

CONTROLLER
OPERATIONAL MANUAL

ANOTRONIC LTD.™

Unit 3, Hollingdon Depot, Stewkley Road, Soulbury, Nr.
Leighton Buzzard, Beds., LU7 0DF. England. Telephone
+44 (01525) 270261 Fax +44 (01525) 270235 E-Mail
sales@anotronic.com
Internet <http://www.anotronic.com>





Table of Contents

TABLE OF CONTENTS	1
CHAPTER 1 OVERVIEW	4
1.1. SYSTEM FRAMEWORK	4
1.2. HARDWARE	5
1.3. SOFTWARE	7
1.4. AXIS CONTROL	9
1.5. UNIT	9
1.6. G CODE LIST	10
CHAPTER 2 PROCEDURE OF TURN ON AND TURN OFF	11
2.1. TURN ON PROCEDURE	11
2.2. TURN OFF PROCEDURE	13
CHAPTER 3 OPERATION INSTRUCTIONS	14
3.1. SCREEN DEPLOYMENT	14
3.2. CONTROL PANEL KEY DESCRIPTION:	17
3.3. I/O OPERATION PANEL	19
3.3.1. Power Control	19
3.3.2. Manual Control	20
3.3.3. Auto Control	21
3.3.4. I/O Point Control	21
3.4. FUNCTION KEY ILLUSTRATION	23
3.4.1. Main Frame	23
3.4.2. Manual	24
3.4.3. Edit	26
3.4.4. Condition Table	27
3.4.5. Depth Table	27
3.4.6. Multi Hole Machine	28
3.4.7. System Parameter	29
3.4.8. RS232	29
CHAPTER 4 FUNCTION MODE DESCRIPTION	30
4.1. BRIEF INTRODUCTION	30
4.2. MANUAL	30
4.2.1. CALIBRATION	31
4.2.1.1. EDGE SEARCH	31



4.2.1.2. TWO POINT	32
4.2.1.3. CENTER OF CIRCLE	37
4.2.1.4. CORNER.....	38
4.2.1.5. MIRROR.....	41
4.2.1.6. Set Check Point	43
4.2.2. HOME SEARCH.....	44
4.2.3. POSITION	45
4.2.4. ELECTRODE.....	47
4.2.5. MOVEMENT SPEED	49
4.2.6. SINGLE MACHINE	50
4.3. COORD	53
4.4. EDIT.....	54
4.4.1. FILE.....	56
4.4.2. DXF File	60
4.4.3. Wire Cut Program.....	63
4.4.4. DIALOG EDIT:.....	66
4.4.4.1. COORDINATE	67
4.4.4.2.POSITION	68
4.4.4.3. MACHINING	70
4.4.4.4. OTHER.....	70
4.4.4.5. M CODE	71
4.4.4.5. SPECIAL PATTERN.....	71
4.4.4.6. SUB PROGRAM.....	71
4.4.4.7. CHANGE GUIDE	72
4.4.5. RECT PATTERN	74
4.4.6. CIRCLE PATTERN	77
4.4.7. SUB-PROGRAM	80
4.5. CONDITION TABLE	82
4.6. DEEP TABLE	86
4.7. MULTI HOLE MACHINE	88
4.7.1. SET	93
4.7.2. Start Point	94
4.7.3.Break Point	96
4.8. SYSTEM PARAMETER.....	98
4.8.1. MCS Parameter.....	99
4.8.2. Custom Parameter.....	100
4.8.3. Alarm List.....	105
4.8.4. Global Data.....	106
4.8.5. Command Data:	107



4.8.6. PITCH Data:	107
4.8.7. AGC:	108
4.8.8. MLC	110
4.8.8.1. I/O BIT	111
4.8.8.2. C BIT	112
4.8.8.3. S BIT	112
4.8.8.4. A BIT	113
4.8.8.5. Timer	114
4.8.8.6. Counter	114
4.8.8.7. Ladder	115
4.8.9. Password	116
4.9. RS232	117
4.9.1. RS232	117
4.9.2. RS232 Port	118
CHAPTER 5 OPERATING EXAMPLE DESCRIPTION	119
CHAPTER 6 PROGRAM CODE	121
6.1. G CODE EXPLANATION	121
6.2. AUXILIARY FUNCTION (M CODE)	132
CHAPTER 7 SYSTEM ALARM MESSAGE	136
7.1. ALARM	136
7.1.1 MOT Alarm	136
7.1.2 OP Alarm	137
7.1.3 INT Alarm	140
7.2. WARNING MESSAGE	144

Chapter 1 Overview

MIRL DRILLING EDM is a PC-BASED CNC controller. It's developed by ITRI machinery Industry Research Laboratories based on intelligent CNC electro-discharge machine's function requirements, and through years' dedication and considerable investment on manpower and renounces to improve I DRILLING EDM functionality and enable its high performance.

1.1. System framework

CNC Drilling EDM Controller framework is as following figure1-1-1.

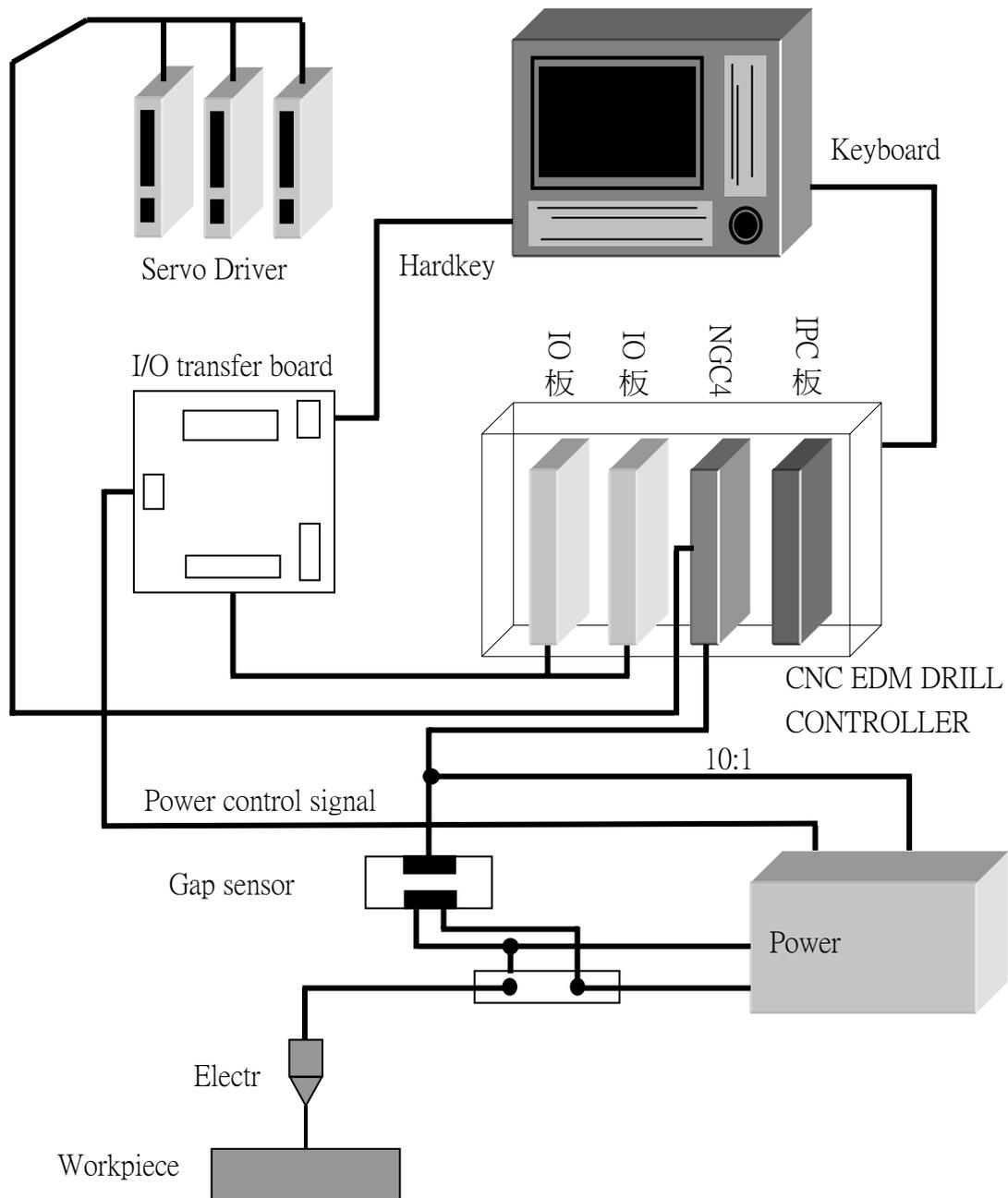


Figure1-1-1 controller framework



1.2. Hardware

The system hardware is divided into (1) controller, (2) LDD display panel, and (3) I/O panel. Please see Figure 1-2-1 as shown below.

(1) Controller

- One control box set
(including: Rack, 6-slot ISA-bus back panel, 150W power supply, one 3.5" FDD)
- IPC(MIC-2352)卡(On-board Pentium MMX 266 CPU、An IDE connector on panel is provided to easily connect with an external device、The on-board Flash Disk expansion slot offers a storage alternative in harsh industrial scenario 含 VGA)- one piece
- RAM/ROM (32MB) - one piece
- CF Disk (32MB) - one piece
- Three (four) spindles motion controller - one piece
- 32I/32O I/O card - two pieces

(2)12.1" TFT LCD display panel

- Size 482mm x 248.8mm x 63mm
- 12.1" TFT LCD
- Screen switch, brightness control
- Control panel (Function Key, Numeric Key, English Letters of the Alphabet Key, etc.)
- External connection to a keyboard is allowed.

(3) I/O panel

- POWER ON/OFF
- EMERGENCY STOP
- Buzzer
- CYCLE START
- FEED HOLD
- RESET
- JOG X、Y、Z、W
- NO GUARD

- JOG/INC
- CW, CCW (SPINDLE)
- PUMP ON/OFF
- COOLANT ON/OFF
- MTC (Manual Tool Change)
- FTC (Finish Tool Change)
- BASE POINT (Electrode switch edge search base point)
- ATC Unclamp
- GUIDE Unclamp
- SPEED(JOG:1%、10%、100%)

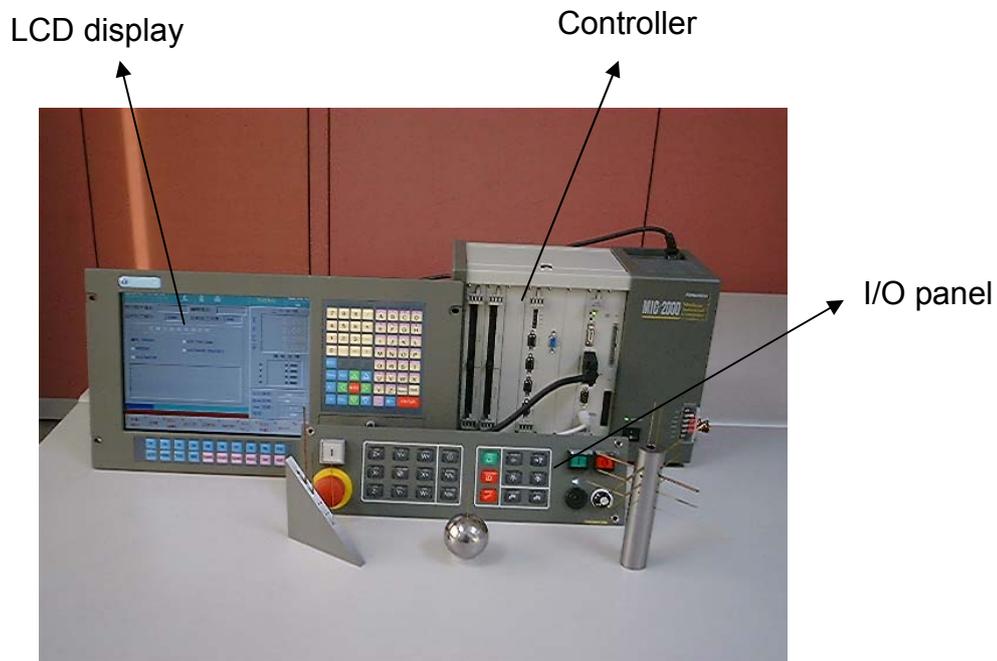


Figure 1-2-1 System Hardware



1.3. Software

- Chinese / English display
- Metric and GB unit display and program setting
- Machine Coord., Work Coord. and Following Error display
- Three manual modes (continuous, inch movement, without protection)
- Home search and position
- Automatic edge, Center of two PT., Locate Center, Locate Corner, work axis Rotation, Mirror.
- GM Code Program and 500 blocks of workpiece program
- Help on Line.
- DXF file and WireCut Program Transfer
- Dialog Edit.
- Provides the dialog program edit function of Rect Patt. Circle Patt., etc.
- Provides the function of calling Mach. SUB PROG with Max. 8 Layers
- Has multi-set mach. criterion database, each set of criterion database can store 100 mach. criteria of operators.
- Can store 100 mach. depths of 100 operators
- Single and Multi Hole Machining
- Machining path display
- Machining Simulation
- Software limite
- Program Set(Start block and end block setting)
- When the Mach. attains the depth, the electrode will automatically detect and increase to the fixed height of the mach. surface.
- Single/ Mul. H. Mach., when the electrode is run out during the mach., shift to the function of automatic compensation.
- Records the Mach. Time
- Variable Deepness Machining
- Backlash and Pitch Error Compensation
- Automatic Electrode Change(need Instruments)
- MPG remote control
- RS-232 file transfer
- Alarm display



- I/O、C、S、A、Timer、Counter and Ladder display on line ◦



1.4.Axis Control

Item	Axis number
Basic Axis	3 axis
Expanded Axis	1 axis (Max 4 axis)
Basic Co-move Axis	3 axis
Expanded Basic Co-move Axis	Max 4 axis
MPG control axis	4 axis
Spindle	1 axis

