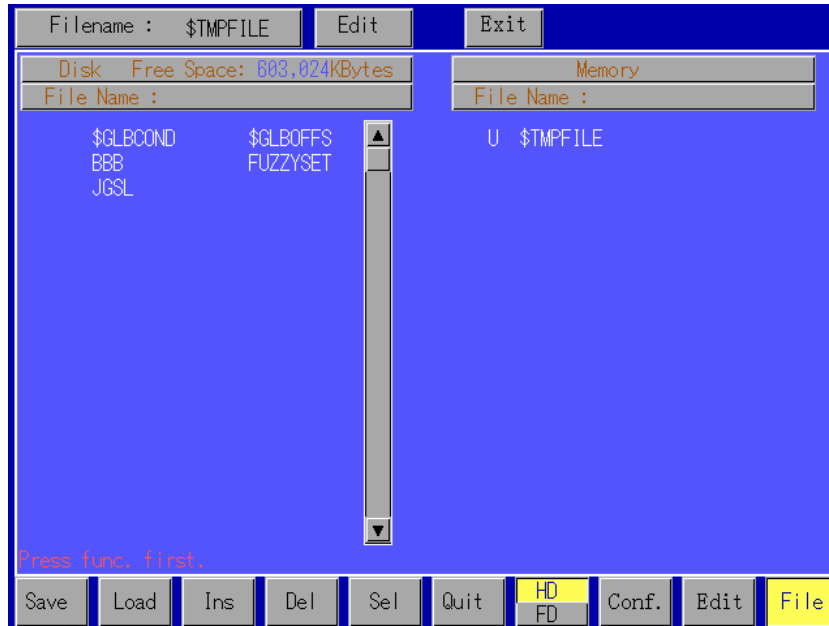


Fig10.1.3

The new function keys in file operation screen are:

- (1) [Save]

Save the current file being edited. Input the file name according to the prompt completing this operation by pressing the [Return] key, then exit this function.
- (2) [Load]

Select the file name to be edited from the file list, press the [Load] button, then the file will be showed in memory file list.
- (3) [Ins]

Insert another file in the current edited file.
- (4) [Del]

After pressing this button, select the file name to be deleted from the file list, then press the [Confirm] button and complete the deleting operation
- (5) [Sel]

Select the file being edited in memory file list and change it to the current file to be edited.
- (6) [Quit]

Exit (not save) the current editing NC file from memory.
- (7) [HD/FD]

This button is the “alternative” button, its default state is “HD(hard disk)”, the file will be read and written in the hard disk. After pressing the button again, change to “floppy disk” state and the file operation in edit mode will be in the floppy disk.
- (8) [Edit]

After pressing this button, the current file operation screen will return to the last edit screen and function buttons.

The cutting can be executed in the [User] mode, after editing the NC file.

PART 4 UTY

The UTY (UTILITY) is the practical mode of the Z system. In this part, the applications and operations methods of several practical tools of the UTY will be introduced.

This part consists of following chapters:

Chapter 11 Password

Chapter 12 Setting

Chapter 11 Password

11.1 Password Screen Configuration

Press the [UTY] mode button, the screen displays as Fig. 11.1.1

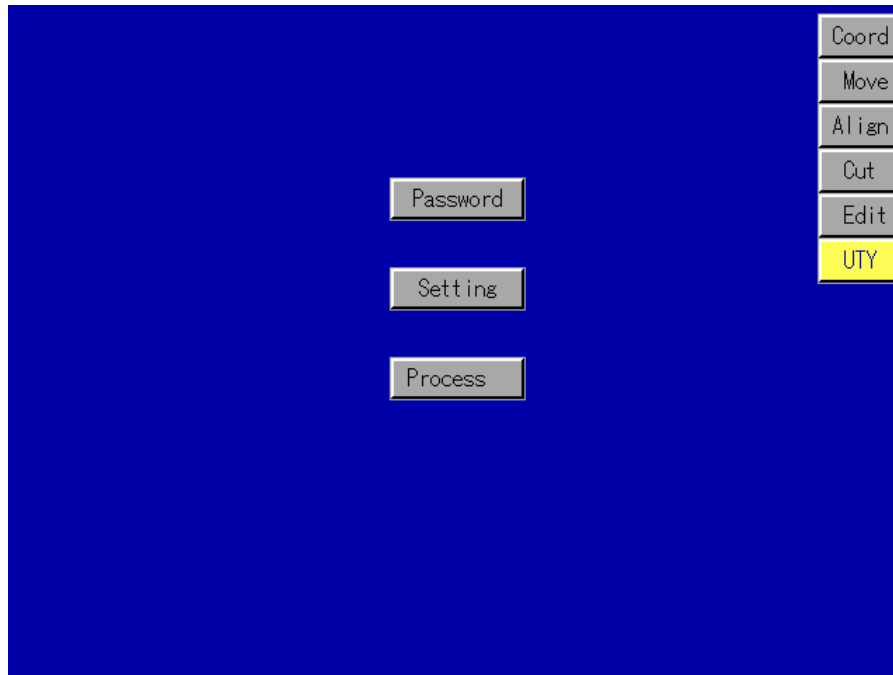


Fig.11.1.1

This screen consists of the [Password], [Setting] and [Process] three sub mode buttons. Select the [Password] button of them, the screen displays the edit scene as Fig. 11.1.2.



Fig.11.1.2

User can create 8 digits password by enter number keyboards under input box.

CHAPTER12 SETTING

12.1 Introduction

This chapter describes the operation setting of A power supply, disk processing and detail function.

The setting includes sub modes of [FLAG], [CHECK] etc.

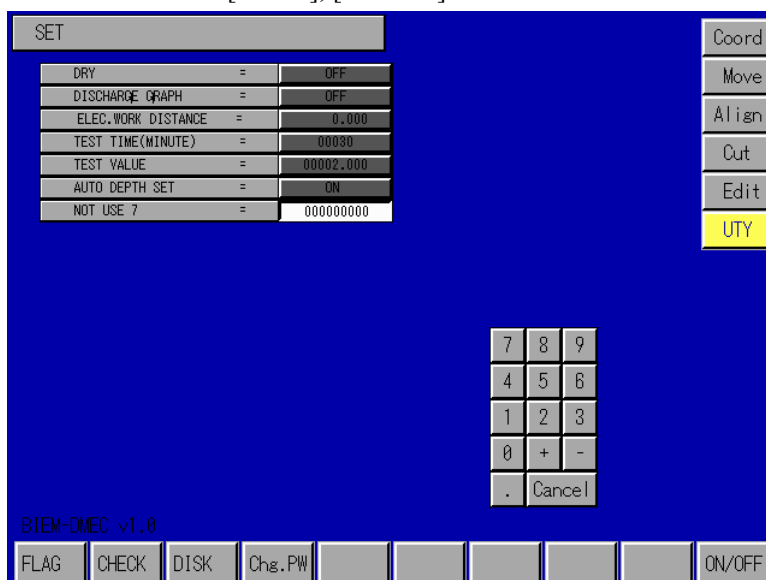


Fig.12.1 Set Sub mode Screen

◆ DRY

This parameter is used to decide whether to execute EDM operation when G01/G02/G03 is executed.

OFF: To execute program with DISCHARGE ON.

ON: To execute program with DISCHARGE OFF.

NOTE! For ST operation by G80 with DRY ON, refer to SET.FLA SET.MOTOR SPEC parameter "G80 DRY IGNORE".

Table 1

DRY	RUN Cutting	ST
OFF	ON	ON
ON	OFF	ON

◆ DISCHARGE GRAPH

This parameter is used to decide whether to display the discharge graph at the lower right of the screen during the cutting.

OFF: Not to display the discharge graph.

ON: To display the discharge graph.

◆ ELECTRODE-WORK DISTANCE

This parameter is used to set the distance between the electrode and the work piece at the start of the cutting.

◆ MEASURING TIME

This parameter is used to set the scale of the axis of abscissas (axis of time) of the discharge graph in minutes.

◆ MEASURING VALUE

This parameter is used to set the scale of the axis of ordinates (axis of cutting length) of the discharge graph.

12.2 FLAG SET

The [FLAG] sub mode is used to set the program starting condition, machine operation pattern, cutting control pattern and power supply specifications.

- Data setting method

The parameter selection can be made by operating the cursor-move keys or press input buttons for data input. After the data input, the data only becomes valid when another input button is pressed, which can be cancelled by pressing the [CANCEL] button, and causing it to be returned to its previous data(with the exception of the parameters requiring ON/OFF setting).

The parameters can be classified by data input pattern into the following four types:

1. ON/OFF parameter

The parameters requiring ON/OFF setting can be set using the button [ON/OFF].

Pressing the [ON/OFF] key causes the ON/OFF state of the specified parameter to be selected alternately.

2. Integral parameter (with +/- sign)

The parameters requiring input of integral data with +/- sign can be set using the [+], [-] and [0] to [9] buttons for data input.

3. Integral parameter (without +/- sign)

The parameters requiring input of integral data without +/- sign can be set using the [0].

4. Numeric data with units (which can be expressed using decimal point with +/- sign)

The parameters displayed yellow for classification as this type can be set using the [+], [-], [.] and [0] to [9] buttons for data input.

12.2.1 FLAG SET FLAG SET

The [FLAG] sub mode is used to set the initial parameter data necessary for execution of the program, This sub mode can be selected by pressing the [FLAG] button, causing the screen to display such a screen image as illustrated in Fig.12.2.1 (a) and (b) can be selected alternately by pressing the [FLAG] button.

NOTE! The marks “*” indicate the characteristic parameters of the machine, all of which are set before its shipment from the factory, not requiring setting change by the user.



Fig.12.2.1(a) SET.FLAG SET.FLAG Screen

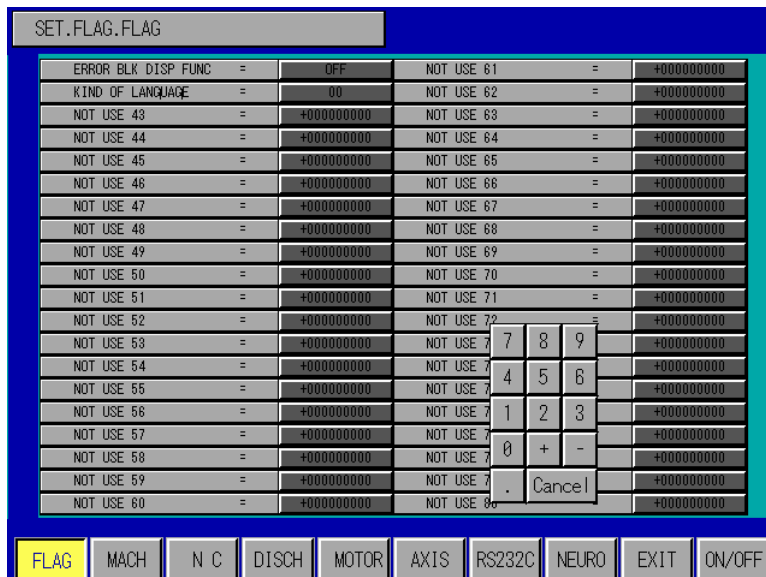


Fig.12.2.1(b) SET.FLAG SET.FLAG Screen

◆ SKIP

This parameter is used to decide whether to execute the NC program with “/” at its beginning.

OFF: To execute program block with “/”

ON: Not to execute program block with “/”

G 12- SKIP OFF

G 11- SKIP ON

◆ NO MAN

This parameter is used to decide whether to cause the CNC supply unit to be automatically. Turned OFF when M02 is executed or an error occurs with parameter DRY set in “OFF” state.

0: Not to turn POWER OFF if the above condition occurs.

1: To turn POWER OFF automatically when M02, SERVO RETURN STOP FLOAT STOP or FLUID TEMPERATURE STOP is executed.

2: To turn POWER OFF if, in addition to such a machine state as mentioned in "1", above ,an error occurs, causing the machine to wait for the operation of the “ACK” key.

1. Setting the parameter with any number more than "2" causes it to be set in the same way as when it is set at “1”.

2. When SET. FLAG SET. FLAG parameter "DRY" is set in "ON" state, the CNC supply unit will not be turned OFF when any of the above conditions occur whether this parameter is set at “1” or “2”.

3. When SET- FLAG SET. FLAG parameter "M03 PATTERN" is set with any other number than “0”, execution of M03 is given priority.

◆ BUZZER

This parameter is used to decide whether to cause the buzzer to continue sounding when M02 is executed, an error occurs or the [OFF] key or the [HALT] key is pressed.

OFF: To cause the buzzer to sound one time.

ON: To cause the buzzer to continue sounding.

Press of the [ACK] key stop the buzzer.

If the machine is in halted state, press of the [ENT] key stops the buzzer, releasing the machine from its halted state to resume its operation.

◆ M03 PATTERN

This parameter is used to decide whether to search M03 when errors listed in Table 2 occur and whether to ignore GOO, GO1, G02, G03, G04, G128 to 928, G30, G60, G80, G81, G82, G65 when searching M03. When M03 is uncovered the program will start from the block including M03.