1.5. Unit

Item	Min. input unit	Min. Command	Max. Stroke
Metric	0.001 mm	0.001 mm	9999.999 mm
Inch	0.0001 inch	0.0001 inch	999.9999 inch
Angle	0.001 deg	0.001 deg	359.999 deg



1.6. G Code List

Group	Function	G Code
Position	Fast Positioning	G00
	No Guard Fast Positioning	G05
	Machine Coordinate Position	G53
	Edge Searching	G80
	Mechanic Origin Searching	G81
	Locate Groove Center	G82
	Locate Inner Hole	G83
	Locate Outer Center	G84
	Locate Circular Cylinder Center	G85
Coordinate	Absolute/Relative Coordinate	G90/G91
	Set Coordinate	G92
Other	Hold for Seconds	G04
	Check Point Set	G06
	Block Skip Cancel	G30
	Block Skip Active	G31
	Condition Table Change	G38

Chapter 2 Procedure of Turn on and Turn Off

2.1. Turn ON Procedure

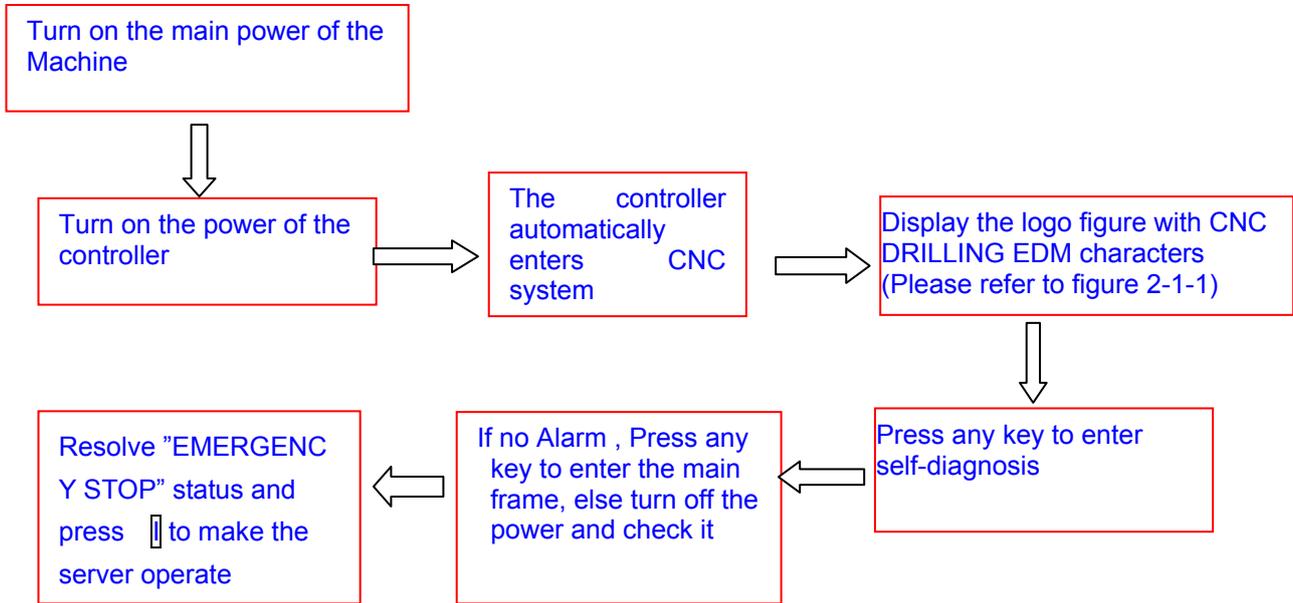


Figure 2-1-1 Turn ON Procedure Flow diagram

In the bottom right corner of the initial screen (Note), it shows the software version and the date. Pressing any key enter the main screen. If no, it enter main screen automatic after seconds. Operator can press **Shift+F10** to show the initial logo screen and see the version number.

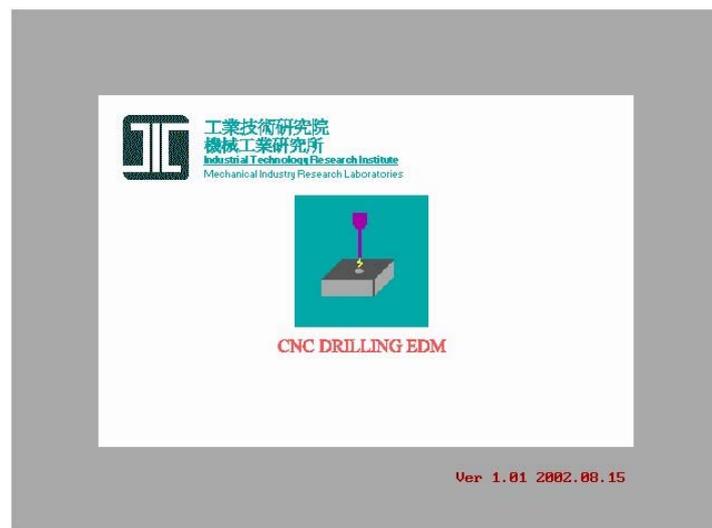


Figure 2-1-2 Turn ON Initial Screen

When the system setting starts to ATC Enable (please see Chapter 4 9.2 Custom Param.) it shows the message “Please Home Search” after booting up as figure 2-1-3. It hints operator to home search. When the home search is not completed, it is not able to enter Single H. Mach and .Mul. H. Mach. mode (please see Chapter 4 8. Mul. H. Mach.). At this time, it just needs to complete the home search or just close the ATC (Auto Tool Change) function. In the machine with AGC (Auto Guide Change), it needs to home search at first. Then it has the machining ability.

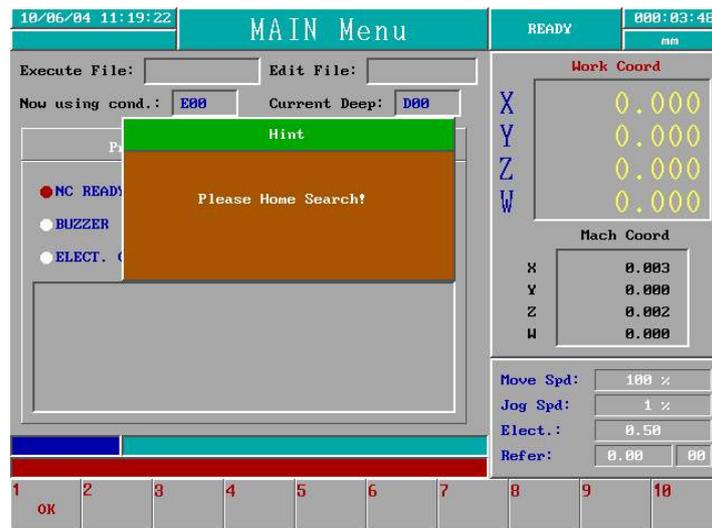


Figure 2-1-3 Please Search HOME Prompt Screen

Note : If that axis has home searched, the axis name of the machine coordinate is marked by “*”. If not, it has no mark.

Note : The initial logo screen of booting up is difference between the difference companies.

2.2. Turn OFF Procedure

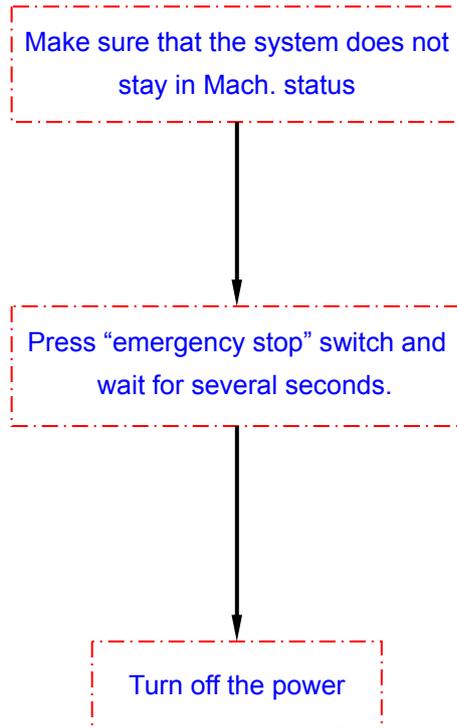


Figure 2-2-1 Turn OFF Procedure Flow Diagram

Note : Before turning off the controller, operator should press the E-Stop key at first to make controller record the status. Then turn off the power. This procedure can avoid the file data would be lost.

Chapter 3 Operation Instructions

3.1. Screen Deployment

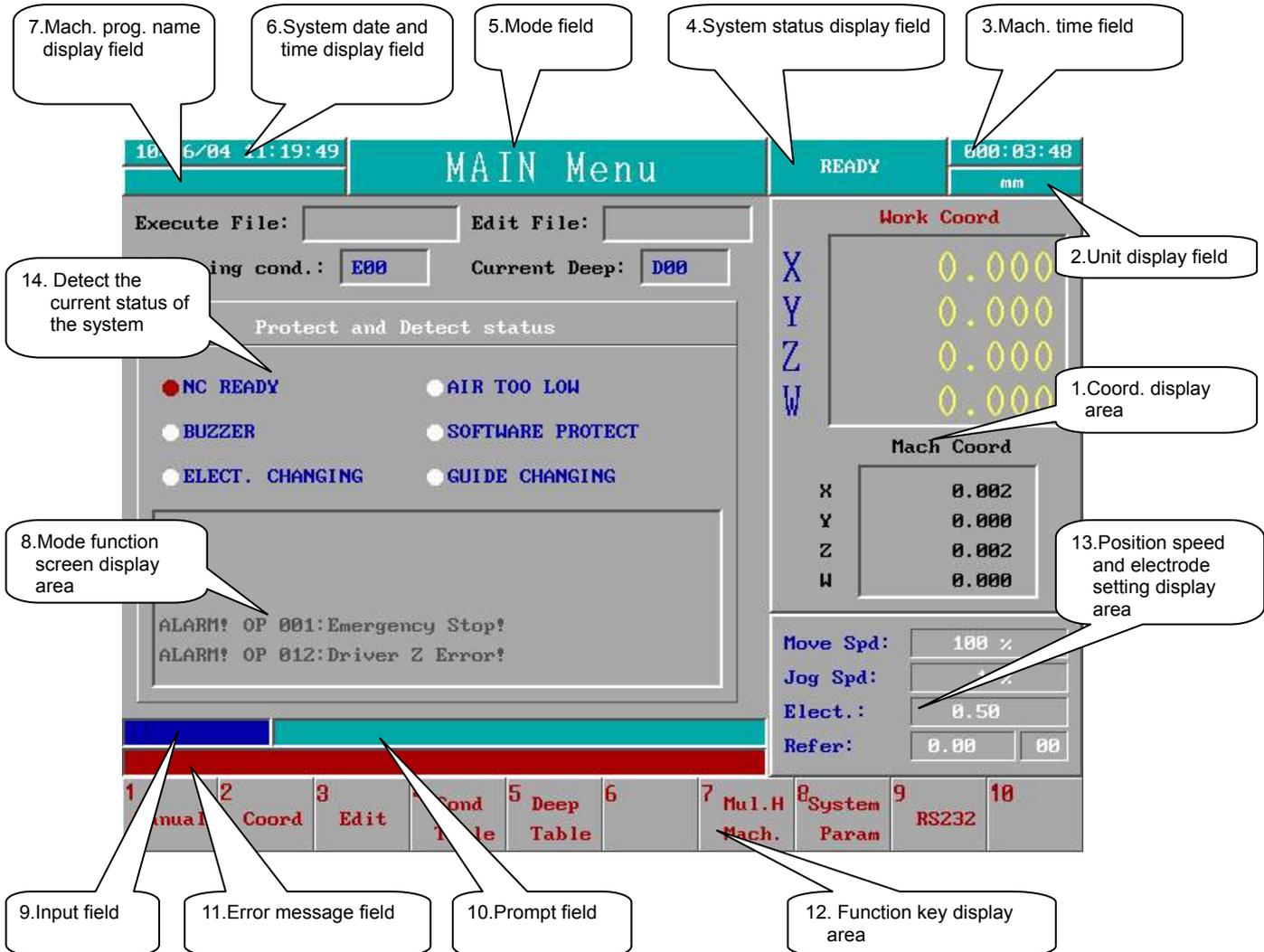


Figure 3-1-1 Screen Deployment

Reading instructions:

To facilitate readers in understanding more clearly about the description of the operating instructions, take note of the following ways of marking.

The keys of the operating panel and the I/O panel: marked in a square external frame, such as **MODE**.

System function key: marked in square external frame and with a gray background, such as **F2 Manual**:

The characters on the window: marked in a gray background, such as **System**



Ready

As shown in Figure 3-1 above, the overall screen is divided into 13 areas. The description of each area is as follows:

(1) Coord Display area

Displays the current work coord and machine coord of the electrode. The unit, in accordance with the different system settings, is divided into metric (mm) and GB (inch). When the unit is displayed in mm, the display min. decimal figure is 0.001mm; when the display unit is in inch, the display min. decimal figure is 0.0001inch.

◎Work coord:

The work coord is the coordinate referred to by the workpiece prog. The coord value can be set in manual mode, or press **MODE** on the control panel, and then the coord set window will be displayed. Input the setting.

◎Machine coord:

The coordinate is a fixed reference one. The Home of the coord system is one certain fixed position of each spindle of the machinery and the operator is not allowed to set it. The operator can be in **F1 Manual** to press **F2 Home Search** and then select to move that particular spindle to machine coord home. Owing to the fact that the machine coord stands for the electrode on the clip relative to the position of one fixed point in the space (machine coord home), the operator cannot alter the value in manual mode. When the electrode is moving in the space, the machine coord value will change with it.

(2) Unit display field:

It displays the currently used length unit, **mm** or **inch**, of the system.

(3) Mach. time field:

It displays the total mach. time of the current work accumulatively, which is displayed in the hr: mm: sec style. The hr stands for hour, mm stands for minute and sec stands for second. The max. can display " 9999 : 59 : 59 ". Regarding the clear and cal. of mach. time, please refer to Chapter 4 Manual Mode.

(4) System status display field:

It displays the current system status, such as the messages of **READY**, **Not**



Ready, Machining, Pause, Reset, etc.

(5) Mode field:

It displays the currently selected function mode of the system, such as the messages of Manual OP., Edit, Condition Tab., Deep Table, etc.

(6) System date and time display field:

It displays the current date and time. The date display method is mm/dd /yy and the time display method is hour: minute: second, such as 07/01/02 12:30:20

(7) Mach. prog. Name display field:

Displays the loaded mach. programs of the current system

(8) Mode function screen display field:

Each mode function screen shift display area

(9) Input field:

All the input data of the system by keyboard: English words and numeral are displayed in the field. After **ENTER** key is pressed, the data is updated to the relative field. At this time, the data input field is cleared. Thus the data input is completed. The relative field for data input is based on the pos. of the cursor.

(10) Prompt field:

Displays the data input range and the relevant prompts for the field where the cursor is placed.

(11) Error message field:

Displays any system error should that occur. When the message is displayed, it is necessary to eliminate this instead of continuing operation.

(12) Function key display field:

It displays the suitable function key for the current window. When the function key is shown in red, this means that the function key operates normally; however, if it is shown in white, the function key does not operate.

(13) Position speed and electrode setting display field:

It displays the movement speed of the current setting, Refer NO., Jog speed and electrode diameter. After the system is started, the pre-set movement speed is 100%, Refer NO. is 00, Jog speed is 1% and electrode diameter is 1

unit.

3.2. Control Panel Key Description:

The deployment figure of the control panel is as shown in Figure 3-2.

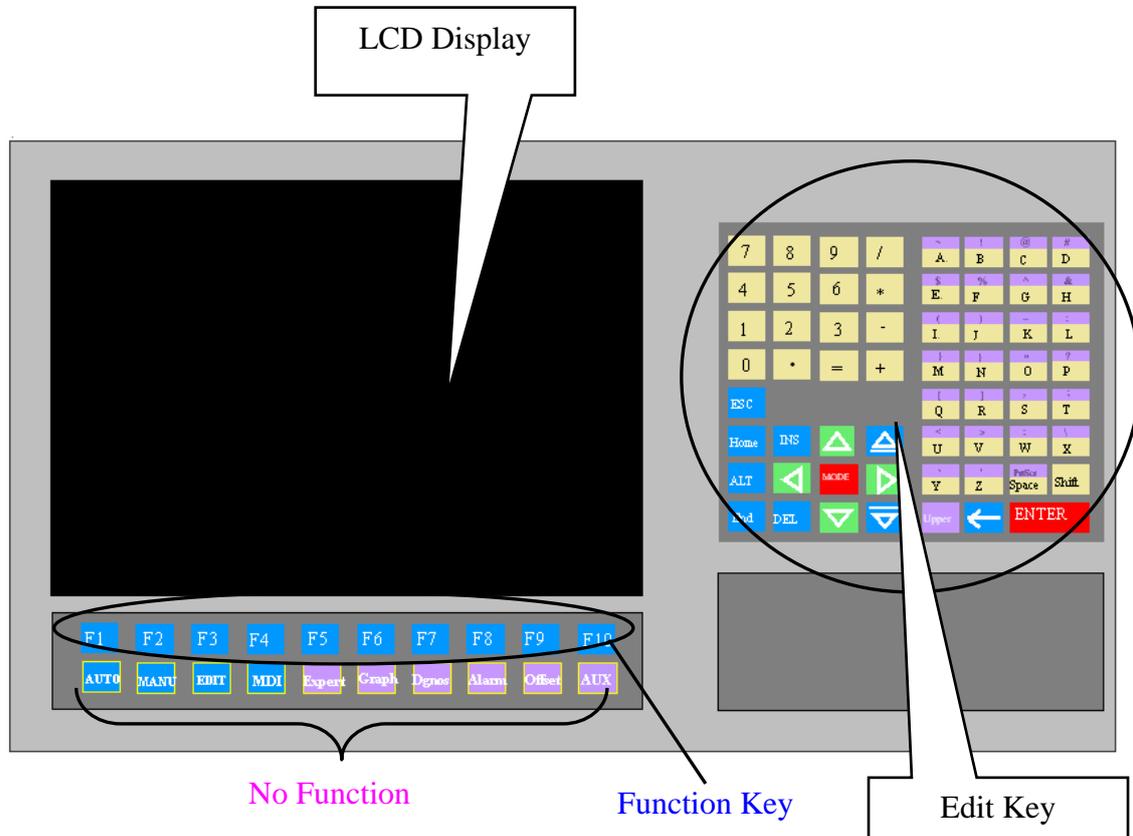


Figure 3-2-1 LCD Panel Deployment Figure

ESC: stands for giving up a motion. For instance, when the data is input in operational mode, the operator can decide to give up the input data and restore to the status before the data is input.

INS: insert character into the current pos. of the cursor.

DEL : in the edit status of the operational mode, the prog. block at the pos. where the cursor is place can be deleted.

CTRL: currently not used

ALT: **ALT**+**F10** is to exit CNC Drilling EDM system and to enter DOS.

SHIFT: **SHIFT**+**F10** displays the initial LOGO screen when the system is turned on.

MODE (or **TAB**): produce the window for coord value input, and is used in coord set.



HOME : moves to the position of the first character of the current string.

SPACE : space key

▲, ▼ : can select mach. param and move the cursor within the work area, the function is equal to the up and down key on the keyboard.

▶, ◀ : move the cursor within the work area, the function is equal to the left and right key on the keyboard.

▲, ▼ : can page up(or page down) in the file.

BKSP or  : delete the last character in the input field prior to the cursor, the function is equal to Backspace on the keyboard.

ENTER: accept the data of the input field.

F1 ~ **F9**: the function key that each mode sets itself.

F10: return to last window or function key.

Eng. letters, numbers, and special symbols: the function is similar to that of the keyboard.

UPPER : Press Upper key and Eng. letter key, and display the symbol that the Eng. letter stands for within the purple area.

3.3. I/O Operation Panel

The I/O operation panel of the system is consisted of the followings: power control, manual function control, automatic function control and I/O point function control as following figure 3-3-1.

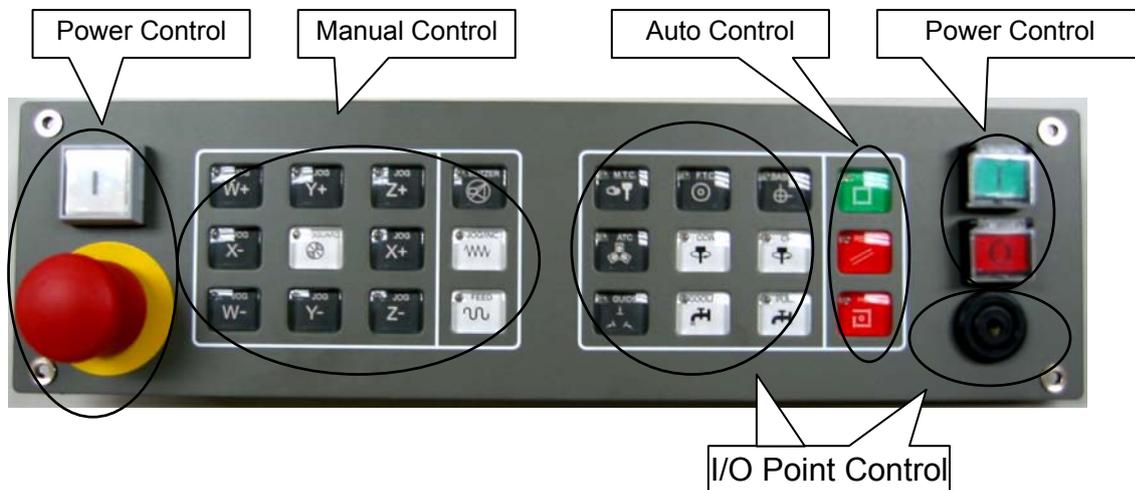


Figure 3-3-1 I/O operation panel

3.3.1. Power Control

(1) Controller power switches : Pushing down the green button on the left will turn on the controller power; pushing down the red button on the right will turn it off. However, the heavy-duty power must be off (the emergency stop button down) before you turn on the controller power, otherwise the controller power cannot be turned on.



Figure 3-3-2 Controller power switches

(2) Heavy-duty power switches : Pressing down the heavy-duty power button will turn on the heavy-duty power, and then the machine is operable. However, when the emergency stop button is down or the controller power is not turned on, the heavy-duty power cannot be turned on. To make sure the accuracy of coordinate position, you should re-perform home point recover when the heavy-duty power is turned on.



Figure 3-3-3 Heavy-duty power switch

(3)Emergency stop button : The heavy-duty power will be turned off after pressing down the button, and the machine will not be operable. The use of this button is when: 1.emergency event occurs and the machine operation must be stopped immediately or; 2.there is the need to turn off the overall system power. To release the emergency stop button, just pull the button up.

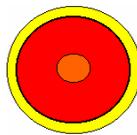


Figure 3-3-4 Emergency stop button

3.3.2. Manual Control

- (1)JOG : this button will be activated when in non-automatic mode or the FEEDHOLD button is pushed down in automatic machining mode. User can press X+, X-, Y+, Y-, Z+, Z-, W+ and W- buttons to move different axes.
- (2)NOGUARD : this button is to protect the electrodes when contacting with machining article and the short circuit occurs. It will set the feed rates of all axes to zero and disable any continuation of axes movements. Therefore, to release the system protection status, you need to push down this button and MANU button at the same time, and then move those axes to get the electrodes away from the article, and solve the short circuit condition.
- (3)F.T.C. : The electrode runs out during machining, used when the electrode is replaced
- (4)SPEED : the speed adjustment button is to adjust the percentage of speed, can be set to 1%, 10% and 100%, of manual feed. If it is in INC JOG mode, the manual feed increments are 1 μ m, 10 μ m and 100 μ m. Every time you push down the button, the speed will circle from the lowest to highest and then lowest to highest again. The speed status is displayed in

the manual buttons display area at the upper right.

- (5)INC JOG : this button allows user to change the manual jog mode from serial to non-serial. The difference is that the axes will continue moving as long as the manual button is pushed down under serial mode; on the contrary, when it's under non-serial mode, the axes move one incremental value every time when the manual button is pushed down, regardless how long it lasts, and the incremental value is based on the speed value.

3.3.3. Auto Control

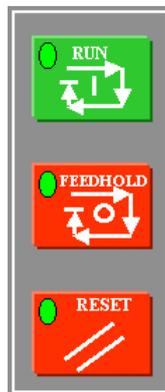


Figure 3-3-5 auto control button

- (1)RUN : Starts to run machining program
- (2)FEEDHOLD : Pause machining program. When pressing this key in auto-running, system will be paused. If operator wants to continue the machining, please press the RUN Key.
- (3)RESET : Reset machining program or the alarm. When operator presses this key, it reset the system. There are two situation operator presses this key. The first, operator reset the system in machining status. The second, operator want to clear the alarm message in alarm happened status.

3.3.4. I/O Point Control

- (1)MTC : Pressing this button, and it execute changing electrode process.
- (2)BASE Pt : In changing electrode process, if the check point is not active, it need to JOG the electrode to check point and press **BASE Pt** to set this



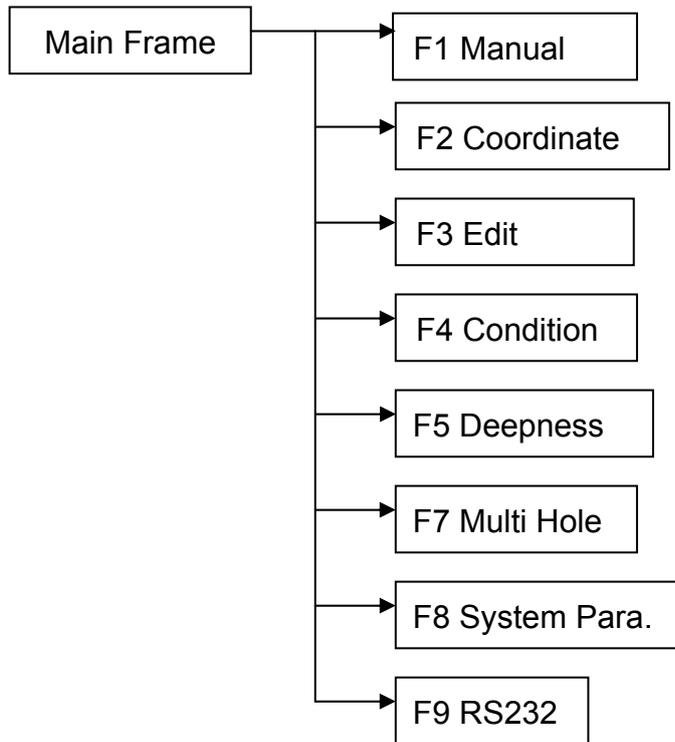
position as the check point. When doing the BreakPT and InitPT. Function, it need to press this button to set the initial status.

- (3) CW : The main spindle rotates clockwise. After pressing this button, the spindle rotates clockwise. If pressing again, the spindle stops.
- (4) CCW : The main spindle rotates counterclockwise. After pressing this button, the spindle rotates counterclockwise. If pressing again, the spindle stops.
- (5) PUMP : The water pump is switched on. After pressing this button, the water is pumped into the electrode. If pressing again, the water pump is stopped.
- (6) COOLANT : The coolant pump is switched on. After pressing this button, the coolant is pumped. If pressing again, the coolant pump is stopped.
- (7) BUZZER Button : Buzzer Active Button. Default status is ON and the LED is ON. The Buzzer can work. If pressing this button, the LED is off and the buzzer can not work.
- (8) ATC : Spindle chuck unclamp. When pressing this button, the spindle chuck is unclamped and operator can pull the electrode clamber out. Releasing it, the spindle chuck is clamped and the spindle chuck can clamp the electrode clamber.
- (9) GUIDE : Guide chuck unclamp. When pressing this button, the guide chuck is unclamped and operator can pull the guide out. Releasing it, the guide chuck is clamped and the guide chuck can clamp the guide.
- (10) BUZZER : When alarm is happening, short and finishing the machining, system will switch on the buzzer for seconds.