Table2

M03 PATTERN \ Error	ARC ATOP	LIMIT STOP	ST STOP
0	×	×	×
1	O	×	×
2	O	×	O
3	O	×	O
4	O	O	O

ARC STOP: ARC stop
Servo return stop

LIMIT STOP: LIMIT stop
Stroke stop

O: To execute M03 search
×: Not to execute M03 search.

*: SET.FLAG SET. FLAG parameter "M03 PATTERN" setting method:

0: Not to execute M03 search.

1: To execute M03 search when ARC STOP occurs.

2: To execute M03 search when ARC STOP/LIMIT STOP occurs.

3: To execute M03 search when ARC STOP/ST STOP occurs.

4: To execute M03 search when ARC STOP/LIMIT STOP/ST STOP Occurs.

*: The following M codes are to be executed even during M03 search.

Table 3

M code	M03 Execution	M code	M03 Execution
M00	○	M10-47	×
M01	○	M50-87	×
M02	○	M95	○
M03	○	M98-99	○
M04	○	M110-147	×
M05	○	M150-187	×
M06	○	M230-247	×
M07	○	M350-387	×
M08	○	M450-487	×
M09	○		

O: To be executed during M03 search..
×: Not to be executed during M03 search.

◆ OPTION STOP

This parameter is used to decide whether to execute M00 operation when M01 code is executed

OFF: Not to execute M00 operation, causing NC program processing to be continued.

ON: To execute M00 operation, causing NC program processing to be stopped(To reexecute the program processing, press the [ENT] key).

◆ MOO STOP AXIS RETURN

This parameter is used to decide whether to return the Z axis, fed by JOG operation during MOO (M01) caused temporary program stop, to its original position or allow the axis to stay at the JOG set position when the [ENT] key is pressed to resume the program execution.

Table 4

Cause of temporary Program stop	Activation of HALT key	Execution of M00/M01
M00 STOP AXIS RETURN		
0	O	O
1	O	×

O: To return Z axis to its original position before resuming program execution.

X: To allow Z axis to stay at its JOG set position for resuming program execution.

When MOO STOP AXIS RETURN is set for O in the above table:

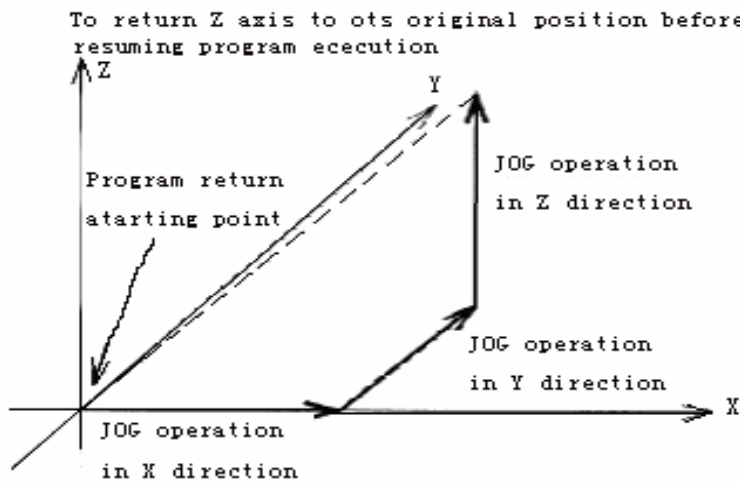


Fig.12.2.1.1

When MOO STOP AXIS RETURN is set for x in the above table:

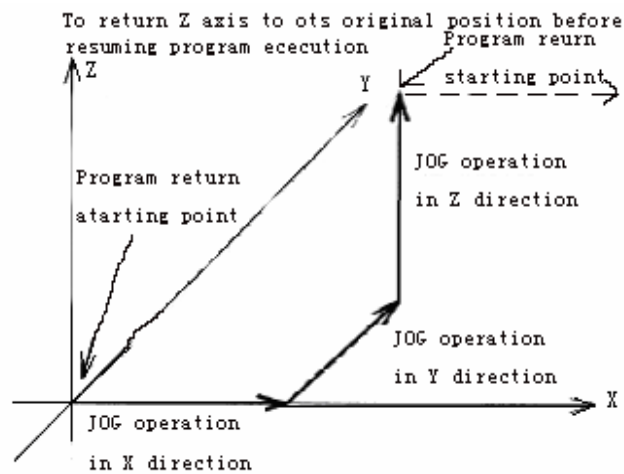


Fig.12.2.1.2

12.2.2 FLAG SET. MACHINE SPEC SET

The "MACHINE SPEC SET" sub mode is used to set the parameters on the machine operation pattern.

The sub mode can be selected by pressing the [MACHINE] button, causing the screen to display such a screen image as illustrated in Fig. 12.2.2 (a). Fig. 12.2.2 (a) and (b) can be selected alternately by pressing the [MACHINE] button.



Fig 12.2.2(a)



Fig 12.2.2(b)

12.2.3 FLAG SET. NC SET

The "NC SET" sub mode is used to set the movement method of NC and control the program execution.

The sub mode can be selected by pressing the [NC] button, causing the screen to display such a screen image as illustrated in Fig. 12.2.3 (a). Fig. 12.2.3 (a) and (b) can be selected alternately by pressing the [NC] button.

NOTE! The marks "*", indicate characteristic parameters of the machine, all of which are set before its shipment from the factory, not requiring setting change by the user.

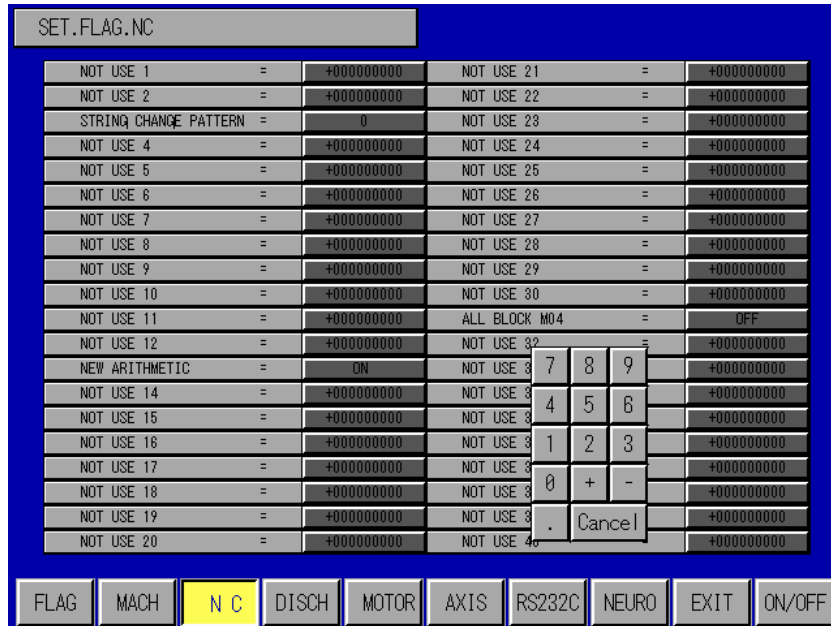


Fig. 12.2.3 (a)



Fig. 12.2.3 (b)

◆ STRING CHANGE TYPE

0: Not change.

1: To change character string A to character string B with input into STRING file in the format of STRING A:STRING B.

2: To change character put between character string A and character string B so that the character follows character string C with input into STRING file in the format of STRING A \$ STRING B: STRING C.

3: To execute the above two character string changes.

◆ NEW ARITHMETIC

This parameter is used to select whether to execute arithmetic operation of multiplication and division in NC program with units put to the multiplicand and dividend. The parameter is to be normally set in state.

OFF: To execute multiplication and division with no unit put to the multiplicand and dividend. The selection of the OFF state causes 1.0 to be mechanically equivalent to 1000.

ON: To execute multiplication and division with units put to the multiplicand and dividend. The multiplier and divisor are coefficients.

[Example]

The multiplicand of 30 in NC program is to be processed as the following data according to the setting of the parameter:

NEW ARITHMETIC OFF: 30000

 ON: 30

[Example of calculation] $10\ 0 \times 0.5$

└────────────────────────── This only is valid.

The above calculation is to be made as shown below according to the setting of the parameter when the input is made in mm:

NEW ARITHMETIC OFF: $10000 \times 500 = 5000000$

 ON: $10 \times 500 = 5000$

◆ ALL BLOCK M04

This parameter is used to select whether to execute G01 (straight cutting) in NC program with M04 automatically inserted into the block involved in order to return the electrode to the cutting start position after the execution of GO 1 (straight cutting).

OFF: To execute G01 without automatic insertion of M04 in its program block.

ON: To execute G01 with automatic insertion of M04 in its program block.

◆ COORDINATE CANCEL

This parameter is used to set the pattern for automatic change of the present coordinate system to G54 system.

0: To change the present coordinate system to G54 system only when G54 is executed.

1: To change the present coordinate system to G54 system when G54 or M02 is executed.

2: To change the present coordinate system to G54 system when G54 or M02 is executed or an error occurs.

◆ ABS/INC CANCEL

This parameter is used to select whether to change the absolute/incremental system to the absolute system by execution of G90 when an error occurs or M02 is executed.

- 0: Not to change the present system until the (ABS/INC) selection code (G90/G91) is executed.
- 1: To change the present system to the absolute system by execution of G90 when M02 is executed.
- 2: To change the present system to the absolute system by execution of G90 when M02 is executed or an error occurs with subsequent activation of the [ACK] key.

◆ MCODE OFFSET

This parameter is used to set the input offset No. in M CODE JUMP.

0 : Not to do M CODE JUMP.

Except 0 : To set the input state of M code in the set offset No.

12.2.4 FLAG SET.DISCHARGE SPEC SET

The "DISCHARGE SPEC SET" sub mode is used to set the parameters necessary for the cutting operation, cutting control and cutting conditions. This sub mode can be selected by pressing the [DISCHARGE] button, causing the screen to display such a screen image as illustrated in Fig. 11.2.4 (a). The Fig. 1 1.2.4 (a) and (b) can be selected alternately by pressing the [DISCHARGE] button.

The marks "**", indicate characteristic parameters of the machine, all of which are set before its shipment from the factory, not requiring setting change by the user.



Fig. 12.2.4 (a)



Fig. 12.2.4 (b)

◆ LORAN SERVOBACK

This parameter is used to set the reference amount by which to servo back the spindle to stop its LOP, AN operation and the amount by which to servo back the spindle to return the spindle to the center of its LORAN operation. The parameter setting is made by data input in the range of 0.0010 to 15.0000 in mm.

If the machine under goes a short-circuit during the cutting in FREE LORAN mode, it will instantaneously stop its LOP, AN operation, causing the spindle to servo back. As long as the machine is left in this state, however, the electrode will remain in contact with the side of the work piece, resulting in failure to discharge the debris caused on the work piece side. Therefore, the spindle undergoes a short-circuit, causing it to servo back continuously, it is necessary to stop the LORAN operation and return the spindle to the center of the LORAN operation.

◆ LORAN STEP UP JUDGE

This parameter is used to set the reference data to judge whether to increase the LORAN STEP level. The higher the setting of the parameter, the less the possibility of increasing the LORAN STEP level. The parameter setting is made by input in the range of 1 to 250. When the LOCK LORAN mode is selected for cutting, the spindle starts its LORAN operation with a small STEP level, gradually increasing the STEP level for finishing until the target STEP level is reached before ending the cutting operation. During this process, the machining state is judged for each cycle of LORAN to increase the STEP level if the cutting performs well or to maintain/decrease the STEP level if the cutting performs poor.

◆ HS LORAN ADVANCE UNIT

This parameter is used to set the servo range of the servo axis and its advance amount for each quadrant LORAN operation. "HS LORAN, is the operation mode in which to limit the cutting axis (servo axis) operation to give priority to the LORAN operation. Therefore, execution of the HS LORAN mode operation causes the servo of the servo axis to be limited to the preset range. During this process, the machining state is judged for each quadrant LORAN operation to advance the servo range of the servo axis in the cutting direction by the amount set by this parameter if the cutting performs well or to maintain/retract its servo range if the cutting performs poor.

The parameter setting is made by input in the range of 0 to 0.0990 in mm.

◆ STD.SERVO SPEED ZONE

This parameter is used to set the distance which to servo back the spindle from the cutting end to change the servo speed. When the servo back distance from the cutting end exceeds the setting of this parameter, the servo speed becomes higher than set by parameter "S". This is to achieve quicker restoration of the machining state if the spindle servo back occurs to a large extent due to the debris produced by machining or other causes.

The setting of the parameter is made by input in the range of 0 to 15.0000 in .mm

◆ AUTO GAP CLEANING ZONE

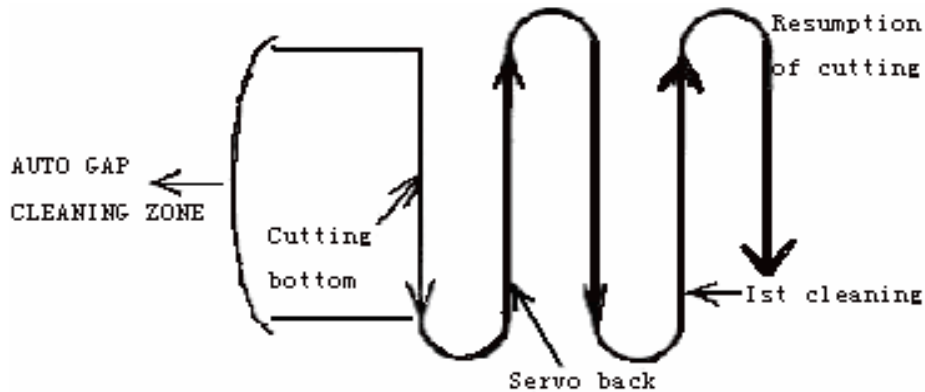
This parameter is used to set the distance which to move the electrode from the cutting bottom and back with circulation of the dielectric fluid for cleaning of the gap by removal specified level of the debris from it when arc is detected or the servo back amount exceeds the rated value.