3.4. Function key illustration

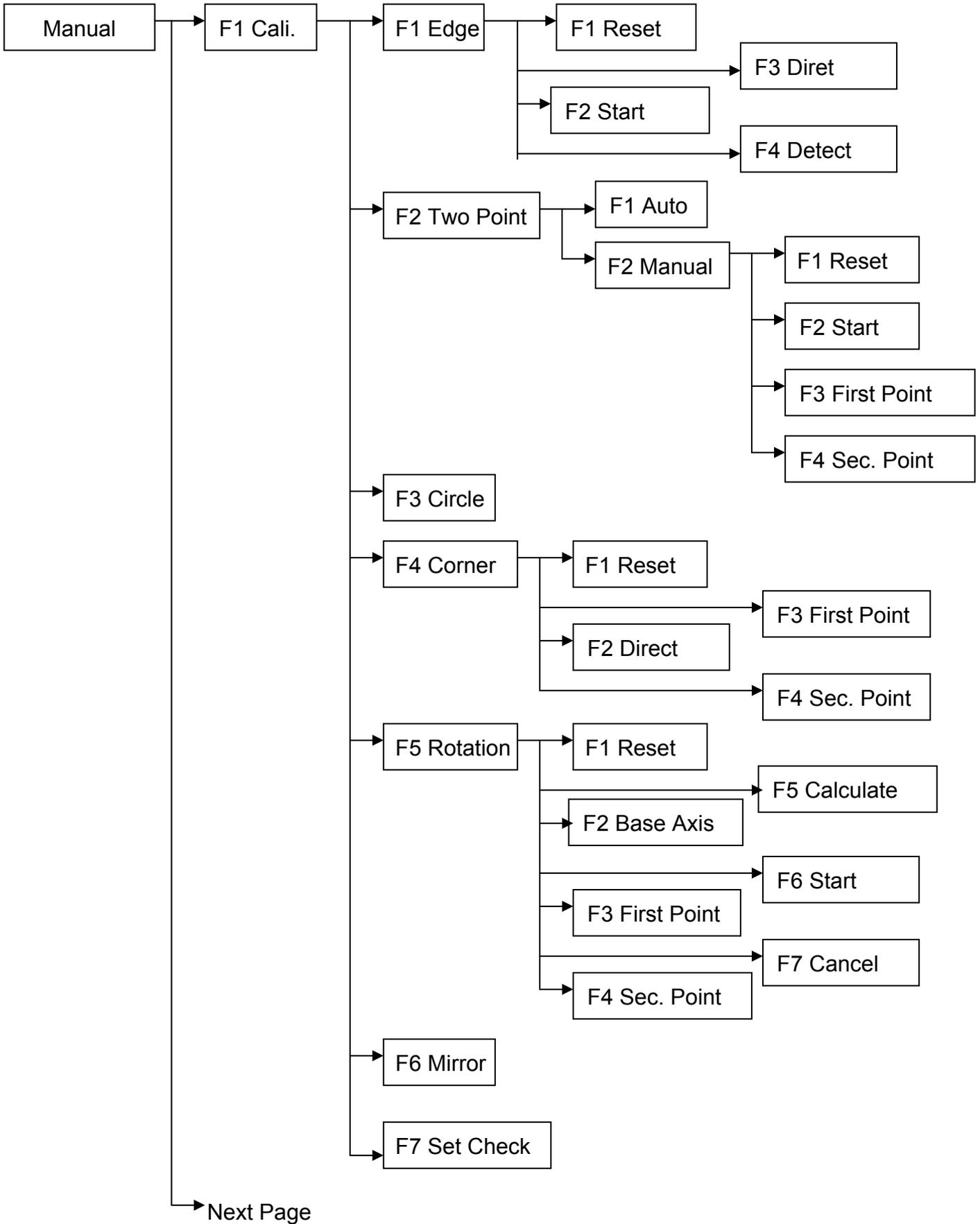
Each window and mode of the system has its relevant function key. The figure below shows the connection of each model and each function key.

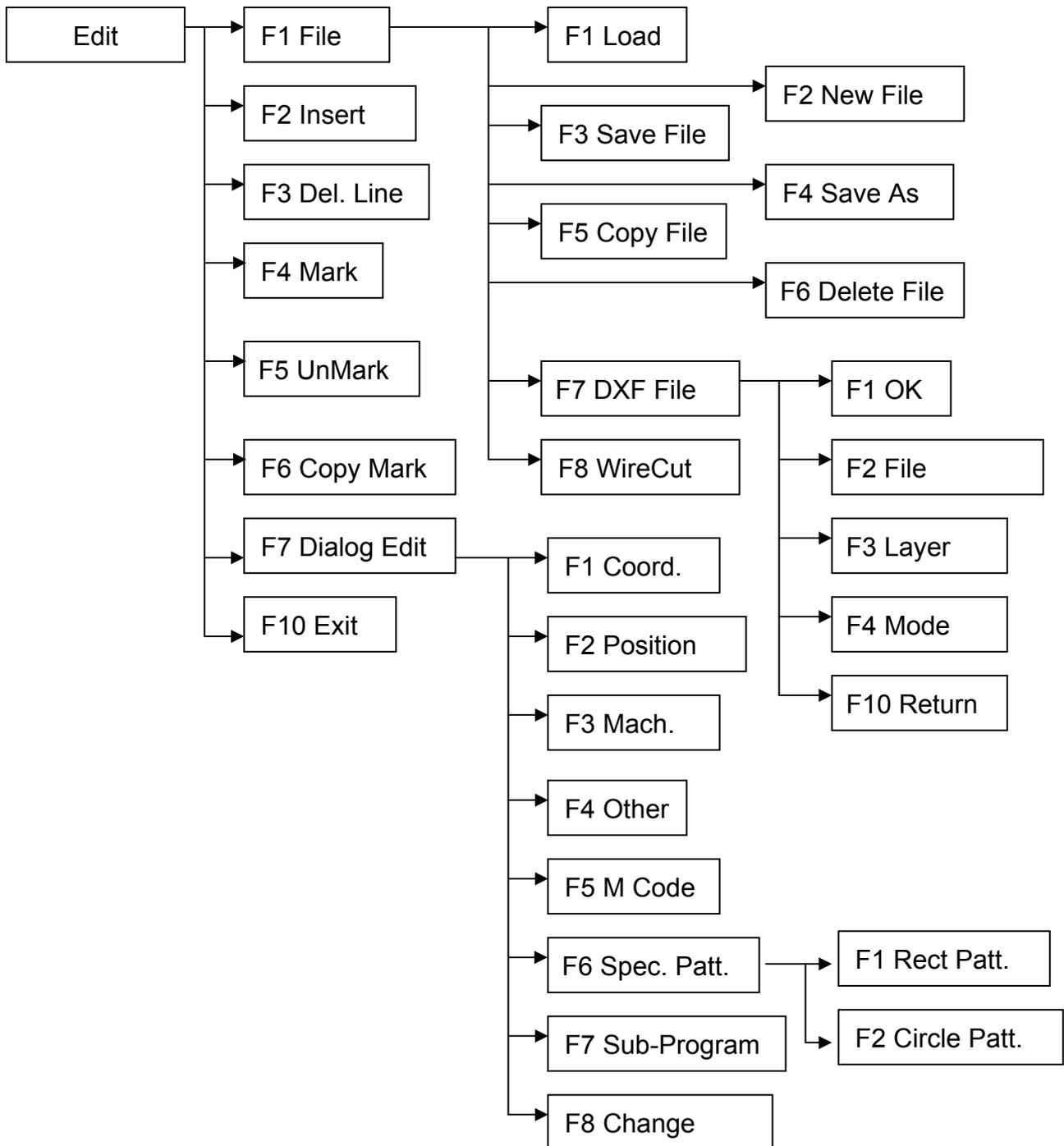
3.4.1. Main Frame





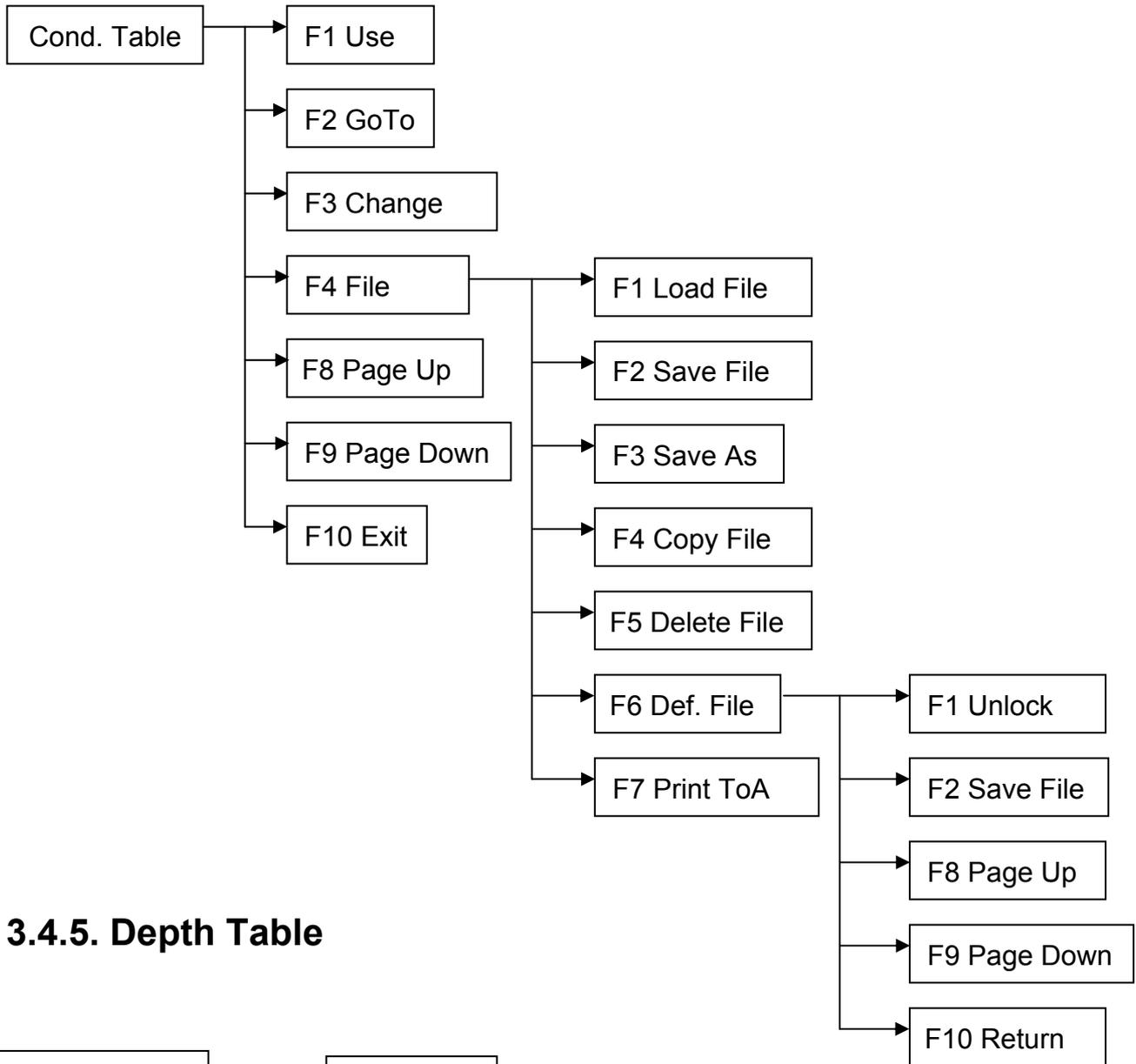
3.4.2. Manual



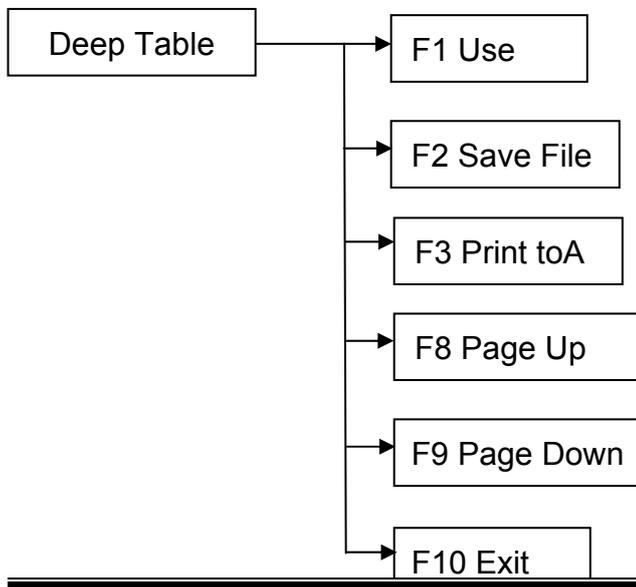
3.4.3. Edit



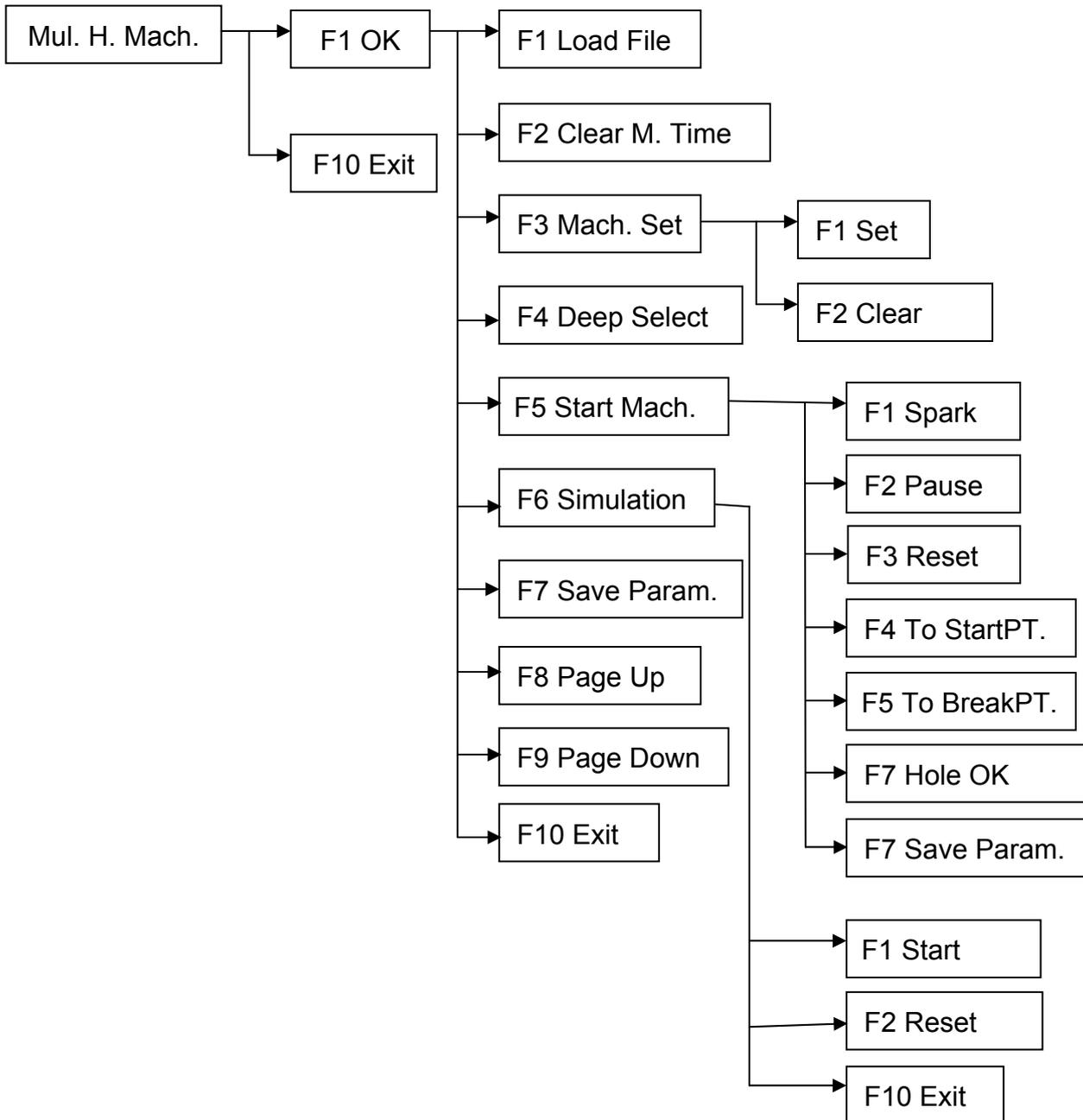
3.4.4. Condition Table



3.4.5. Depth Table

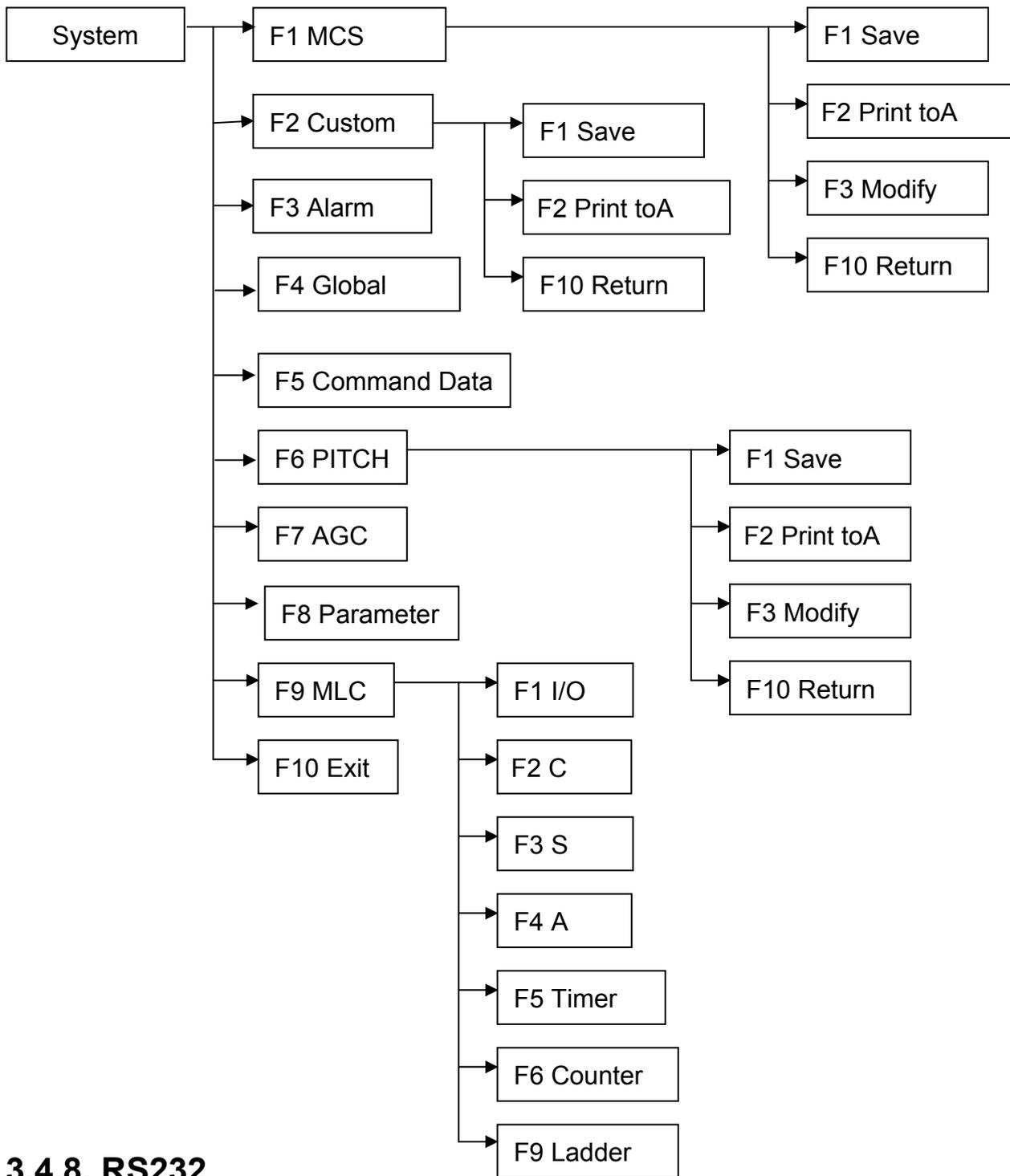


3.4.6. Multi Hole Machine

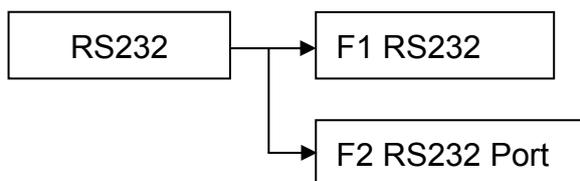




3.4.7. System Parameter



3.4.8. RS232



Chapter 4 Function Mode Description

4.1. Brief introduction

This chapter introduces the usage, purpose, and the operation method. The system includes the function modes of Manual OP., Edit, Condition Table, Deep Table, Multi Machine, System Parameter and RS232. The respective introduction to each function mode is as follows.

4.2. MANUAL

Manual OP. is mainly suitable for each setting and test of the system and workpiece.

The operating screen is as shown in Figure 4-2-1 below. The relevant functions include workpiece calibration, system home search and position, system electrode size and move speed setting and single mach. mode. Thus, Manual OP. includes **F1 Cali.**, **F2 Home Search**, **F3 Pos.**, **F4 Electrode Size**, **F5 Move Speed** and **F6 Sinle H. Mach.** These function keys are used to start the relevant sub-mode and the setting window. The following article introduces more of these function keys.

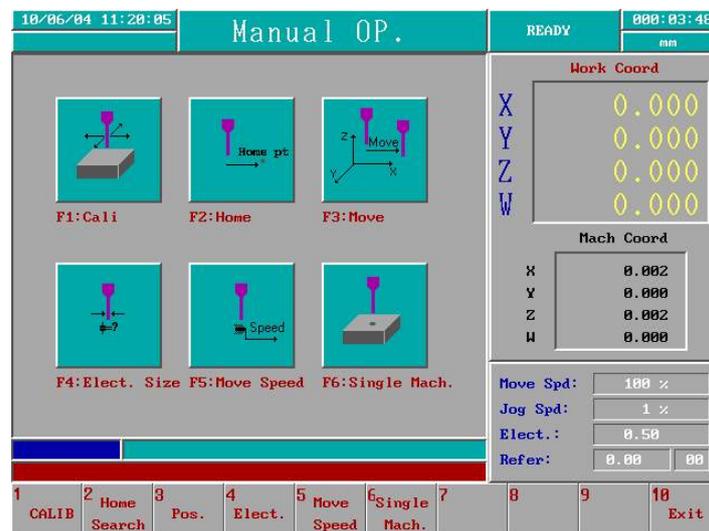


Figure 4-2-1 Manual OP. Screen

4.2.1. CALIBRATION

CALIB is mainly suitable for workpiece base pt. search, such as workpiece edge or center. This can be divided into **EDGE Search**, **Two Point**, **Center of Circle**, **Corner**, **Rotation**, **Mirror** and **Set Check**. Seven types of sub modes are respectively started in **F1 Edge Search**, **F2 TWO POINT**, **F3 CIRCLE**, **F4 Corner**, **F5 Rotation**, **F6 Mirror** and **F7 Set Check**. About the window screen of CALIB, please see Figure 4-2-2 below. The following introduces these three types of sub modes first.

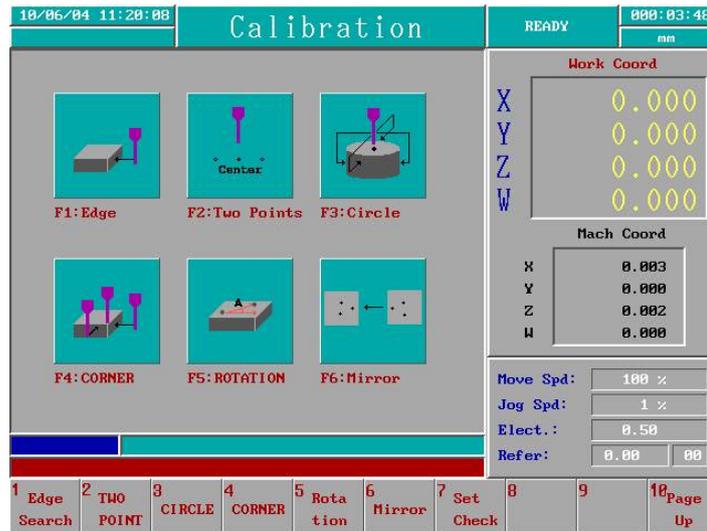


Figure 4-2-2 CALIB Mode Screen

4.2.1.1. EDGE SEARCH

Edge search is mainly suitable for workpiece edge search. The function of **Coord. Set** can be used to set the coord value of the workpiece edge into the work coord. Then it is very convenient for an operator to set the mach. reference coord. The operating screen is as shown in Figure 4-2-3.

Method of operation:

1. Set the edge search route: Press **F3 Direct**. At this time, the cursor will move to **X+** or select the desired edge search direction by using up and down key, such as **X+**, **X-**, **Y+**, **Y-**, **Z+**, **Z-**, **W+** and **W-**.
2. Set detect num.: Press **F4 Detect Num.** and the cursor moves to the field for surface detect num. Follow the cue of the prompt field to input edge search num.

3. Start edge search: Press **F2 Start** to start edge search. At this time, the platform moves slowly in the selected edge search direction. The system status field of the monitor display **Machining**; meanwhile, the prompt field displays **Detecting surface, waiting...** After the edge search is completed, the system status field displays **READY**. Meanwhile, the prompt field displays **Edge Searching Finish!**. At this time, the detect mean field relative to edge search direction displays the average coord value of the workpiece edge and displays each detected value at the bottom of the screen.
4. Set coord set: After the edge search, press **F5 Coord. Set**. At this time, the window for input coord set is displayed. Input the work coord value after edge search into the individual field. Press **ENTER** or **F1 OK**, and the system will automatically calibrate the electrode radius and update the work coord value (it is necessary to set electrode size first).

When the motion is started, press **F1 Reset** any time to stop the running of the motion. At this time, the system stops and the system status field displays **READY**. At the same time, the prompt field displays **Break surface detecting**.

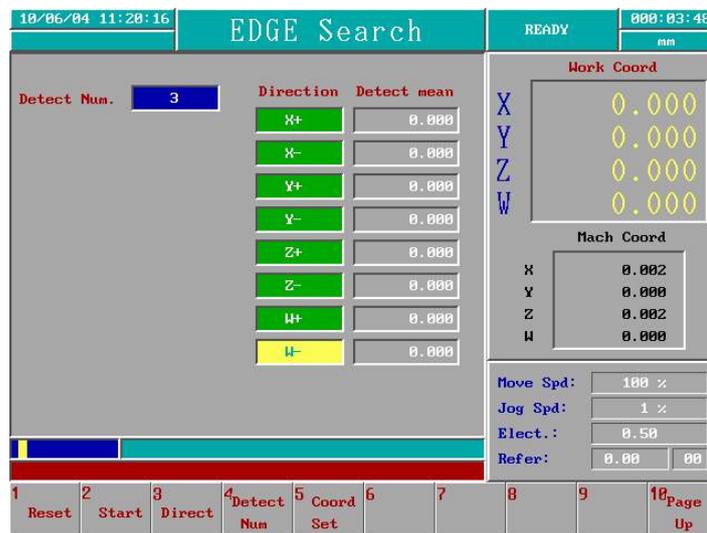


Figure 4-2-3 Edge Search Screen

4.2.1.2. TWO POINT

Two point is mainly suitable for searching the central position between any two edge points of the workpiece. This can facilitate the operator in

setting the mach. reference coord. The operating screen is as shown in Figure 4-2-4 below.