The setting of the parameter is made by input in the range of + 1000 to + 14000 in terms of number of pulses.

◆ AUTO CLEANING TIMES

This parameter is used to set the number of times by which to make repetitions of moving the electrode back and forth between the distance set by parameter "AUTO GAP CLEANING ZONE" for automatic cleaning of the gap. If this parameter is set at "0", the gap cleaning function is not operative, if this parameter is set at "1", the gap cleaning function occurs as illustrated in Fig. 12.2.4.1 below:

[Example]



Fog.12.2.4.1

◆ * IP MAX

This parameter is used to set the upper limit of "IP" (Main power supply peak current).

◆ * SFD 0-SFD 9

This parameter is used to set the internal data for the cutting feed speed set by parameter "S". SFD0 corresponds to parameter "S" set at "0" and SFD1 to parameter "S" set at "1". The higher the setting of this parameter, the lower the feed speed.

12.2.5 FLAG SET.MOTOR SPEC SET

The "MOTOR SPEC SET" sub mode is used to set the parameters necessary to determine the machine tool control axis operation pattern and the motor speed. This sub mode can be selected by pressing the [MOTOR] button, causing the screen to display such a screen image as illustrated in Fig. 12.2.5 (a). The Fig.12.2.5 (a) and (b) can be selected alternately by pressing the [MOTOR] button.

NOTE! The marks “*”, indicate characteristic parameters of the machine, all of which are set before its shipment from the facto factory, not requiring setting change by the user.

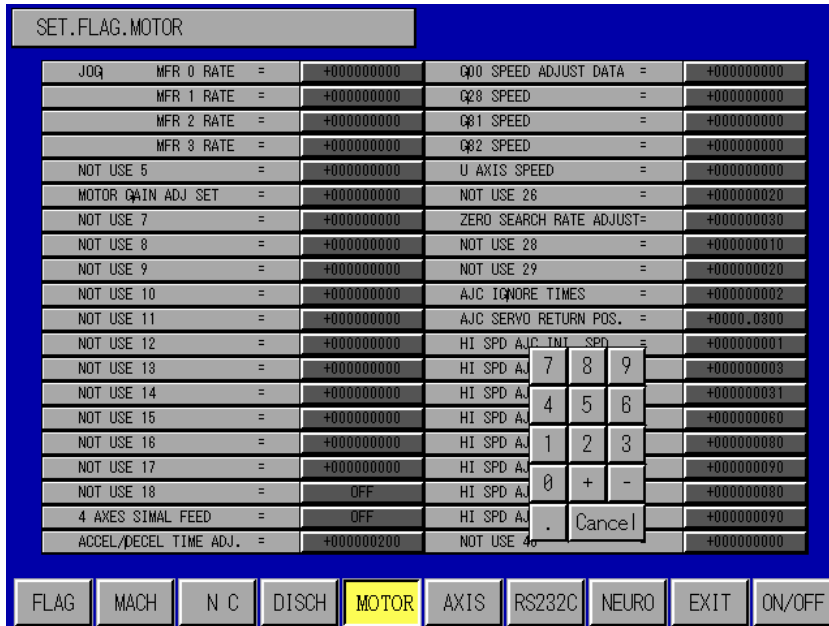


Fig.12.2.5 (a)



Fig.12.2.5 (b)

- ◆ JOG MFR 1 RATE
This parameter is used to set JOG MFR0 feed speed.
The setting of the parameter is made by input in the range of 0 to 255 (If the parameter is set at "0", the JOG MFR0 feed speed can achieve its maximum level).
- ◆ JOG MFR 1 RATE
This parameter is used to set the JOG MFR1 feed speed.
The setting of the parameter is made by input in the range of 0 to 255. (If the parameter is set at "0", the JOG MFR1 feed speed can achieve its maximum level.)
- ◆ JOG MFR 2 RATE
This parameter is used to set the JOG MFR2 feed speed.
The setting of the parameter is made by input in the range of 0 to 255. (If the parameter is set at "0", the JOG MFR2 feed speed can achieve its maximum level.)
- ◆ JOG MFR 3 RATE
Pressing MFR3 one time will send a pulse, and cause the movement of one pulse.
Pressing this key continuously will cause continuous movement. This parameter is used to set the JOG MFR3 feed speed. The setting of the parameter is made by input in the range of 0 to 255. (If the parameter is set at "0", the JOG MFR3 feed speed can achieve its maximum level.)
- ◆ 4 AXES SIMUL FEED
This parameter is used to select whether to feed the four axes simultaneously or the three axes X/Y/Z simultaneously with subsequent feed of the U axis when GOO is executed.
OFF: To feed the three axes X/Y/Z simultaneously with subsequent feed of the U axis.
ON: To feed the 4 axes simultaneously.
But the four axes can be feeder simultaneously only in the case that the motor s of the 4 axes are all DC motors or AC motors. -
- ◆ *4. ACCEL/DECEL TIME
This parameter is used to set the acceleration/deceleration of the electrode to reach its travel speed set in the JOG, FEED. or other similar modes.
The higher the setting of the parameter, the lower the acceleration and deceleration of the electrode to reach its set travel speed.
- ◆ G00 SPEED RATE
This parameter is used to set the G00 feed speed.
The setting of tile parameter is made. by input in the range of 0 to 255. (If the parameter is set at "0", the G00 feed speed can achieve its maximum level.)
- ◆ G28 SPEED
This parameter is used to set the speed of the electrode for its return to the reference point.
The setting of tile parameter is made by input in the range of 0 to 255. (Setting the parameter at "0" causes the electrode to return to the reference point at the maximum speed.)
- ◆ G81 SPEED
This parameter is used to set the speed of the electrode for its travel to the limit.
The setting of tile parameter is made by input in the range of 0 to 255. (Setting the parameter at "0" cause the electrode to travel to the limit at the maximum speed.)

- ◆ **G82 SPEED**
This parameter is used to set the speed of the electrode for its travel halfway. The setting of tile parameter is made by input in the range of 0 to 255. (Setting the parameter at "0' causes the electrode to travel halfway at the maximum speed.)
- ◆ **U AXIS SPEED**
This parameter is used to set the speed of U axis movement.
The setting of tile parameter is made by input in the range of 0 to 255. (Setting the parameter at "0' causes the speed achieve its maximum level.)
- ◆ **AJC ZERO SEARCH RATE**
This parameter is used to set the zero point searching rate.
The higher the setting of the parameter, the lower the searching rate. The setting of the parameter is made by input in the range of 0 to 255. (If the parameter is set at "0", the searching rate can achieve its maximum level.)
- ◆ **AJC IGNORE TIMES**
This parameter is used to set the timing to start AJC operation after the start of the cutting. When the cutting starts, the AJC operation signal will be turned ON. Just after starling its cutting operation, however, the machine operates under no load, making AJC operation at this stage meaningless. Therefore, this parameter can be used to set the number of times by which to ignore AJC operation after the start of the cutting. For example, if the parameter is set at "2", the AJC operation will not start unless the state below the reference voltage occurs two times or more from the start of the cutting. The setting of the parameter is made by input in the range of 0 to 255.
The parameter setting has an influence on NC codes G24 (Hi SPD AJC) and G25 (CUTTING PATH AJC).
- ◆ **AJC SERVO RETURN POS**
This parameter is used to set the difference between the AJC start and end positions.
When starting its AJC operation, the electrode will back by the distance set by this parameter at a servo speed before performing the JUMP operation.
The setting of the parameter is made by decimal input in the range of 0 to 10.0 in mm
The parameter setting has an influence on NC codes G24 (HI SPD AJC) and G25(CUTTING PATH AJC).

[Example]

When AJC SERVO RETURN SITE is set at "+0001.0000", the electrode will back at a servo speed by 1.0mm before starting its AJC operation as illustrated in the following drawing:

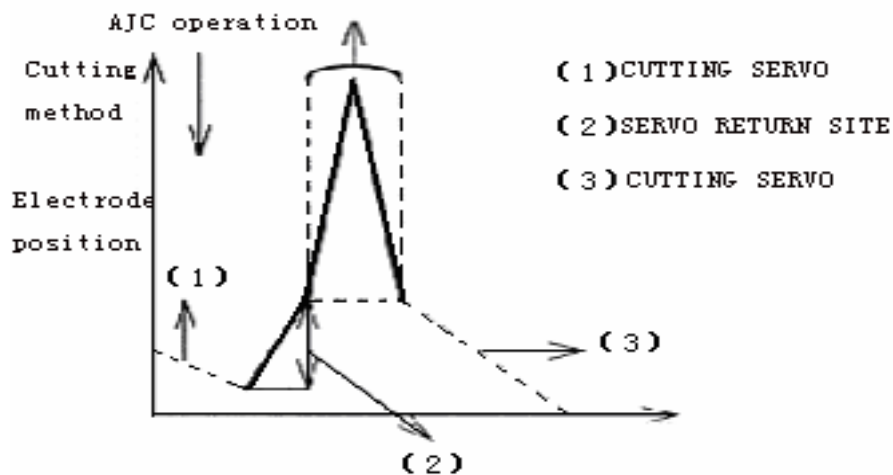


Fig.12.2.5.1

- ◆ **HI SPD AJC INI SPD**
 This parameter is used to set the origin speed of high speed AJC(G24).
 The bigger the set value is, the higher the origin speed is.
 The input range is from 1 to (Hi SPD AJC MAXIMUM SPEED-2).
- ◆ **HI SPD AJC AUXI ADD SPD**
 This parameter is used to set the speed increased in the approach time of the high speed AJC.
 The bigger the set value is, the bigger the increment of the speed is.
 The input range is from 1 to (HI SPD AJC HIGHEST SPD-Hi SPD INI.SPD).
- ◆ **HI SPD AJC HIGHEST SPD**
 This parameter is used to set the maximum speed of the high speed AJC
 The bigger the set value is, the higher the maximum speed is.
 The input range is from 2 to 31.
- ◆ **HI SPD AJC AUXI TIME**
 This parameter is used to set the approach time of the high speed AJC
 The bigger the set value is, the longer the approach time is.
 The input range is from 1 to 300.
- ◆ **HI SPD AJC UP ACCE TIME**
 This parameter is used to set the time needed to accelerate the speed in the rising of the high speed AJC.
 The bigger the set value is, the longer the needed time is.
 The input range is from 1 to 50.
- ◆ **HI SPD AJC UP SLOW TIME**
 This parameter is used to set the time needed to decelerate the speed in the rising of the high speed AJC.
 The bigger the set value is, the longer the needed time is.
 The input range is from 1 to 50.

◆ HI SPD AJC DN ACCE TIME

This parameter is used to set the time needed to accelerate the speed in the descending of the high speed AJC.

The bigger the set value is, the longer the needed time

The input range is from 1 to 50.

◆ HI SPD AJC DN SLOW TIME

This parameter is used to set the time needed to decelerate the speed in the descending of the high speed AJC.

The bigger the set value is, the longer the needed time is.

The input range is from 1 to 50.

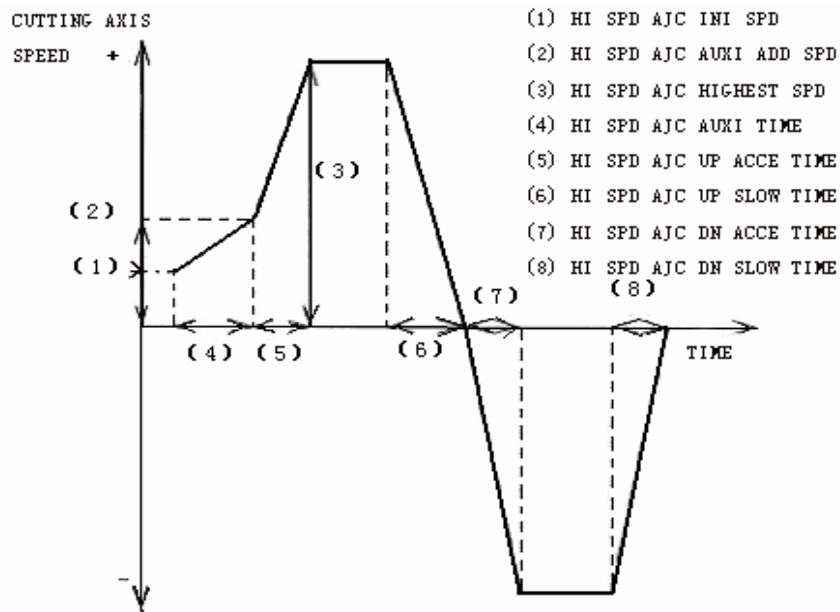


Fig. 12.2.5.2

- ◆ **G80 SPEED RATE**
 This parameter is not in use at present
- ◆ **ST RETURN VALUE(FIG. 11.2.5.3)**
 This parameter is used to set the amount by which to return the electrode from the position of its first contact with the work piece caused by G80 command. The G80 operation causes the electrode to return by this set amount from its work piece contact position before performing the next ST operation.
 The setting of the parameter is made by decimal input in mm.
- ◆ **ST SITE TIMES**
 This parameter is used to set the number of times by which to cause the electrode to perform ST operation for measurement of the position of its contact with the work piece. The average of the measurements will be determined as the final stop position.
- ◆ **G80 WATCH SPEED RATE**
 This parameter is used to set the speed at which to cause the electrode to perform ST operation by G80 command, checking that the electrode travels at the speed thus set for contact with the work piece.
 The setting of the parameter is made by input in the range of 0 to 255.
- ◆ **ST ERROR BAND**
 This parameter is used to set the allowable variation in the measurements of the position at which to cause the electrode to come into contact with the work piece during its ST operation executed by G80 command.
 The setting of the parameter is made by decimal input in mm.