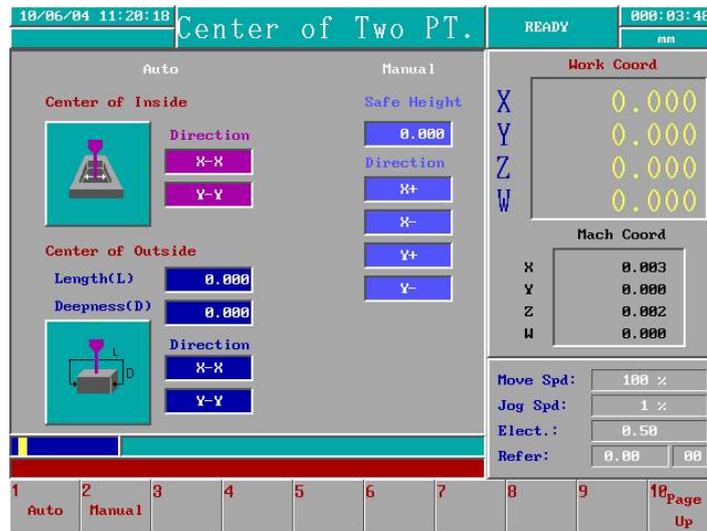


Figure 4-2-4 Two Point Screen

The operation of two point can be divided into two types: auto and manual. Auto mode can complete the two point search by way of simple settings while the application of the manual mode is more flexible. Select to press **F1 Auto** or **F2 Manual** to select the operation mode.

(1)Auto

Operate two point in auto mode. The operating screen is as shown in Figure 4-2-5 below.

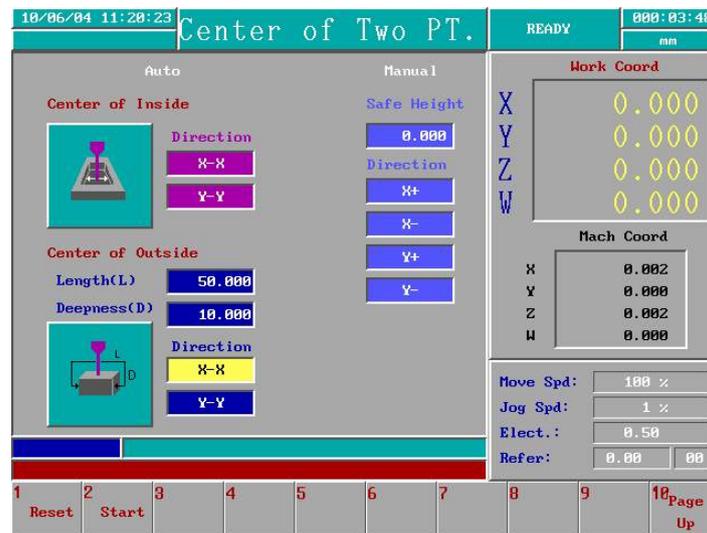


Figure 4-2-5 Two Point Auto Mode Screen

Operating instructions:

1. Set detect method and direction: the detect method can be divided into Inside DIR and Outside DIR. Each detect direction can further be divided into two types: X-X direction and Y-Y direction. Therefore, there are a total of four types for selection. After entering auto mode, the cursor is placed in the X-X field of inside detect. Use the up and down keys to select and start the desired type.
2. Set the format of outside detect: during the outside detect, it is necessary to set S = "Detect Leng." and Detect (D) as shown in the figure. You can understand what S="Detect Leng." and Detect (D) mean. Use the up and down keys to select S = "Detect Leng." or Detect (D) and then set by numeric key input. After completing the setting, use the up and down keys to move the cursor and select the desired direction. During the inside detect, it is not necessary to set.
3. Start two point: Press **F2 Start** and start the motion of two point. At this time, the system status of the screen displays **Machining**. Meanwhile, the prompt field displays **Detecting!**. After completing the motion, the electrode automatically moves to the central position of the edges of the

two points and the system status field displays **READY**. Meanwhile, the prompt field displays **Finish Enter of Two Pts!**.

When the motion is run, press **F1 Reset** to stop the motion and the system status field displays **READY**. Meanwhile, the prompt field displays **Break Detecting!**.

Caution: It is necessary to pay attention to the setting value of S = "Detect Leng." and Detect (D) to avoid the workpiece bending the electrode by collision.

(2) Manual

Operate two point in manual mode. The operating screen is as shown in Figure 4-6 below.

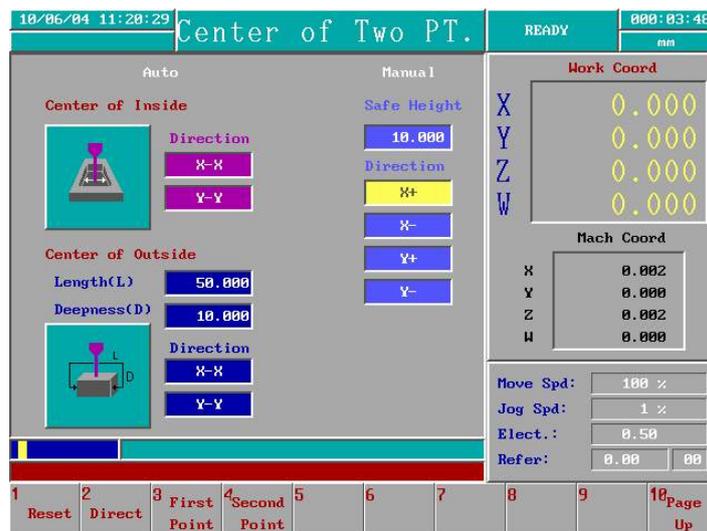


Figure 4-2-6 Two Point Manual Mode Screen

Operating instructions:

1. Set the edge search direction of the first point: divided into four edge search directions: **X+**, **X-**, **Y+**, **Y-**. Move the electrode to the edge search position of the first point near the workpiece with no contact. Then press **F2 Direct**, and the cursor moves to X-field immediately. Use the up and down keys to select the desired direction.
2. Start the edge search of the first point: Press **F3 First Point**, and the platform moves in the edge search direction set in 1. At this time, the



system status field of the screen displays **Machining**. Meanwhile, the prompt field displays **Detecting!** until it touches the edge of the workpiece. Then the first point search is completed. At this time, the function key **F3 First Point** displays in white, which means no function, and the coord value of the first point has been recorded in the system.

3. Set W safety height: When completing the edge search of the second point, electrode rise the safety height first and then moves to the center of the two point. Due to this, in order to avoid colliding with the setting of the workpiece, the W safety height cannot be set to 0.
4. Set the edge search of the second point: the same as in description of 1. Move to the edge search of the second point near the workpiece without contact. Press **F2 Direct** to set the edge search direction.
5. Start the edge search of the second point: Press **F4 Second Point**, and the platform moves to the edge search direction set in 3. until it touches the edge of the workpiece. Then the search of the second point is completed. At this time, W increases the distance of the safety height first and then moves to the center of the two points. After the motion is completed, the status field of the system displays **READY**. At the same time, the prompt field displays **Finish Center of Two Pts!**.

When the motion is started, press **F1 Reset** any time to stop the motion. At this time, the system stops and the status field of the system displays **READY**. Meanwhile, the prompt field displays **Break Detecting!**. When the edge search of the second point is started, reset the motion of the edge search. The edge search value of the first point is still kept. At this time, the function key **F3 First Point** still displays in white. Unless it is restarted again, the edge search value of the first point can be cleared, too. At this time, the function key **F3 First Point** changes to normal red words and the function restored.

4.2.1.3. CENTER OF CIRCLE

Center of circle is mainly suitable for searching the position of the circle hole of the workpiece or the circle center of the cylinder. This can facilitate an operator in setting mach. reference coord.

The operating screen of circle center mode is as shown in Figure 4-2-7. It can be divided into two types: center inside the circle hole and center outside the cylinder.

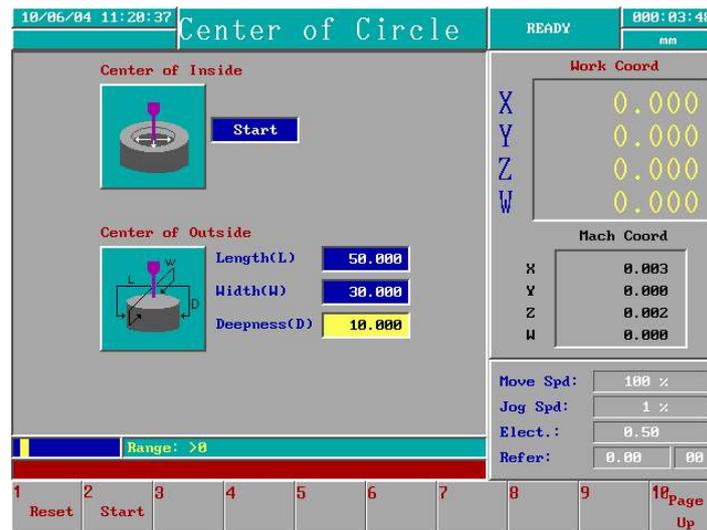


Figure 4-2-7 Center of Circle Screen

Operating instructions:

1. Set circle center type: Once it enters the circle center mode, the cursor is placed in the executive field inside the center. Use the up and down keys to select the desired center of inside or that of outside.
2. Set the format of center of outside: When the circle center of outside is detected, it is necessary to detect the length, width and depth as shown in the figure. The meanings of the length, width and depth can be understood. Use the up and down keys to move the selection and then input the value by numeric keys respectively. When the circle center of inside is detected, it is not necessary to set.
3. Start center of circle detect: Press **F2 Start**, and start the motion of center of circle detect. At this time, the status field of the screen display **Machining**. Meanwhile, the prompt field displays **Detecting!**. After the motion is completed, the electrode move to the position of center of circle and the status field of the system display **READY**. Meanwhile, the prompt field display **Finding Center of Circle!**

When the motion is started, press **F1 Reset** any time to stop the motion. At this time, the system stops and the status field of the system displays **READY**. Meanwhile, the prompt field displays **Break Detecting!**.

Caution: It is necessary to detect the setting values of the length, width and depth to avoid the workpiece bending the electrode by collision.

4.2.1.4. CORNER

Corner is mainly suitable for searching the corner position of the workpiece. This can facilitate an operator in setting mach. reference coord.

There is a limited condition in corner mode. Before positioning the corner of the workpiece, it need to be calibrated to parrel to X or Y axis. The operating screen of circle center mode is as shown in Figure 4-2-8.

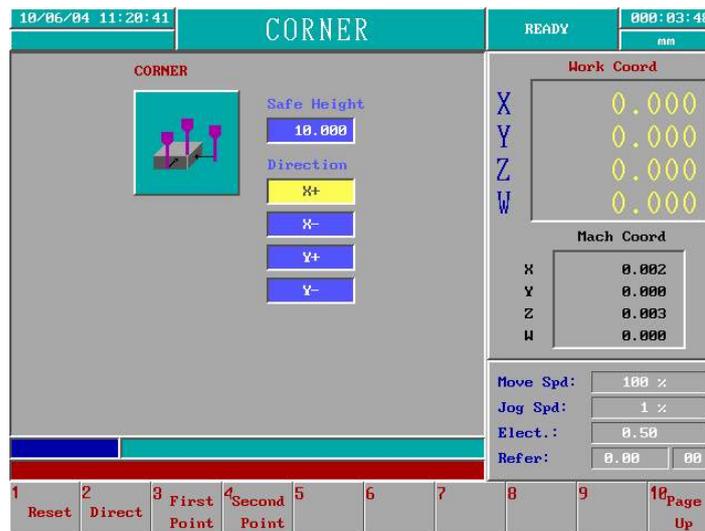


Figure 4-2-8 Corner Screen

The operating instrument in Corner mode is same as in Center of Two PT. It need to execute the first point edging, then the second point edging. Finally, the electrode moves to the corner. Therefore, there are the first point and the second point edging in operating. After edging the first point, operators cannot key in the electrode size. And the first point is edging X + or X-, the second point edging is limited to Y+ or Y-.



Operating instructions:

1. Set the edge search direction of the first point: divided into four edge search directions: X+, X-, Y+, Y-. Move the electrode to the edge search position of the first point near the workpiece with no contact. Then press **F2 Direct**, and the cursor moves to X-field immediately. Use the up and down keys to select the desired direction.
2. Start the edge search of the first point: Press **F3 First Point**, and the platform moves in the edge search direction set in 1. At this time, the system status field of the screen displays **Machining**. Meanwhile, the prompt field displays **Detecting!** until it touches the edge of the workpiece. Then the first point search is completed. At this time, the function key **F3 First Point** displays in white, which means no function, and the coord value of the first point has been recorded in the system.
3. Set W safety height: When completing the edge search of the second point, electrode rise the safety height first and then moves to the corner of the workpiece. Due to this, in order to avoid colliding with the setting of the workpiece, the W safety height cannot be set to 0.
4. Set the edge search of the second point: the same as in description of 1. Move to the edge search of the second point near the workpiece without contact. Press **F2 Direct** to set the edge search direction.
5. Start the edge search of the second point: Press **F4 Second Point**, and the platform moves to the edge search direction set in 3. until it touches the edge of the workpiece. Then the search of the second point is completed. At this time, electrode rise the safety height first first and then moves to the corner of the workpiece. After the motion is completed, the status field of the system displays **READY**. At the same time, the prompt field displays **Corner OK!**.