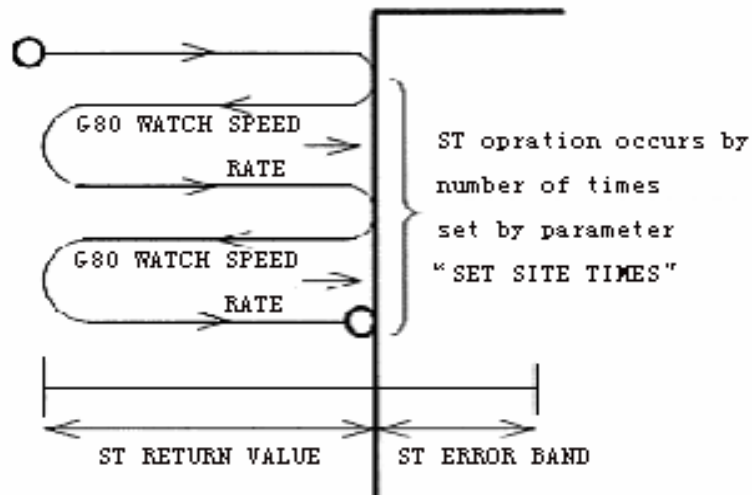


Fig. 12.2.5.3 Setting of ST Movement

◆ G80 DRY IGNORE

This parameter is used to select whether to ignore G80 operation when parameter "DRY" is set in "ON" state.

OFF: To execute G80 operation when parameter "DRY" is set in "ON" state

ON : To ignore G80 operation when parameter "DRY" is set in "ON" state

12.2.6 FLAG SET.AXIS SPEC SET

The "AXIS SPEC SET" sub mode is used to set the parameters necessary to determine the motor specifications for each axis, as well as the parameters for its backlash and pitch offset. This sub mode can be selected by pressing the [AXIS] button, causing the screen to display such a screen image as illustrated in Fig. 12.2.6, which allows the above-mentioned parameters to be set for the X axis. The Fig. 12.2.6 screen can be changed for the X, Y, Z and U axes in order by pressing the [AXIS] button

NOTE! The "*" marked parameters, which are specific to the machine, have been set before its shipment from the factory, not requiring setting change by the user.

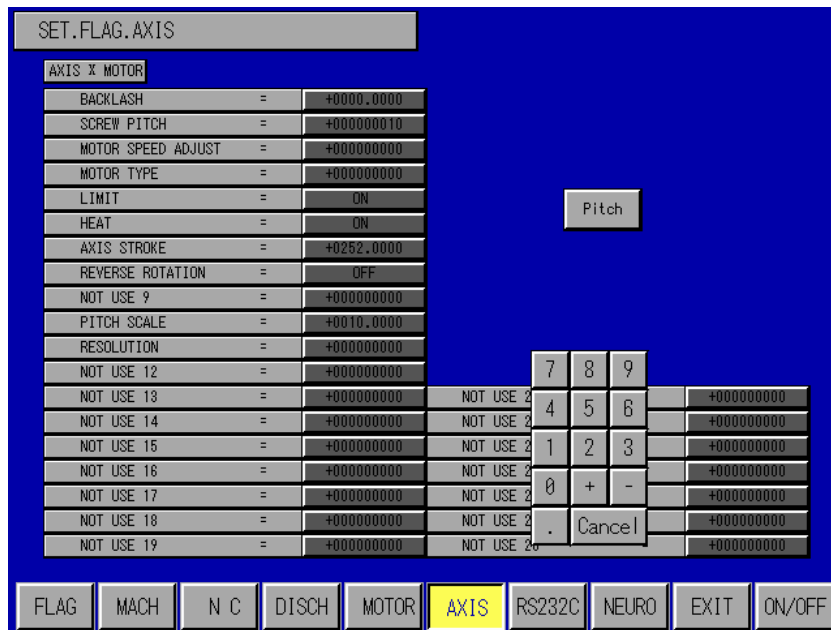


Fig.12.2.6 SET.FLAG SET.AXIS SPEC SET Screen

◆ *BACKLASH

This parameter is used to set the backlash in the axis caused when its feed direction is reversed.

The setting of the parameter is made by decimal input in mm.

If this parameter is set for the revolution axis, its backlash offset keeps original value.

Number of pulses to be offset x 10. For example, when 5 pulses is to be inputted for offset, set the parameter at "50".

◆ *SCREW PITCH

This parameter is used to set the amount of the axis feed to be effected by output of 1 pulse to the motor.

The relationship between the setting of the parameter and the axis feed amount is five by the following expression:

Axis feed amount=10÷SCREW PITCH input data.

[Example]:

100 → Axis feed of 0.1 pm by output of 1 pulse to the motor

10 → Axis feed of 1.0 pm by output of 1 pulse to the motor

8 → Axis feed of 1.25 pm by output of 1 pulse to the motor

1. This setting is specific to the machine, undergoing no change in the axis feed

NOTE! amount per pulse by screw pitch change.

2. Improper setting of the screw pitch causes the axis to fail to operate properly.

◆ * MOTOR SPEED CONTROL

This parameter is used to control the DC pulse motor axis feed speed. Apart from SET.FLAG SET.MOTOR SPEC SET sub mode speed control, the setting of this parameter has an influence on the speed of the axis feed in all operations except cutting.

The parameter, when set at "0", causes the axis to be fed at the maximum speed, decreasing the axis feed speed as its setting is larger.

The setting of the parameter is made by input in the range of 0 to 255.

◆ *MOTOR TYPE

This parameter is used to set the type of the motor used for the axis feed.

[Example] MOTOR TYPE = 0: DC motor

= 1: Pulse motor (10 phase motor)

= 2: Pulse motor (8 phase motor)

= 3: Pulse motor (4 phase motor)

◆ *LIMIT

This parameter is used to decide whether to cause LIMIT ERROR when the axis operation is subjected to input of the machine limit switch.

OFF: To be selected when the axis is not provided with limit switch.

ON: To be selected when the axis is provided with limit switch.

◆ *HEAT

This parameter is used to decide whether to cause HEAT ERROR when the axis operation is subjected to input of the machine HEAT switch.

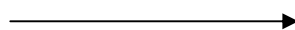
OFF: To be selected when the axis is not provided with HEAT switch.

ON: To be selected when the axis is provided with HEAT switch.

◆ *AXIS STROKE

This parameter is used to set the stroke of the axis from its (-) limit to its (+) limit in 0.1 pm. If the axis comes into contact with its (+) limit for its initial stroke limit, this setting is used as a basis to determine its coordinate value.

Axis feed amount by 1 pulse output 300000 pulses



No. of pluses input in AXIS STROKE

Axis stroke 300mm Set AXIS STROKE by input of 300.0000.

◆ *REVERSE ROTATION

This parameter is used to set the direction in which to feed the axis relative to the rotation of the motor in the forward direction.

OFF: To feed the axis in (-) direction relative to the rotation of the motor in the forward direction.

ON: To feed the axis in (+) direction relative to the rotation of the motor in the forward direction.

◆ *PITCH SCALE

This parameter is used to set the interval at which to perform the screw pitch offset

[Example]

Setting parameter PITCH SCALE by input of 10.0000 causes the screw pitch offset to be performed at an interval of 10mm.

◆ *RESOLUTION

This parameter is used to set the number of pulses required for one revolution of an axis when the axis is set as a revolution axis on the option screen.

[Example]

Setting parameter RESOLUTION by input of 360000 causes the NC unit to judge the specified axis to have make one revolution when 360000 pulses are output to the motor.

* The setting of an axis as a revolution axis can be made using parameter "AS SIGN" on the option screen.

12.3 CHECK

The 'Check' is the function to check the input/output units.

The "Check" function, not necessary to operate as long as the machine is working in normal condition, can be selected by pressing the [CHECK] button, causing the screen to display such a screen image as illustrated in Fig. 12.3.

SET · CHECK · IN/OUT								
ADR	CENTENTS	DATA	ADR	CENTENTS	DATA	ADR	CENTENTS	DATA
000	ON TIME	= **	030	CP	= **	056	PIKA2 (IP0)	= **
002	OFF TIME	= **	032	BC	= **	058	PIKA3 (IP7)	= **
004	AJC SELECT	= **	034	AJCH	= **	05A	NOW6	= **
008	IP SELECT	= ****	036	HTI	= **	05C	R_AXIS_CLUTCH	= **
008	HP/SV/MA	= ****	038	NW	= **	05E	IP15	= **
00A	OC/LD/AL	= ****	03A	GR	= **	**	*****	**
00C	NOT USE	= **	03C	FAJC	= **	**	*****	**
00E	X MOTOR	= **	03E	SHPS	= **	**	*****	**
010	Y MOTOR	= **	040	MACH	= **	**	*****	**
012	Z MOTOR	= **	042	NOMAN	= **	**	*****	**
014	U MOTOR	= **	044	BUZZER	= **	**	*****	**
016	V MOTOR	= **	046	HIGH HT	= **	A	7 8 9	**
01A	V SELECT	= ****	048	LOW HT	= **	B	4 5 6	**
01C	I SELECT	= ****	04A	TPC ON	= **	C	1 2 3	**
01E	C SELECT	= ****	04C	NEW PIKA	= **	D	0 + -	**
020	LIMIT XYZU	= 3F	04E	PUMP	= **	E	F Cancel	**
020	HEAT	= 7	050	POL	= **	**	**	**
020	ZERO	= 8	052	PIKA0	= **	**	**	**
028	SIG IN	= 8000	054	PIKA1	= **	**	**	**

A0000H	A0060H	A00A0H	A00E0H	A0130H				EXIT
--------	--------	--------	--------	--------	--	--	--	------

Fig.12.3

12.3.1 IN/OUT (A0000H)

Pressing the [A0000H] button causes the screen to change to display such screen image as illustrated in Fig. 12.3.1.

SET · CHECK · IN/OUT								
ADR	CENTENTS	DATA	ADR	CENTENTS	DATA	ADR	CENTENTS	DATA
000	ON TIME	= **	030	CP	= **	056	PIKA2 (IP0)	= **
002	OFF TIME	= **	032	BC	= **	058	PIKA3 (IP7)	= **
004	AJC SELECT	= **	034	AJCH	= **	05A	NOW6	= **
006	IP SELECT	= ****	036	HTI	= **	05C	R_AXIS_CLUTCH	= **
008	HP/SV/MA	= ****	038	NW	= **	05E	IP15	= **
00A	OC/LD/AL	= ****	03A	OR	= **	**	*****	**
00C	NOT USE	= **	03C	FAJC	= **	**	*****	**
00E	X MOTOR	= **	03E	SHPS	= **	**	*****	**
010	Y MOTOR	= **	040	MACH	= **	**	*****	**
012	Z MOTOR	= **	042	NOMAN	= **	**	*****	**
014	U MOTOR	= **	044	BUZZER	= **	**	*****	**
016	V MOTOR	= **	046	HIGH HT	= **	A	7 8 9	**
01A	V SELECT	= ****	048	LOW HT	= **	B	4 5 6	**
01C	I SELECT	= ****	04A	TPC ON	= **	C	1 2 3	**
01E	C SELECT	= ****	04C	NEW PIKA	= **	D	0 + -	**
020	LIMIT XYZU	= 8F	04E	PUMP	= **	E	F Cancel	**
020	HEAT	= 7	050	POL	= **	**	**	**
020	ZERO	= 8	052	PIKA0	= **	**	**	**
028	SIG IN	= 8000	054	PIKA1	= **	**	**	**

A0000H	A0060H	A00A0H	A00E0H	A0130H				EXIT
--------	--------	--------	--------	--------	--	--	--	------

Fig.12.3.1 SET.CHECK.IN/OUT (A0000h)Screen

12.3.2 IN/OUT (A0060H)

Pressing the [A0060H] button causes the screen to change to display such screen image illustrated in Fig. 12.3.2.