When the motion is started, press **F1 Reset** any time to stop the motion. At this time, the system stops and the status field of the system displays **READY**. Meanwhile, the prompt field displays **Break Detecting!**. When the edge search of the second point is started, reset the motion of the edge search. The edge search value of the first point is still kept. At this time, the function key **F3 First Point** still displays in white. Unless it is restarted again, the edge search value of the first point can be cleared, too. At this time, the



function key F3 First Point changes to normal red words and the function restored.

4.2.1.5. MIRROR

Mirror is mainly suitable for that some mold can not be machined by the front face, only by back face. In this case, operators can use X or Y axis mirror function to transfer the original coordinate to the other direction of X or Y axis.

Assuming the original program as follows

```
G92 X0.0 Y0.0;  
G00 X10.0 Y10.0;  
M36;  
G00 X10.0 Y20.0;  
M36;  
G00 X10.0 Y30.0;  
M36;  
G00 X20.0 Y15.0;  
M36;  
G00 X20.0 Y25.0;  
M36;  
G00 X30.0 Y20.0;  
M36;
```

After mirror function, the machining effect is as figure 4-2-13. The original holes before mirror function(gray color) are on the right side of X axis. After X axis mirror, the holes (black color) are on the left side of X axis.

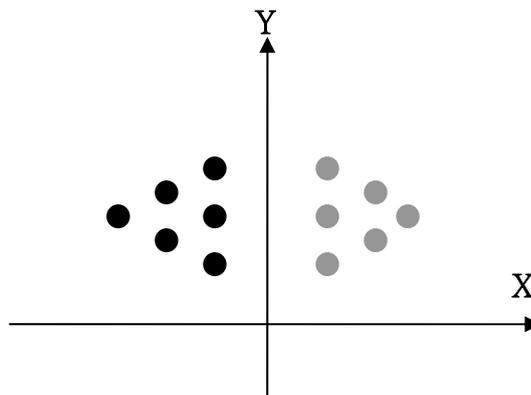


Figure 2-2-13 Mirror

Assuming the work coordinate is(0.0, 0.0) and the mach. coordinate is (-10.0, -10.0) before machining. The coordinate of the original machining holes are as following table. After running the X axis mirror, the machining holes are mapping to the other side of X axis by Y axis. The coordinate of mirrored holes are as the table.

hole	Original coordinate		Y axis Mirrored coordinate	
	Work	Machine	Work	Machine
1	(10,10)	(0,0)	(-10,10)	(-20,0)
2	(10,20)	(0,10)	(-10,20)	(-20,10)
3	(10,30)	(0,20)	(-10,30)	(-20,20)
4	(20,15)	(10,5)	(-20,15)	(-30,5)
5	(20,25)	(10,15)	(-20,25)	(-30,15)
6	(30,20)	(20,10)	(-30,20)	(-40,10)

The operating screen is as figure 4-2-14, divided into 01 MirrorCancel , 02 X Axis Mirror , 03 Y Axis Mirror and 04 XY Axis Mirror four items .

The setting shows in the second field of Ref. in Position speed and electrode setting display field. When mirror is cancel, it show "00"; it show "X" in X axis mirror, show "XY" in X and Y axis mirror.



Figure 4-2-14 Mirror Screen

Operating instructions:

Press **F6 Mirror**, then choose the mirror function by up and down keys. When mirror is cancelling, it show the message "Mirror Cancel!", It show "Mirror Enable" in running mirror function, Now, the mirror is active, do the machining.

4.2.1.6. Set Check Point

Set Check Point is mainly suitable for setting the check point in easy method. About the check point, please refer the check point parameters in 4.8.2. CUSTOM. This function is mainly setting the check point coordinate after calibration. The screen is as figure 4-2-15.

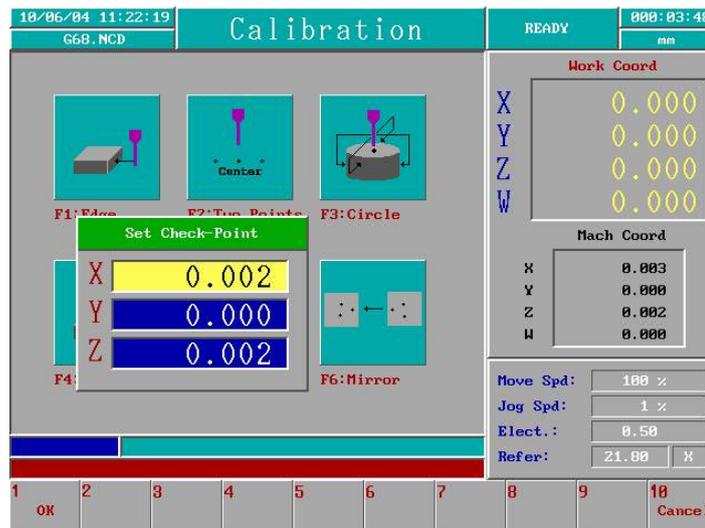


Figure 4-2-15 Set Check Point Screen

Operating instructions:

Please move the electrode to the position above the check point. Then press **F7 Set Check Pt**. In this moment, system copy the machine coordinates to the Set Check Pt window as figure 4-2-15. Operator can modify the check point coordinate by numeral key in each field. Finally, press **F1 OK**, to set these coordinates to check point. Review these settings in CUSTOM frame. If operator wants to save the check point coordinate, please press **F2 Save**; If no save, it recover to original value after rebooting up.

4.2.2. HOME SEARCH

This function allows an operator to search for the origin point of the machine. To make the soft-limit run normally, it is necessary to start the motion of home search. It is recommended to start home search after turning on the system. This can avoid the occurrence of error resulting from the operation.

Home search can be divided into five modes of home search **X**, **Y**, **Z**, **X+Y+Z** and **W**. The operating screen is as shown in Figure 4-2-16 below. Press **F2 X**, **F3 Y**, **F4 Z**, **F5 X+Y+Z** or **F6 W** to select and start the desired direction of home search. When the function of home search is started, the system status field of the screen displays **Machining**. Meanwhile, the prompt field displays **Home searching, waiting...**. When the motion is completed, the status field of the system displays **READY**. Meanwhile, the prompt field displays **Home searching Finish!**, and the coord value of the machine is cleared to zero.

When the function of home search is started, press **F1 Reset** to stop the motion. The status field of the system displays **READY**. Meanwhile, the prompt field displays **Break home searching**.



Figure 4-2-16 Home Search Screen

4.2.3. POSITION

An operator can use the function to move the electrode quickly to the set coord (can be set as work coord, rel. coord or machine coord) and to set the movement position or distance of X, Y Z and W respectively. The operating screen is as shown in Figure 4-2-17 below.

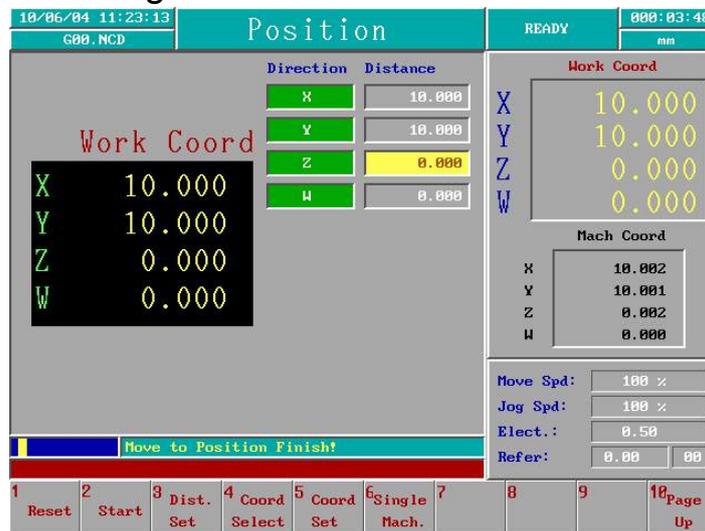


Figure 4-2-17 Position Screen

Operating instructions:

1. Coord selection: Select coordinate to move. Press **F4 Coord. Select** and the pull-down menu appears. Use the up and down keys to select the desired movement coordinate. After the selection is confirmed, the coord display area located in the left is immediately updated to the coordinate.
2. Coord set: can be set in Step 1. The selected coord is the current coord value, but the machine coord cannot be set. Press **F5 Coord. Set** and the window for setting the coord is displayed. Use the up and down keys to select the desired input the spindle of the coord value and then input the value by numeric keys. Finally, press **F1 OK**, and the set coord value be updated to the coordinate in the position mode window.
3. Distance set: Set the desired moving distance or position. If it is a relative coordinate, the setting value refers to the moving distance. If it is set as a work coord or machine coord, it is the position coord of the coordinate. Press **F2 Dist. Set** and the cursor moves to the field of the moving distance. Use the up and down keys to select the moving



direction and then input the value by numeric keys.

4. Start position: Press **F2 Start**, and the system automatically move the electrode to the original set point. When the function is started, the system status field of the screen displays **Machining**. Meanwhile, the prompt field displays **Move to position, waiting...**. After the motion is completed, the status field of the system displays **READY**. The prompt field displays **Move to Position Finish!** ◦

For instance: select the work coordinate and set the distance as X:10.0 Y:20.0 Z:30.0 W40.0. Then press **F2 Start**, and the system starts from the position of X, Y, Z to the work coord of X10.0 Y20.0 Z30.0 W40.0.

The function key **F6 Single Mach.** provides an operator with the position function. Move the electrode to the desired mach. point and then press **F6 Single Mach.** to start the steps of Single Mach. Regarding the detailed description about Single Mach., please see 4.2.6.SINGLE MACHINE in Chapter 4.

4.2.4. ELECTRODE

This function provide operator to set the electrode size, further change the electrode. In 4 axis close-loop control machine with AGC, it provide user to change guide automatically. The screen is as figure 4-2-18. The screen of no AGC machine is as figure 4-2-19.

Operating instructions:

1. Input the electrode size : key in the electrode size value by numeral.
2. Run the change process : if changing electrode, please move the cursor to Change New Electrode field, and press **F2 Start** to start the change process. In AGC machine, operator can move the cursor to Change Guide and Elect., and press **F2 Start** to start the change guide and electrode process.

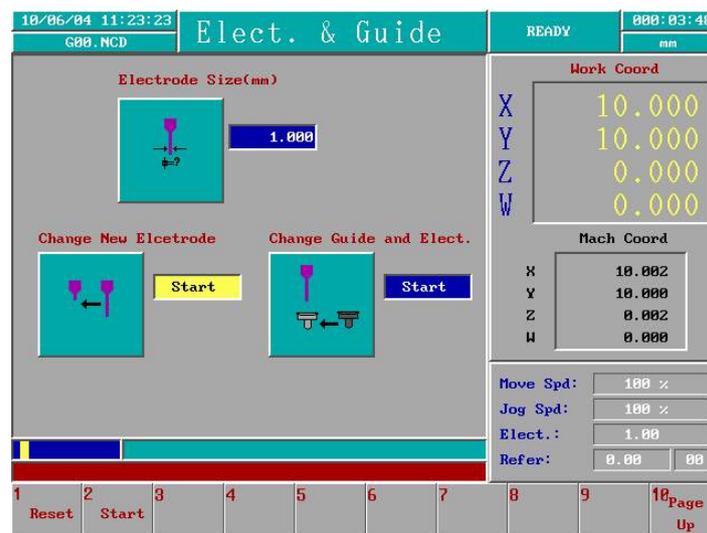


Figure 4-2-18 Change Guide and Electrode Screen

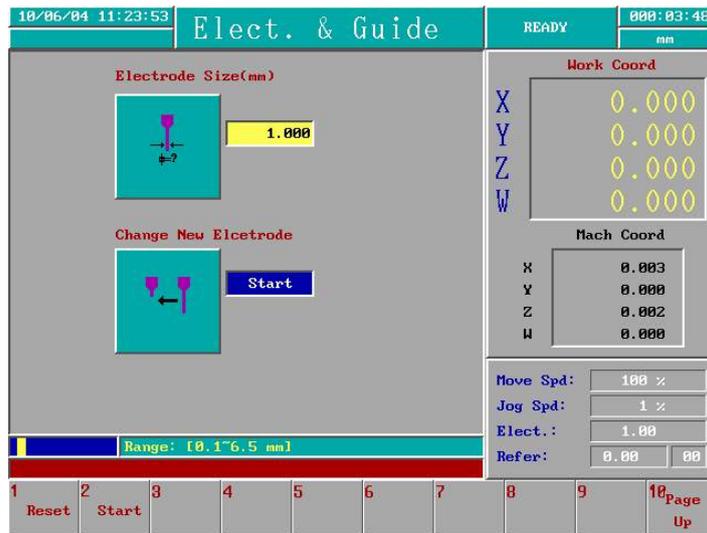


Figure 4-2-19 Change Electrode Screen

4.2.5. MOVEMENT SPEED

The function is to provide an operator with setting the move speed of the electrode during machining. The screen is as shown in Figure 4-2-20 below.