SET · CHECK · IN/OUT								
ADR	CENTENTS	DATA	ADR	CENTENTS	DATA	ADR	CENTENTS	DATA
060	TOOL COUNT =	00	08B	MCODE IN 73 =	80	09E	ARM HOOK =	00
061 =	80	08C	UAXIS ORQ SW =	00	09F	ARM S LMT(RL) =	80
062 =	00	08D	TC POINT SW =	80	***	*****=	**
063 =	80	08E	CLAMP =	00	***	*****=	**
064 =	00	08F	UNCLAMP =	80	***	*****=	**
065 =	80	090	ARM T LMT(MQ) =	00	***	*****=	**
066 =	00	091	ARM T LMT(HD) =	80	***	*****=	**
067 =	80	092	ARM V LMT(UP) =	00	***	*****=	**
080	MCODE IN 50 =	00	093	ARM V LMT(DN) =	80	***	*****=	**
081	MCODE IN 51 =	80	094	ARM H LMT(FR) =	00	***	*****=	**
082	MCODE IN 52 =	00	095	ARM H LMT(RL) =	80	***	*****=	**
083	DRAIN ON =	80	096	MAG PIN LMT =	00	A	7 8 9	**
084	DRAIN OFF =	00	097	TOOL COUNT =	80	B	4 5 6	**
085	FLUSH ON =	80	098	TOOL EXIST =	00	C	1 2 3	**
086	FLUSH OFF =	00	099	TOOL FIRST =	80	D	0 + -	**
087	OIL ON =	80	09A	ARM T LMT(MD) =	00	E	F Cancel	**
088	OIL OFF =	00	09B	ARM V LMT(M1) =	80			**
089	TC POINT LMT =	80	09C	ARM V LMT(M2) =	00			**
08A	UAXIS ORQ LMT =	00	09D	ARM S LMT(FR) =	80			**

A0000H	A0060H	A00A0H	A00E0H	A0130H				EXIT
--------	--------	--------	--------	--------	--	--	--	------

Fig.12.3.2 SET.CHECK.IN/OUT (A0060h)Screen

12.3.3 IN/OUT (A00A0H)

Pressing the [A00B0H] button causes the screen to change to display such screen image as illustrated in Fig.12.3.3

SET · CHECK · IN/OUT								
ADR	CENTENTS	DATA	ADR	CENTENTS	DATA	ADR	CENTENTS	DATA
0A0	MCODE OUT 10	= **	0B3	MAQA LOCK PIN	= **	0D4	- - - - -	= 00
0A1	MCODE OUT 11	= **	0B4	MAQAQIN TURN	= **	0D5	ST_CANCEL SW	= 80
0A2	MCODE OUT 12	= **	0B5	HEAD UNCLAMP	= **	0D6	FLOAT SENS	= 00
0A3	MCODE OUT 13	= **	0B6	STOPPER OFF	= **	0D7	PUMP ON SW	= 80
0A4	MCODE OUT 14	= **	0B7	CLAMP LAMP	= **	0D8	ACK SW	= 00
0A5	MCODE OUT 15	= **	0B8	UNCLAMP LAMP	= **	0D9	OFF SW	= 80
0A6	MCODE OUT 16	= **	0B9	HOOK OPEN/CLS	= **	0DA	HALT SW	= 00
0A7	CLAMP LED	= **	0BA	STOPPER ON	= **	0DB	ENTER_RST_SW	= 80
0A8	UNCLAMP LED	= **	0BB	PUMP2	= **	0DC	PUMP_OFF_SW	= 00
0A9	TC POINT LED	= **	0BC	NOT USE	= **	0DD	THERMO SENS	= 80
0AA	TC PNT CLAMP	= **	0BD	MECHA CLAMP	= **	0DE	- - - - -	= 00
0AB	TC PNT LK PIN	= **	0BE	ARM TURN2	= **	A	7 8 9	= 80
0AC	FLUSH	= **	0BF	ARM VERTICAL2	= **	B	4 5 6	= **
0AD	UAXIS ORG LED	= **	0C0	JOQ XYZU	= 00	C	1 2 3	= **
0AE	U_AXIS_CLUTCH	= **	0C4	MFR 0,1/2/3	= 00	D	0 + -	= **
0AF	U_AXIS_LOCK	= **	0D0	- - - - -	= 00	E	F Cancel	= **
0B0	ARM HORIZON	= **	0D1	- - - - -	= 80			= **
0B1	ARM TURN	= **	0D2	- - - - -	= 00			= **
0B2	ARM VERTICAL	= **	0D3	- - - - -	= 80			= **

A0000H	A00B0H	A00A0H	A00E0H	A0130H				EXIT
--------	--------	--------	--------	--------	--	--	--	------

Fig. 12.3.3 SET. CHECK. IN/OUT (A00B0H) Screen

12.3.4 IN/OUT (A00E0H)

Pressing the [A00E0H] button causes the screen to change to display such screen image as illustrated in Fig.12.3.4.

SET · CHECK · IN/OUT								
ADR	CENTENTS	DATA	ADR	CENTENTS	DATA	ADR	CENTENTS	DATA
0E0	PATLIGHT	= **	112	TANK_HALF	= 00	**	*****	**
0E1	PATLIGHT ERR	= **	113	TANK_PUMP	= 80	**	*****	**
0E2	= **	114	TANK_UP_SW	= 00	**	*****	**
0E3	= **	115	TANK_DOWN_SW	= 80	**	*****	**
0E4	HSPC_UNIT	= **	116	TANK_AUTO_SW	= 00	**	*****	**
0E5	01_UNIT	= **	117	TANK_MANU_SW	= 80	**	*****	**
0E6	= **	118	TANK_SET_SW	= 00	**	*****	**
0E7	= **	120	PW_ON_OFF SIQ	= 01	**	*****	**
0E8	= **	121	SOURCE_OF SIQ	= 80	**	*****	**
0E9	= **	122	ALARM 1	= 00	**	*****	**
0EA	= **	123	ALARM 2	= 80	**	*****	**
0EB	= **	124	ALARM 3	= 00	**	*****	**
0EC	= **	125	ALARM 4	= 80	**	*****	**
0ED	= **	126	ALARM 5	= 00	**	*****	**
0EE	= **	127	ALARM6	= 80	**	*****	**
0EF	= **	**	*****	= **	**	*****	**
0F0	S CODE	= **	**	*****	= **	**	*****	**
110	TANK UP	= 00	**	*****	= **	**	*****	**
111	TANK DOWN	= 80	**	*****	= **	**	*****	**

A0000H	A0060H	A00A0H	A00E0H	A0130H				EXIT	
--------	--------	--------	--------	--------	--	--	--	------	--

Fig. 12.3.4 SET. CHECK. IN/OUT (A00E0H) Screen

12.3.5 IN/OUT (A0130H)

Pressing the [A0130H] button causes the screen to change to display such screen image as illustrated in Fig.12.3.5.

SET · CHECK · OUT								
ADR	CENTENTS	DATA	ADR	CENTENTS	DATA	ADR	CENTENTS	DATA
130 =	8000	14F	CANCEL LED =	**	**	*****=	**
132 =	8000	150	POWER_ON LED =	**	**	*****=	**
134 =	8000	151	POWER_OFF LED =	**	**	*****=	**
136 =	8000	152	SOURCE_OFF LED=	**	**	*****=	**
140	COORD MD LED =	**	153	FAN_ON_OFF LED=	**	**	*****=	**
141	IDOU_MODE LED =	**	154	TANK_UP =	**	**	*****=	**
142	CENTERING LED =	**	155	TANK_DOWN =	**	**	*****=	**
143	NEURO_MD LED =	**	**	*****=	**	**	*****=	**
144	MANUAL_MD LED =	**	**	*****=	**	**	*****=	**
145	MULT.CAV.M LED=	**	**	*****=	**	**	*****=	**
146	USER_MODE LED =	**	**	*****=	**	**	*****=	**
147	N.LERN MD.LED =	**	**	*****=	**	**	*****=	**
148	PROGRAM_M LED =	**	**	*****=	**	**	*****=	**
149	MACHINE_M LED =	**	**	*****=	**	**	*****=	**
14A	SETTING MD LED=	**	**	*****=	**	**	*****=	**
14B	BLANK_MD LED =	**	**	*****=	**	**	*****=	**
14C	DISP MD LED =	**	**	*****=	**	**	*****=	**
14D	PUMP LED =	**	**	*****=	**	**	*****=	**
14E	FILE_OPE LED =	**	**	*****=	**	**	*****=	**

A0000H	A0060H	A00A0H	A00E0H	A0130H				EXIT
--------	--------	--------	--------	--------	--	--	--	------

Fig. 12.3.5 SET. CHECK. IN/OUT (A0130H) Screen

PART 5 FILE OPERATION AND OTHER

This part will introduced the operation methods and the properties, operation methods of many modes in the CNC supply unit.

This part consists of following chapters:

Chapter 13 File Operation

Chapter 14 List of Error/Halt, Message and Comments.

Chapter 13 File Operation

This chapter describes the function of saving, loading and deleting the parameter data input as a file in the mode such as "Alignment", "Multiple cutting", "Assist cutting", "Manual cutting".

13.1 File Operation

If there is the [File] button in the mode screen such as alignment, multiple cutting, assist cutting, manual of the Z system, the file operation can be executed in these modes(The file operation can not be executed of there is no [File] button.). Press the [File] button, the screen displays the file operation scene as Fig.13.1.1.

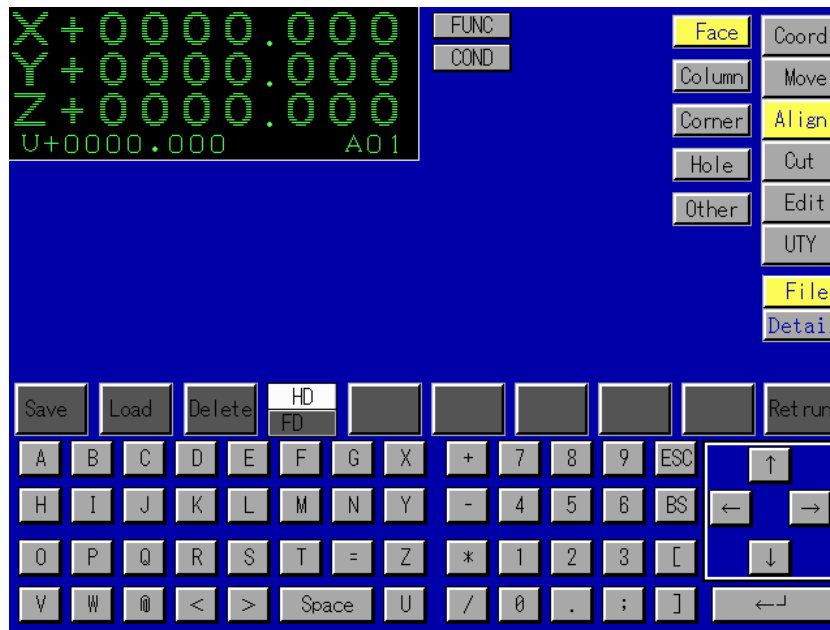


Fig.13.1.1

The simulation keyboard to be used for file operation is in the low half part of the screen. The save, loading and deleting of mode file can be operated by pressing the function buttons character buttons, digit and direction buttons. The file operation will be introduced with the alignment approach screen as an example.

13.2 File Operation Method

With the approach face alignment screen as an example, press the [File] button in this Screen displays as Fig.13.2.1.

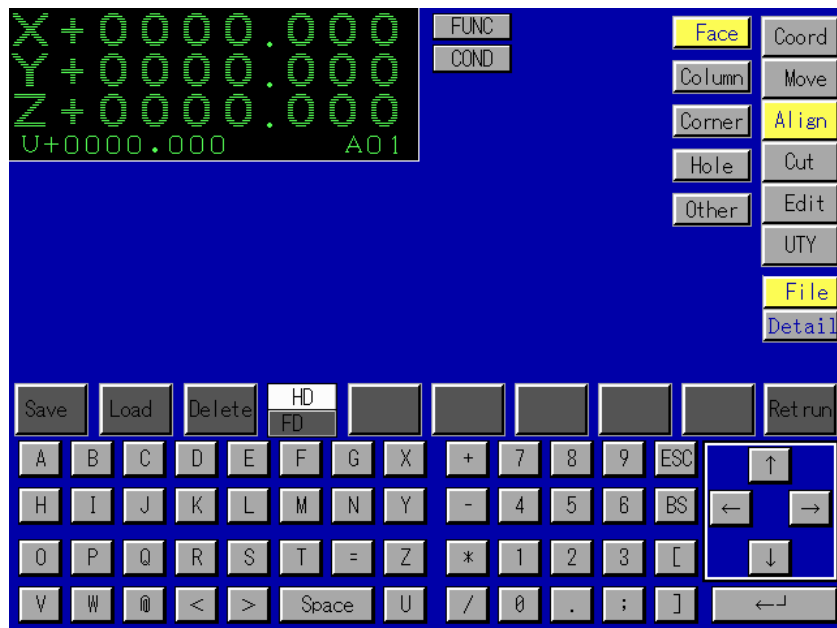


Fig.13.2.1

The operation method of each function button will be introduced as follows.

- [Save]

This key is used to save the data input as a file. Press the [Save] button, the screen changes to Fig.13.2.2.

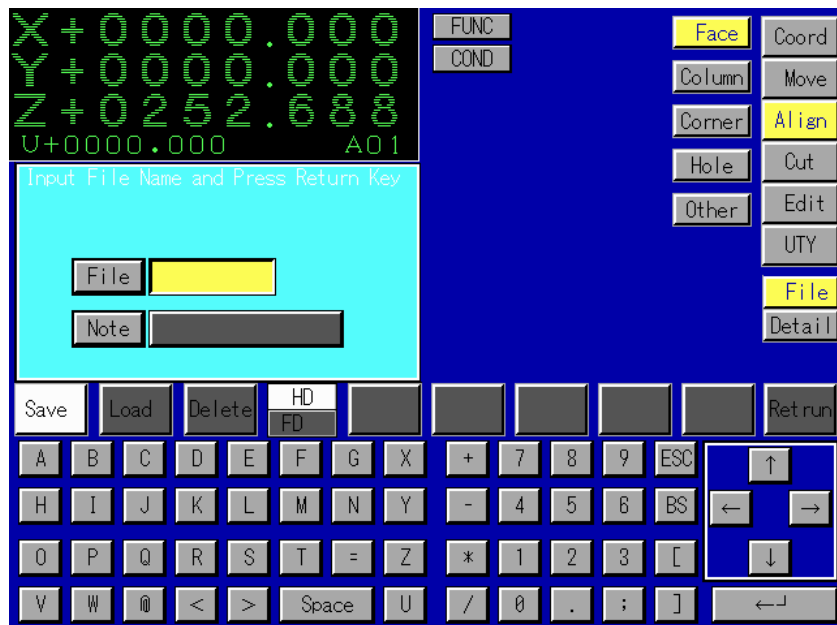


Fig.13.2.2

Press the alphabet or digit button to input the file name. Press the [CR] key, the condition data of the current approach face alignment mode will be saved in a new set file. If you want to input the

comments of file, you can press the [↓] key after inputting the file name and input in "Comment" column. Finally, press the [CR] key to complete the save operation.

- [Load]

This key is used to select the needed file from all saved files in this mode and load this File. Press the [Load] button, the screen displays as Fig.13.2.3.

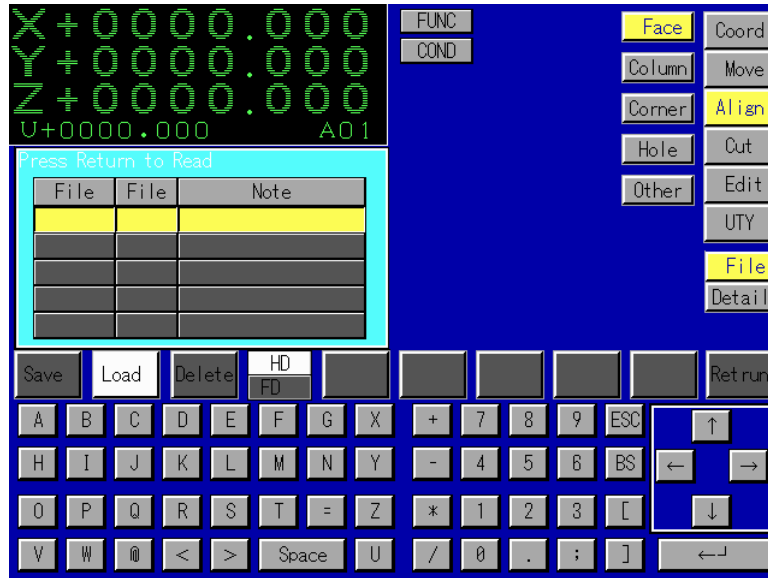


Fig.13.2.3

Use the [↑], [↓] key to select the needed file name. After pressing the [Data input] button to confirm, the values of each input item in the approach face screen will change to the values of loaded file.

- [Delete]

This key is used to delete the saved file. Press the [Delete] button, the screen displays as Fig.13.2.4.

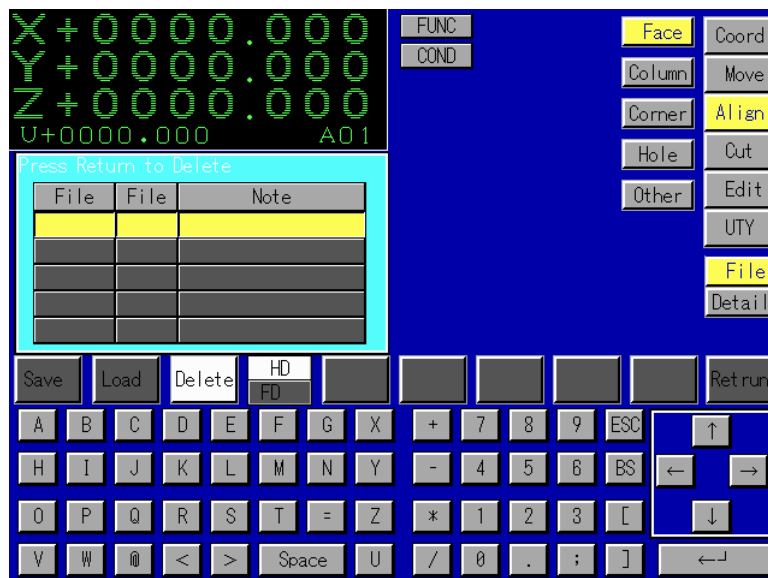


Fig.13.2.4

Press the [↑], [↓] button to select the file name to be deleted. After pressing [CR] to confirm, the file will be deleted.

- [HD/FD]

The [HD/FD] state of the key can be selected alternately each time it is pressed. If "HD" state is selected, the current file operation will be executed in the hard disk. If "FD" state is selected, the current file operation will be executed in the floppy disk.

- [Return]

After pressing the [Return] key, the file operation screen disappears and returns to the last screen.

[Example of file operation]

Save the data, as a file with a file name of "DMEC" and a comment of "Z1" Then delete this file.

(1) Press the [File] button.

(2) Press the [Save] button.

(3) Press the [D], [M], [E], [C] buttons.

(4) After pressing the [↓] key to move to the comment column, press the [Z], [1] buttons

(5) Press the [CR] key to save the file.

(6) Press the [Delete] button.

(7) Use the [↑], [↓] keys to select the file name DMEC.

(8) Press the [CR] button to complete the "Delete" operation.

CHAPTER 14 LIST OF ERROR/HALT MESSAGES AND COMMENTS

14.1 Introduction

This chapter describes the messages and comments to be displayed during the operation of the NF power supply unit when it develops such errors as represented by these messages and comments. The NF power supply system is equipped with a self-checking function designed to classify the status of the EDM into three levels to display an error according to the level to which the machine status corresponds as shown below:

◆ Error message

If the machine develops such an error as makes the continuation of its operation impossible, an error message indicating the occurrence of the error will be displayed, causing it to stop its operation. Therefore, if such an error message appears on the screen, check the contents of the message before pressing the [ACK] key. Remove the error and press the [ENT] key to resume the machine operation.

◆ Halt message

If the machine develops such an error as allows resumption of its operation, a halt message indicating the occurrence of the error will be displayed, causing it to stop its operation on a temporary basis. Therefore, if such an error message appears on the screen, check the contents of the message and then remove the error before pressing the [ENT] key, which, if the error has been removed allows the machine operation to be resumed. To stop the operation, press the [OFF] key.

◆ COMMENT message

If the machine develops such an error as allows continuation of its operation or such a condition as requires a comment to be made on the condition, a comment message will be displayed to call the operator to pay attention to the error or condition.

14.2 Message Display

◆ Direct display on the screen

The above messages appears on the screen in such ways as described below if the selected screen mode is one that allows the machine operation to be executed.

Error message..... Appears in a red zone at the bottom of the screen.

Halt message..... Appears in a green zone at the bottom of the screen.

Comment message..... Appears in a zone at the left center of the screen.

◆ Indirect display on the screen

When a comment message is displayed on the screen, another comment message occurs at the same time, the latter will appear on the screen in such a way as describe in the following.

14.3 Causes of Error/Halt Messages and Their Remedies

[Reference] “***” indicates the file name.

[A]

◆ Arc has been detected. Press [ENT] key to continue clean surface or [OFF] key to stop.

Cause: This message is displayed when the arc detector detects arc during the machining operation.

Remedy : Check the machining condition, flushing and other relevant factors before cleaning the work piece surface to be cut.

◆ Area checksum error has occurred. Press [ACK] key.

Cause: This message is displayed when an error occurs between SYSTEM1 and SYSTEM2.

Remedy: Check the board and connector. If the error occurs frequently, contact with DMEC.

◆ Air pressure has run low. Press [ENT].

Cause: This message is displayed when the air pressure has run low.

Remedy: Check the air pressure and press the [ENT] key to reexecute the operation or the [OFF] key to discontinue it.

◆ ATC offset setting is wrong. Press [ACK] key

Cause: This message is displayed when the set value of the exchange site of ATC is minus, or the stroke is exceeded, so ATC offset setting is wrong.

Remedy: Set rightly sites of X,Y,Z axes of exchange sites of ATC

◆ ATC mechanical malfunction. Press [ACK] key.

Cause: This message is displayed when input from sensor in ATC execution is all under 10 seconds.

Remedy: Check the specified sensor, make it return with hand.

◆ Axis operation cannot be executed as POWER is OFF. Press [ACK] key.

Cause: This message is displayed when axis operation is executed as power is off.

Remedy : Turns on power. Automatically check why the CNC supply unit turns off, and remove the error.

[B]

- ◆ Circular command cannot be used for offset approach (IN/OUT). Press [ACK] key

Cause: This message is displayed when offset approach is executed by circular interpolation.

Remedy: Execute offset approach by straight interpolation. (3.5 p m or more for straight interpolation)

- ◆ Circular command specified radius is too small. Press [ACK] key.

Cause: This message is displayed when G02/G03 command is executed with I/J/K command not executed or the radius set at "0".

Remedy: Execute I/J/K command or if the radius is set at "0", execute G01 for straight interpolation. The minimum executable radius is 0.1 ~.fi' for the G type and 1 µm for the C type.

- ◆ CONDITON No. is not found. Press [ACK] key

Cause: This message is displayed when machining condition is used in NC program with its corresponding Machining Condition No. not present either in the private file involved or the COND file.

Remedy: Check that the Condition No. is correct. Alternatively, create the specified condition No. in the private file involved or the COND file.

- ◆ Check magazine pot. After check, press [ENT] key to continue.

Cause: This message is displayed when the ATC operates for tool change with no tool present in the magazine pot involved in the tool change.

Remedy: Fit the magazine pot involved in the tool change with a tool to be loaded into the head and press the [ENT] key. To stop the ATC operation, press the [OFF] key.

- ◆ Coordinate setting be executed in this coordinate system. Press [ACK] key.

Cause: This message is displayed when G92/G97 command is executed in the machine coordinate system (G959).

Remedy: G959 coordinate system cannot be used for coordinate setting. Therefore, execute another "work coordinate system" command code or delete the G92/G97 command.

[D]

- ◆ Data exceeds specified limits during processing. Press [ACK] key.

Cause: This message is displayed when trigonometric function, assignment statement or other data is programmed with input of any numeric value not defined for the system.

Remedy: Check the processing of the data, where the error has occurred data of more than 9 digits can be input.

- ◆ Disk is write-protected. Remove write-protect and press [ENT] key.

Cause: This message is displayed when data is written onto a floppy disk when it is in write-protected state.

Remedy: Set the write-protect switch on the back of the floppy disk to the "write Enable position".

- ◆ Disk has not been set, set disk and press [ENT] key, Press [OFF] key to stop.

Cause: This message is displayed when the [ENT] key is pressed with no floppy disk set in the disk drive.

Remedy: Set a floppy disk in the disk drive and press the [ENT] key operation, press the [OFF] key.

- ◆ Data is too large (small). Press [ACK] key.

Cause: This message is displayed when any command is executed with data input exceeding the number of digits specified for it.

Remedy: Correct the NC program.

- ◆ Division by "0" has been made, Press [ACK]key.

Cause: This message is displayed when NC program contains calculation involving division by "0".

Remedy : Check the calculation.

[E]

- ◆ Error has occurred. Press [ACK] key

(** indicates the error No.)

Cause: This message is displayed when any error occurs, the message of which is not defined.

Remedy : Contact DMEC for consultation.

- ◆ Electrode moves to HEAT limit switch. Remove electrode and press [ACK] key

Cause: This message is displayed when the axis has moved to the HEAT switch (limit).

Remedy : Set the parameters "HEAT" and "LIMIT" on the SET.FLAG SET-AXIS SPEC sub mode screen at "OFF" and execute the JOG operation or other similar commands to move the axis from the HEAT switch in the reverse direction After this movement is completed be sure to return the parameter "HFAT" and "LIMIT" to their original states.

[F]

- ◆ *** file is not found. Press [ACK] key.

Cause: This message is displayed when Q command is executed for execution of a file on the disk although the file is not present on the disk.

Remedy: Replace the disk with one which contains the file involved executed or use the current user disk to create the file to be executed for its execution on that disk.

- ◆ ***file data is wrong. Press [ACK] key

Cause: This message is displayed when the data of the file involved has been destroyed.

Remedy: Check the contents of the file involved and create its data properly

- ◆ *** file format is wrong. Press [ACK] key.

Cause: This message is displayed when the COND/OFFSE NC format is wrong.

Remedy: Correct the format of the file involved.

- ◆ Failure in 10 starting has occurred. Turn SOURCE ON again.

Cause: This message is displayed when POWER is turned ON with failure in I/O loading.

Remedy: Turn off the CNC supply unit temporarily and then turn it on again. If the same error recurs, contact DMEC for consultation.

- ◆ Fluid temperature is too high. Press [ENT] key. Press [OFF] key to stop.
Cause: This message is displayed when the fluid temperature exceeds 55° C.
Remedy: Stop the cutting before the temperature goes down. Conform the set temperature when the machine have oil-cooler device.
Press the [ENT] key to reexecute the operation or the [OFF] key to discontinue it.

- ◆ Fluid level has run low. Press [ENT] key. Press [OFF] key to stop.
Cause: This message is displayed when the fluid level has run low during the operation.
Remedy: Check the pump and the machining tank opening/closing valve before pressing the [ENT] key to reexecute the operation or the [OFF] key to discontinue it.

- ◆ Fire extinguisher has been activated. Press [ACK] key
Cause: This message is displayed when the fluid temperature has risen to such a Investigate the cause of the rise in the fluid temperature and control the fluid be activated.
Remedy: Level as involves possible occurrence of a fire, causing the fire extinguisher to temperature.

[H]

- ◆ HALT STOP. Press [ENT] key to execute or [OFF] key to stop.
Cause: This message is displayed when axis.
Remedy: Press the [ENT] key to reexecute the program or the [OFF] key to discontinue its execution.

- ◆ HEAT STOP has occurred. Press [ACK] key.
Cause : This message is displayed when the table is operated with the POWER switch "ACK" or the motor is overloaded.
Remedy: Press the [ACK] key to clear the error. Then, turn the POWER SOURCE switch "OFF" and then turn the switch "ON" again. If even after SOURCE/POWER "ON", the machine will not operate, check the fuse.

- ◆ Header machining condition is wrong. Press [ACK] key.
Cause: This message is displayed when the machining condition file has been created in any other format than specified.
Remedy: Refer to the CHAPTER "SYSTEM FILE" in the INSTRUCTION MANUAL [CONTROL SYSTEM] to create the machining condition file in a correct format.

◆ Header offset format is wrong. Press [ACK] key.

Cause: This message is displayed when the offset term has been created in any other format than specified.

Remedy: Correct the format of the machining condition and offset term on the NC program.

◆ If statements exceed 1000 in number, Press [ACK] key.

Cause: This message is displayed with "POWER RECOVERY"= 3 set in the SET-FLAG SET-MACHINE SPEC sub mode, the cumulative number of IF statements executed after pressing of the [ENT] key exceeds 1000.

Remedy: Reduce the number of IF statements in the NC program or set parameter "POWER RECOVERY"- 0 in the SET-FLAG SET-MACHINE SPEC Sub mode.

[I]

◆ Initialization of mechanic coordinate failed. Press [ACK] key.

Cause: This message is displayed when initialization of mechanic coordinate failed in the power-up of the power supply. And when power turns on after pressing [ACK] key, initialization of mechanic coordinate is not done, this message is displayed too.

Remedy: Check mechanic coordinate and remove the error.

[L]

◆ Linear cutting command is wrong. Press [ACK] key.

Cause: This message is displayed when G00/G01 code is executed with input of I/J/K command.

Remedy: Delete the I/J/K command from the G00/G01 command block and correct the program using the specified format.

◆ Local machining conditions exceed 100 in number. Press [ACK] key.

Cause: This message is displayed when the number of local machining conditions (set. in a private file) exceeds 100 specified as the maximum allowable limit for such setting.

Remedy: Reduce the number of machining conditions set in the private file by taking measures such as using the machining conditions in the COND file.

◆ Local offset terms exceed 500 in number. Press [ACK] key.

Cause: This message is displayed when the number of local offset terms (set in a private file) exceeds 500 specified as the maximum allowable limit for such setting.

Remedy: Reduce the number of offset terms set in the private file by taking measures such as using the offset terms in the OFFSET file in the SYSTEM file.

◆ LIMIT STOP has occurred. Press [ACK] key.

Cause: This message is displayed when the electrode has come into contact with the machine limit, stopping its operation.

Remedy: Press the [ACK] key to clear the error. Check the electrode travel direction.

[M]

◆ MOO (M01) STOP. Press [ENT] key to execute or [OFF] key to stop.

Cause: This message is displayed when MOO or M01 is executed during the program execution.

Remedy: Press the [ENT] key to continue the program execution or the [OFF] key to discontinue it.

◆ M04 code cannot be used for arc. Press [ACK] key.

Cause: This message is displayed when M04 is executed for arc code(G02,G03)

Remedy: Use M04 only for line code(M01).

◆ M03 is not found. Press [ACK] key.

Cause: This message is displayed when M03 is not found in NC program although parameter M03 in the SET. FLAG SET-FLAG sub mode has been set so that if ARC, LIMIT STOP or ST STOP occurs, the NC program will be searched for M03, at which position to perform its execution.

Remedy: Insert a M03 block into the program at an appropriate position. Alternatively, set the parameter "M03" in the SET.FLAG SET. FLAG sub mode at "0".

◆ M98 loop frequency cannot be set at 0. Press [ACK] key.

Cause: This message is displayed when the calling frequency of sequence number is 0 in calling M98(calling sub-program).

Remedy :Do not execute "M98N****L0".

◆ M99 is not found. Press [ACK] key.

Cause: This message is displayed when M98 is executed for call of a subprogram, which contains no M99 block, causing the program to be decoded to the end.

Remedy : Check the program for presence of M99 in its subroutine subprogram and add a M99 block to the subprogram if no such block is present in it.

◆ Machine coordinate is minus (-). Press [ACK] key.

Cause: This message is displayed when any axis is fed or its coordinate is set with its machine coordinate left on the (-) side.

Remedy : Execute JOG operation or G81 command to feed the axis to its limit.

NOTE! When the CNC supply unit is turned "ON", be sure to feed the axis to their limits to set their machine origins.

◆ More than 20 programs have been executed. Press [ACK] key.

Cause: This message is displayed when the number of files execute by MACRO function and M98 has exceeded 20.

Remedy: Correct the total number of files to 20 or less.

◆ Machine Lock is ON. Press [ACK] key.

Cause: This message is displayed when ATC is executed with "MACHINE LOCK" set in "ON" state.

Remedy: Set "MACHINE LOCK" in "OFF" state or check the NC program to correct it.

◆ Magazine pot contains tool. Remove tool and press [ENT] key. Press [OFF] key to stop.

Cause: This message is displayed when ATC is executed for tool change to house the tool in the magazine pot, which contains another tool.

Remedy: Remove the tool contained in the magazine pot into which to house the tool for tool change and press the [ENT] key. To stop the operation, press the [OFF] key.

◆ Main program contain M99 Press [ACK] key

Cause: This message is displayed when M99 exists in the main program.

Remedy: Check that the program execution has not been started from the subprogram delete M99.

◆ Main program contains M199.

Cause: This message is displayed when the main program contains M199, which is a code for return of Q ASSIST with sequence No. specified for execution.

Remedy: Delete M199 from the main program.

[N]

◆ NC program format is wrong. Press [ACK] key.

Cause: This message is displayed when any of the blocks between the cursor indicated one and the fourth ahead of on has been programmed in a wrong command format.

Remedy: Create the NC program in a correct command format.

◆ NC program is too large. Press [ACK] key

Cause: This message is displayed when the size of the NC program executed and exceeds approximately 500Mbytes on a cumulative basis.

Remedy: Create a program so that its size does not exceeds 500Mbytes on a cumulative basis.

◆ No circular cutting for movement of this axis. Press [ACK] key.

Cause: This message is displayed when circular interpolation is executed with optional parameter "ASSIGN" set to command simultaneous movement of "Taper Axis (3-4)" and "Detail (6-8)".

Remedy: Execute the circular interpolation not commanding simultaneous movement of these axes, but programming their movements in two separate blocks.

◆ No data file exists in drive 2. Set disk and press [ENT] key.

Cause: This message is displayed when no data file exists in drive 2.

Remedy: In the case of the hard disk built-in version, the occurrence of the error suggests possible failure in its software system installation. If the error occurs in the case of the floppy operated version, set a user disk and press the [ENT] key.

◆ Nesting exceeds 50 times. Press [ACK] key.

Cause: This message is displayed when the frequency of subprogram call using M98 exceeds 50 times.

Remedy: Correct the NC program so that the nesting does not exceed 50 times.

◆ Negative value can't be input. Press [ACK] key.

Cause: This message is displayed when any negative value is input in S code

Remedy: Correct the NC program with input of positive data in S code.

Learning error (-1)

Cause: This message is displayed when the setting of roughness is wrong in neuron learning.

Remedy: Check the setting of roughness.

◆ Learning error (5)

Cause: This message is displayed when the roughness level cannot be more higher in neuro learning.

Remedy: Cancel the selection of [ROUGH] in neuro learning mode.

◆ Learning error (6)

Cause: This message is displayed when neuro learning.

Remedy: Cancel the selection of [ROUGH] in neuro learning mode.

[O]

◆ [OFF] key has been activated to stop. Press [ACK] key.

Cause: This message is displayed when the [OFF] key has been activated to stop the program execution completely.

Remedy: Press the [ACK] key to release the machine from this state.

◆ OFFSET No. is not found. Press [ACK] key.

Cause: This message is displayed when offset is used in NC program with its corresponding OFFSET No. not present either in the private file involved or the OFFSET file.

Remedy: Check that the OFFSET No. is correct. Alternatively, create the specified OFFSET No. in the private file involved or the OFFSET file.

◆ Original limit is not found. Press [ACK] key'.

Cause: This message is displayed when arriving the mechanical limit, the mechanical original is not found.

Remedy: Check the wiring of motor driver and the setting of speed of original search.

[P]

◆ Parameter data exceeds specified range.. Press [ACK] key

Cause: This message is displayed when parameter data exceeds the specified range in executing B code or C code.

Remedy: Modify the parameter date which exceed the specified range, refering the parameter data file and the NC program.

◆ Parenthesized nesting exceeds 19 times. Press [ACK] key.

Cause: This message is displayed when parenthesized nesting exceeds 19 times.

Remedy: Correct the parenthesized nesting to or below 19 times.

- ◆ PITCH or REVOLUTION has been set at "0". Press [ACK] key.
Cause: This message is displayed when the parameter "REVOLUTION" or "SCREW PITCH" on the SET.FLAG SET-AXIS SPEC sub mode screen has been set at "0".
Remedy: Set the parameters "REVOLUTION" and "PITCH SCREW" on the SET. FLAG SET.AXIS SPEC sub mode screen at proper values.

- ◆ Pitch position error. Press [ACK] key.
Cause: This message is displayed when inner data error occurs in pitch offset movement.
Remedy: Check the main board and connector. If this error occurs frequently, please contact with DMEC.

- ◆ Plane selection is wrong. Press [ACK] key.
Cause: This message is displayed when circular interpolation is executed with a wrong plane selected.
Remedy: Check the program.

- ◆ POWER has been turned OFF. Press [ACK] key.
Cause: This message is displayed when the CNC supply unit has been turned OFF during NC program execution, requiring investigation into the cause of the POWER OFF for its elimination.
Remedy: When the parameter "NO-MAN" has been in ON state, the above message will be displayed on the screen if the NO-MAN function is activated. This message will also be displayed on the screen when the CNC supply unit is turned OFF while the NC program is being executed.

- ◆ Processing format is wrong. Press [ACK] key.
Cause: This message is displayed when the block, the processing format of which is wrong, is executed.
Remedy: Check the processing format and correct it.

- ◆ PTR parity error (horizontal) has occurred. Press [ACK] key.
Cause: This message is displayed when the PTR detects an error in the paper tape during its reading operation.
Remedy: Correct the paper tape and check the data displayed on the SET.FLAG SET.232C SPEC sub mode screen.
Check that the connection with the PTR has been made properly. Also clean the light receiver of the PTR device.

[Q]

- ◆ Q command file is not found. Press [ACK] key.

Cause: This message is displayed when Q command specified file is not present on the RAM, user disk or hard disk.

Remedy: Check that the file name is correct, Alternatively, copy the specified file on the RAM, user disk or hard disk.

- ◆ Q command nesting exceeds 7 times. Press [ACK] key

Cause: This message is displayed when Q command (MACRO function, G83, etc.) is used exceeding 7 times.

Remedy: Correct the NC program so that the Q command nesting does not exceed time.

- ◆ Q loop frequency cannot be set at "0".

Cause: This message is displayed when Q ASSIST with sequence No. specified for Execution is executed with the specified sequence No. set for loop frequency of "0" by return statement M199.

Remedy: Do not execute "M199L0".

[R]

- ◆ Read or write from user file is wrong. Press [ACK] key.

Cause: This message is displayed when read or write from the user file has failed.

Remedy: Use another user disk. (Periodically make the backup copy of the user disk.)
Please backup user disk and hard disk regularly.

◆ Return position has not been set. Press [ACK] key.

Cause: This message is displayed when "Reference Point Return" code (G28-G928) or "Return to Coordinate Set Point" (G30) is executed with no reference point setting or coordinate setting made before the execution of the return code.

Remedy: Be sure to make the reference point setting or coordinate setting before executing the return code.

[S]

◆ Servo back has exceeded the set level Clean the surface being machined and press the [ENT] key.

Cause: This message is displayed when the servo back during the cutting operation has exceeded the level set by parameter "SERVO BACK PULSE" displayed on the SET-FLAG SET.DISCHARGE SPEC sub mode screen.

Remedy: Check the machining condition, flushing and other relevant factors before cleaning the workpiece surface to be cut. Alternatively, check that the parameter "SERVO BACK PULSE" and the cutting start position have been set properly.

◆ SOFTWARE LIMIT STOP has occurred. Press [ACK] key.

Cause: This message is displayed when movement exceeds the mechanic STROKE range set in SET. OPERATION

Remedy: Move in the STROKE range with JOG Check axis movement codes in program executing.

Or check the range of SOFTWARE LIMIT STOP in SET. OPERATION.

◆ Specify axis movement direction. Press [ACK] key.

Cause: This message is displayed when movement axes are not specified in the ST STOP movement(G80), limit movement(G81).

Remedy : Specify movement axes.

◆ System error (21) has occurred. Press [ACK] key.

Cause: This message is displayed when the NC unit has suffered an error on its internal software or hardware.

Remedy: Provide DMEC with detailed information about the error (including the NC program involved and parameter data).

◆ System error (22) has occurred. Press [ACK] key

Cause: This message is displayed when the NC unit has suffered an error on its internal software or hardware.

Remedy: Provide DMEC with detailed information about the error (including the NC program involved and parameter data).

◆ System error (37) has occurred. Press [ACK] key

Cause: This message is displayed when the NC unit has suffered an error on its internal software or hardware.

Remedy: Provide DMEC with detailed information about the error (including the NC program involved and parameter data).

◆ System error (38) has occurred. Press [ACK] key.

Cause: This message is displayed when the NC unit has suffered an error on its internal software or hardware.

Remedy: Provide DMEC with detailed information about the error (including the NC program involved and parameter data).

◆ Sequence No. is not found. Press [ACK] key.

Cause: This message is displayed when, although M98/M99/IF/JUMP command is executed, the sequence No. to which to jump is not found.

Remedy: Check the sequence No. and correct it.

◆ Starting and ending points are different in radius.

Cause: This message is displayed when circular interpolation is executed with wrong data input made.

Remedy: Check the circular interpolation command. Alternatively, Check that the parameter "A CIRCLE POINT" in the SET-FLAG SET-NC SPEC sub mode has been set properly.

◆ STRING FILE is wrong. Press [ACK] key.

Cause: This message is displayed when the STRING FILE is created in a wrong format or contains any character impossible to input from the keyboard.

Remedy: Create the STRING FILE in a correct format.

◆ STROKE STOP has occurred. Press [ACK] key.

Cause: This message is displayed when the electrode has traveled beyond its stroke limits.

Remedy: Execute the JOG operation or other commands to feed the electrode within its stroke limits. If this error occurs during the program execution, check the axis feed command or other relevant commands in the C program.

◆ ST STOP has occurred. Press [ACK] key.

Cause: This message is displayed when the electrode has come into contact with the work piece or the program has been executed with the POWER switch "OFF".

Remedy: Press the [ACK] key to clear the error before activating the ST key (which allows the ST function to be ignored) to move the electrode apart from the work piece by executing the JOG operation or other similar commands.

Alternatively, check the POWER switch.

◆ ST ERROR BAND has been exceeded. Press [ACK] key.

Cause: This message is displayed when variation in the position data measured by the ST operation has exceeded the level set by the parameter "ST ERROR BAND" displayed on the SET-FLAG SET-MOTOR SPEC sub mode screen.

Remedy: Check that the parameter "ST ERROR RAND" has been set at a proper value. Alternatively, check that the workpiece and electrode are wet or contaminated. Wipe the workpiece and electrode if wet or contaminated.

◆ Select MFR and press [ACK] key.

Cause: This message is displayed when hardware failure occurs, not causing MFR to set in any of its positions 0 to 3.

Remedy: Set MFR in any of its positions 0 to 3 when the CNC supply unit is turned "ON".

[T]

◆ The current tolerance of the processing condition is over. Please change the COND, and press [ENT] key.

Cause: This message is displayed when the cutting current exceeds the tolerance of the power supply.

Remedy: Change the parameter data, making it sure that the cutting current is in the tolerance of power supply.

◆ The processing of loran is wrong. Press [ACK] key.

Cause: This message is displayed when the combination of cutting parameter of each loran movement, or the setting of loran movement is not right.

Remedy: Check the cutting parameter and setting of loran.

◆ This B code is not available. Press [ACK] key.

Cause: This message is displayed when B code is programmed in any other format than specified for it.

Remedy: Correct the NC program containing the B code.

◆ This C code is not available. Press [ACK] key.

Cause: This message is displayed when C code is programmed in any other format than specified for it.

Remedy: Correct the NC program containing the C code.

◆ This D code is not available. Press [ACK] key.

Cause: This message is displayed when D code is programmed in any other than specified for it.

Remedy: Correct the NC program containing the G code.

◆ This G code is not available. Press [ACK] key.

Cause: This message is displayed when G code is programmed in any other format than specified for it.

Remedy: Correct the NC program containing the G code.

◆ This H code is not available. Press [ACK] key

Cause: This message is displayed when H code is programmed in any other format than specified for it.

Remedy: Correct the NC program containing the H code.

◆ This L code is not available. Press [ACK] key

Cause: This message is displayed when L code is programmed in any other format than specified for it.

Remedy: Correct the NC program containing the L code or subprogram.

◆ This M code is not available. Press [ACK] key

Cause: This message is displayed when M code is programmed in any other format than specified for it.

Remedy: Correct the NC program containing the M code.

◆ This N code is not available. Press [ACK] key.

Cause: This message is displayed when N code is programmed in any other format than specified for it.

Remedy: Correct the NC program containing the N code or subprogram.

- ◆ This O code is not available. Press [ACK] key
Cause: This message is displayed when O code is programmed in any other format than specified for it.
Remedy: Correct the NC program containing the O code or subprogram.

- ◆ This P code is not available. Press [ACK] key
Cause: This message is displayed when P code is programmed in any other format than specified for it .
Remedy: Correct the NC program containing the P code or subprogram.

- ◆ This Q code is not available. Press [ACK] key
Cause: This message is displayed when Q code is programmed in any other format.
Remedy: Correct the NC program containing the Q code or subprogram.

- ◆ This T code is not available. Press [ACK] key.
Cause: This message is displayed when T code is programmed in any other format than specified for it.
Remedy: Correct the NC program containing the T code or subprogram.

- ◆ This code must be used independently. Press [ACK] key
Cause: This message is displayed when MACRO function or G83 is not used in an than specified for it.
Remedy: Program the command, written together with MACRO function or G83 in the same block, in a separate block.

- ◆ This code cannot be used. Press [ACK] key
Cause: This message is displayed when any code is used in any other format than specified for it as in the case where M code or T code specified as "disabled" is used in NC program.
Remedy: Check the NC program and code to a proper one.

- ◆ This character is not available. Press [ACK] key.
Cause: This message is displayed when any character not defined for the NF system is used in NC program.
Remedy: Delete the characters. For characters available for the NF system, refer to section 3.3 "Key Description" in CHAPTER 3 "OPERATION".

14.4 Causes of Comments and Their Remedies

[Reference].....**" indicates the file name

[A]

◆ Arc has occurred. M03 search is in progress

Cause: This message is displayed when arc has occurred with parameter "M03" on the SET-FLAG sub mode screen set at "1" or higher, causing the program to be searched for M03.

Remedy: The above message will automatically disappear from the screen after the execution of M03 is completed.

[C]

◆ Cannot write file. Remove write-protect

Cause: This message is displayed when file is written into a floppy disk when the disk is in write-protected state.

Remedy: Set the floppy disk in Write Enable state.

◆ Cannot write file. Set disk

Cause This message is displayed when file write is executed when there is no user disk set in the drive.

Remedy: Set a floppy disk in the drive properly.

◆ Changeover to oil is in progress. Wait.

Cause: This message is displayed when the fluid is changed over from water to oil.

Remedy: The above message will automatically disappear from the screen after the changeover is completed.

◆ Cutting has ended. Wait until program end.

Cause: This message is displayed when the timer-set cutting operation has ended before the timer-set time is reached. (G85T)

Remedy: Wait until the program ends.

◆ FILE operation cannot be executed unless EDIT operation is ended.

Cause: This message is displayed when the file being currently edited is called while another file is being executed.

Remedy: Load the currently executed file for editing or select another operation mode.

- ◆ Feed of dielectric fluid is in progress. Wait.

Cause: This message is displayed when dielectric fluid is being fed to the machining tank during the oil machining.

Remedy: The above message will automatically disappear from the screen when tank is filled with dielectric fluid, causing the FLOAT switch to be turned with lighting-up of the FLOAT switch's red LED.

[J]

- ◆ JOG operation is disabled

Cause: This message is displayed when the JOG operation cannot be executed while the machine is halted.

Remedy: This case occurs when the machine is halted just after the methane is reexecuted from halting status.

- ◆ JOG operation trace-over has occurred

Cause: This message is displayed when, while the machine is halted, the JOG operation is performed 10 times or more to execute the axis feed in a different direction or the feed of a different axis for Each time.

Remedy: Note that the axis travel by the JOG operation cannot be traced back along its path with the electrode returned in a straight line as it is not stored in the memory if executed 10 times or more.

[L]

- ◆ LIMIT STOP has occurred. M03 search is in progress

Cause: This message is displayed when LIMIT STOP has occurred with parameter "m03" on the SET-FLAG sub mode screen set at "2" or "4", causing the program to be searched for M03.

Remedy : The above message will automatically disappear from the screen when the execution of M03 is completed.

- ◆ Loran position error. Press [ACK] key.

Cause: This message is displayed when inner data errors occur.

Remedy: Check the main board and connector.

If the error occurs frequently, please contact with DMEC.

[M]

- ◆ Memory backup failure has occurred. Note this failure.

Cause: This message is displayed when the memory backup battery has undergone a drop in its voltage.

Remedy: Replace the battery.

[P]

- ◆ Parameter has exceeded range. Input data again.

Cause: This message is displayed when parameter has been set with data input exceeding the range set on the SET-DISCHARGE SPEC sub mode screen.

Remedy: Set the red-indicated parameter again by input of data in the range specified for it. If the input data is in the specified range, check the parameter setting displayed on the SET-DISCHARGE SPEC sub mode screen.

- ◆ POWER has been turned OFF. Press [ACK] key.

Cause: This message is displayed when the automatically POWER OFF is set, and the automatic POWER OFF takes role, or when the NC program is executing, the CNC supply unit turns off.

Remedy: Check the reason of that the CNC supply unit turns off in the executing of NC program, and remove the error.

- ◆ Press [ACK] key to stop buzzer.

Cause: This message is displayed when the warning function of continual buzzer has been turned "ON".

Remedy: Press [ACK] key to stop buzzer

- ◆ Press POWER switch.

Cause: This message is displayed when the CNC supply unit to the machine tool has been turned "ON".

Remedy: Either the JOG or EXECUTE key is not operative for protection of the machine. In a given time period (approximately Four seconds), the message will automatically disappear from the screen, indicating that the machine is ready for operation.

- ◆ Program end.

Cause: This message is displayed when the program has been ended in a proper way (with M02 executed).

Remedy: The above message will automatically disappear from the screen when the next program is executed.

◆ System error (100) has occurred.

Cause: This message is displayed when any change in the machining condition or the like has failed.

Remedy: No remedy for the error is required if the subsequent operation allows the change to be made properly. If the change cannot be executed, load the system all over again or install the system again.

◆ STROKE STOP has occurred. M03 search is in Progress

Cause: This message is displayed when STROKE STOP has occurred with parameter "M03" on the SET-FLAG sub mode screen set at "1" or "4", causing the program to be searched for M03.

Remedy: The above message will automatically disappear from the screen when the execution of M03 is completed.

◆ ST STOP has occurred. M03 search is in progress.

Cause: This message is displayed when ST STOP has occurred with parameter "M03" on the SET-FLAG sub mode screen set at "3" or "4", causing the program to be searched for M03.

Remedy: The above message will automatically disappear from the screen when the execution of M03 is completed.

◆ SOFT LIMIT STOP has occurred. M03 search is in progress

Cause: This message is displayed when SOFT LIMIT STOP has occurred with parameter "M03" on the SET-FLAG sub mode screen set at "1" or "4", causing the program to be searched for M03.

Remedy: The above message will automatically disappear from the screen when the execution of M03 is completed.

◆ Write of file has involved disk I/O error.

Cause: This message is displayed when file is not written in the disk in a proper way.

Remedy: Replace the disk with another user disk.

◆ Write of file has caused disk to be full.

Cause: This message is displayed when file is written into the disk, the remaining capacity of which is insufficient for the size of the file.

Remedy: Delete any unnecessary file(s) from the disk or replace it with another user disk.