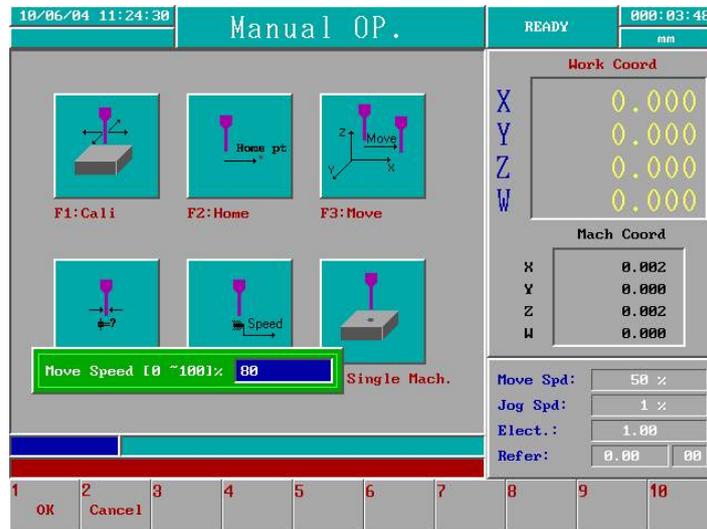


Figure 4-2-20 Move Speed Screen

Operating instructions:

1. Input the value of move speed: Input the value of move speed by numeric keys.
2. Make sure of the input value: Press **ENTER** or **F1 OK** to make sure of the input size value, and the move speed field in the right bottom corner is updated as the input size value immediately. If **ESC** or **F2 Cancel** is pressed, the window for setting is closed.

4.2.6. SINGLE MACHINE

This function provides the system with machining a single hole. The purpose is to start a single mach. or the test of workpiece material spark param. Through the spark mach. test of a single hole, it can search the best spark param and the amount of electrode consumption for formal mach. param set as shown in Figure 4-2-21.

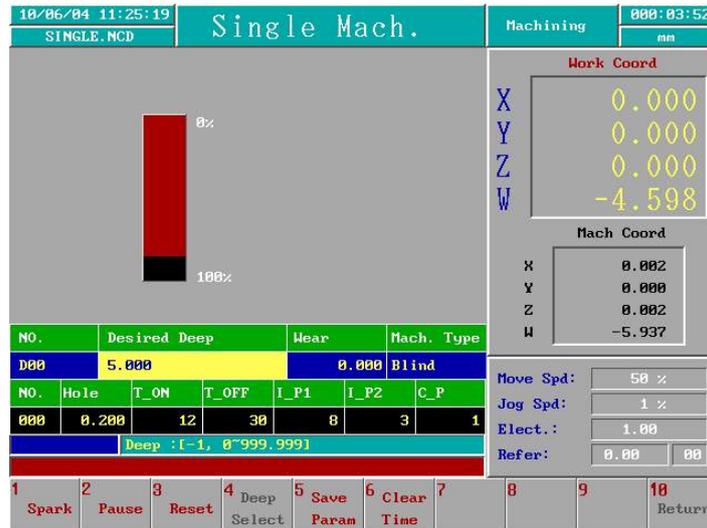


Figure 4-2-21 Single Mach Screen

In the upper part of the single mach screen, there is a bar chart (black background, red foreground), which can display the current mach. progress (0%~100%). The length of the red foreground stands for mach. progress.

In the lower part of single mach. screen is the settings of the currently used condition table and deep table. The system can store multiple sets of mach. deep table settings and mach. condition table settings. During the spark machining, it will load one set of deep table and mach. condition table settings respectively. The first two rows of the table are the setting value of the deep table currently loaded. The last two rows are the setting value of the set of condition table currently loaded. The NO. field of deep table is the group no. of the deep table currently loaded, and the NO. field of condition table is the group no. of the condition table currently loaded. Regarding to the description of condition table, please see 5. Condition Table in Chapter 4. About the description of deep table, please see 6. Deep Table in Chapter 4.

Under the single mach. mode, to facilitate a user in operating, the setting of condition table and deep table currently used can be modified and saved



directly. However, it is not allowed to modify depth setting during machining. In addition, if the current file of condition table is COND.DAT, the condition setting cannot be saved to COND.DAT. Regarding the modification of Def. Cond COND.DAT, please see 5.Condition Table in Chapter 4.

Operating instructions:

1. Set mach. deep: Press **F4 Deep Select**, the window “Input the desired depth code” is displayed. After the group no. of the deep table is input, press **ENTER** or **F1 OK** and the depth set value of the group is loaded. The other setting method is to use the left and right key to switch each field of deep table. And then input the param set of deep table respectively. Finally press **F5 Save Param.**, and the input depth set is saved to the depth value of the group. This can facilitate an operator in setting the depth value of the group in the single mach. screen.
2. Set mach. condition: Use the up and down keys to move the cursor to the row for setting the mach. condition table. After using the left and right key to move the cursor to the **NO.** field of the condition table, use the up and down keys to switch and load the condition value setting in sequence. Or use the left and right keys to move the cursor to each field of the condition table, and then input the condition value setting respectively, in accordance with the hint of the prompt field. Finally, press **F5 Save Param.**, and the input condition setting is saved to the cond value. If the current condition table file is Def. Cond COND.DAT, saving does not function here.
3. Clear M. Time: The work time field is placed in the upper right part of the screen, recording the total mach. time. To understand the time required by a single mach, it is necessary to set the total mach. time to zero. Press **F6 Clear M. Time**. Thus it is able to return the total mach. time to 0.
4. Spark mach: Press **F1 Spark**, and start a single spark mach immediately. During the spark mach process, the spark param of the condition table can be altered at any time, but the mach. depth setting of the deep table cannot be altered. Press **F5 Save Param.**, and save the condition value and depth value of the group. If the file of condition

table is COND.DAT, the condition table cannot be saved. The function key of the system provides the functions of pause and reset. During machining, press **F2 Pause**, and the system stops spark mach. immediately,. Then press **F1 Spark**, and the system continue the spark mach. If **F3 Reset** is pressed during machining, the system opens a prompt window as shown in Figure 4-2-22. Press the function key **F1 Yes**, and the system is reset. Press the function key **F2 No**, and the system continues mach. You can observe the mach. completion percentage in the upper part of the window to see the current mach. status and progress.



Figure 4-2-22 Machining Reset Screen

Assuming the machining has not arrived at the machining deepness in drilling, but the operator does not want to machine this hole. Operator can press **F7 Hole OK**, and it will move the electrode upward to safe position above the workpiece.

4.3. COORD

An operator can use this to set the coord content displayed in the upper part of coord display area on the right of the screen. The screen is as shown in Figure 4-3-1 below.

In the main menu, press **F2 Coord**. The screen opens the selection box **Select coordinate to show**. Use the up and down keys on the panel to select the coordinate to show. When **ENTER** or **F1 Ok** is pressed, the system updates the coordinate in the upper part of the coord display area. For instance, select **Show working coord** as the working coord value shown in Figure 4-3-1. If **Coord Value Set** is selected, display the **Coord Set** window for the coord value of the current coordinate input and setting.

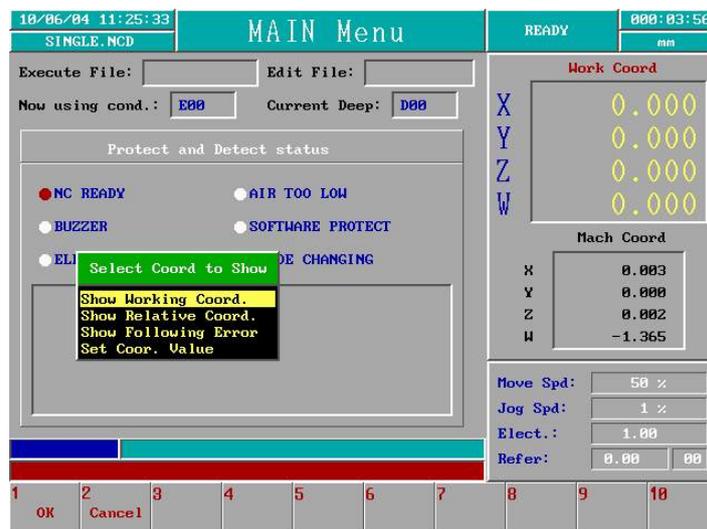


Figure 4-3-1 Coord Screen

4.4. EDIT

This model provides an operator with the functions of editing mach. prog and of retrieving and editing files.

In the main menu, press **F3 Edit**, and the system enters the prog edit mode. This provides an operator with the integration environment of edit mach prog and the function of file handling. Under the prog edit mode, it is able to directly key in English characters or edit the mach. prog by the numeric key method. Or enter the dialog edit mode and edit in dialog window method. In addition, it still provides the relevant auxiliary handling functions of file, insert line, delete line, copy mark, etc. to facilitate an operator in operating and editing. When entering the edit mode of the prog, the screen is as shown in Figure 4-4-1. The following describes each function key:

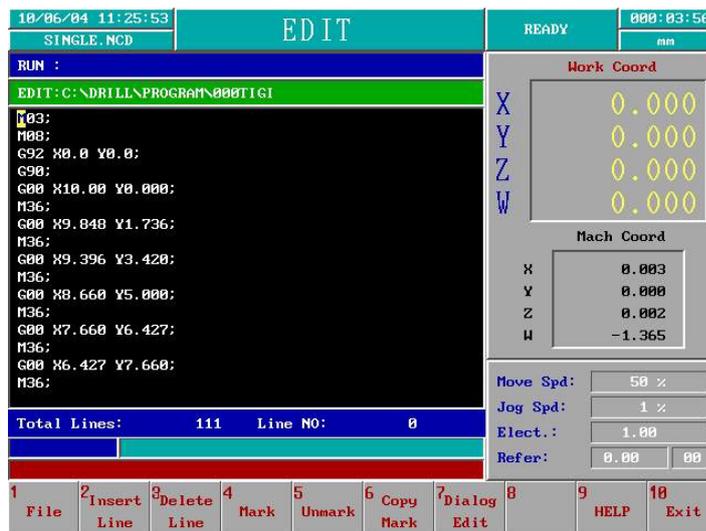


Figure 4-4-1 Edit Mode Screen

The following program is suggested. It is proposed that the starting blocks are M03(spindle CW) or M04(spindle CCW) and M08(fluid ON), the end block is M02(program end, fluid off and spindle stop).

M03;	→Spindle CW
M08;	→Fluid ON
G92 X0.000 Y0.000 Z0.000 W0.000;	→Set Work coordinate to (0,0,0)
M36 D0 E0;	→Drilling with D Group 0 and E Group0
G00 X10.000 Y0.000 Z0.000;	→Absolute Position (10,0,0)
M36 D0 E0;	→ Drilling with D Group 0 and E Group0
G00 X0.000 Y10.000 Z10.000;	→ Absolute Position (10,0,0)
M36 D1 E1;	→ Drilling with D Group 0 and E Group0
G00 X10.000 Y10.000 Z10.000;	→ Absolute Position (10,0,0)
M36 D1 E1;	→ Drilling with D Group 0 and E Group0
M02;	→ program end, fluid off and spindle stop

This program is variable deepness machining with 4 holes in work coordinate (0,0,0), (10,0,0), (0,10,10) and (10,10,10). The group of condition table is 0 or 1 and the group of deepness table is 0 or 1. About the deepness table and condition table, please refer to 4.6 DEEP and 4.5 CONDITION.

- ◆ **FILE:** After pressing **F1 File**, enter the relevant handling window to the file. It is used in handling the relevant handling work to the file. Regarding the detailed description, please see 4.4.1 File.
- ◆ **INSERT LINE:** After pressing **F2 Insert Line**, insert a space line into the prog block where the current cursor places.
- ◆ **DELETE LINE:** Press **F3 Delete Line** and delete the prog block where the current cursor places or the marked prog area.
- ◆ **MARK:** After pressing **F4 Mark**, mark the prog block of single line or multi-line. It is used in marking the desired prog block which needs being marked, deleted or executed.
- ◆ **UNMARK:** After pressing **F5 UnMark**, cancel the selected area.
- ◆ **COPY MARK:** After pressing **F6 Copy Mark**, copy the currently marked prog block to the prog block where the cursor places currently.
- ◆ **DIALOG EDIT:** After pressing **F7 Dialog Edit**, enter the dialog edit mach prog window. The dialog window method facilitates an operator in editing mach. prog. About the detailed description, please see 4.4.2 Dialog Edit.

4.4.1. FILE

This provides five functions of file retrieving , copying or deleting, etc. This includes open a file, save a file, save as another file, copy file, or delete file.

Under the edit mode, press **F1 File** to enter the file mode. The screen is as shown in Figure 4-4-2. The description of each function key of the file mode is as follows:



Figure 4-4-2 Prog Edit-File Screen

(1) LOAD FILE

The function can load the workpiece prog on the disc into the system memory. Press **F1 Load File** to start the function. The screen is as shown in Figure 4-4-3.

Operating method:

Use the left and right keys to select the work field. The upper file name field is to display the desired open file name. The left bottom file name field is the relevant file of the file folder. The desired file to open can be selected. The disc folder name list field in the right bottom is the folder directory of each disc for switching to each disc. When operating, use the up and down keys to move the cursor to select the file or disc folder name. Open file method one: use the left and right keys to move the cursor to the bottom file name field, and then use the letter key or numeric key on the panel to input the complete file name directly. Press **ENTER**. Method two is to

move the cursor to the left bottom field and then use the up and down keys to select the file. Then use **ENTER** key to select the file name where the cursor places.



Figure 4-4-3 Open File Screen

(2) SAVE FILE

The purpose of the function is to save the mach. prog file in the system memory to the disc and to display the prompt that the file saving is completed on the screen. Press **F2 Save File** to start the function.

Note : When system is in background edition, in order to avoid influencing the executing program, the executing program can be edited but not be saved. After finishing execute, it can be saved.

(3) New File

The purpose of the function is to clear the editing text and open a new file to provide editing. Press **F3 New** to start the function.

(4) SAVE AS

The purpose of the function is to save the mach. prog file in the system memory to another new file. Use **F3 Save As** to start the function and the operating method of the operating window for Input or select new file name is different from that of open file. If you succeed in saving the file, the report of file saving completed is displayed on the screen.

(5) COPY FILE

The purpose of the function is to copy a file. Use **F4 Copy File** to start the function. The window screen is as shown in Figure 4-4-4. The system requires the operator to input the original file (source file) and target file. Press **F1 Source** to open file selection window and select the source file. Then press **F2 Target** to select or set the target file. After selecting and setting the source file and target file, press **F3 Copy File**, and the system produces a file with the same content as that of the source file. The file name of the file is that of the target file input by the operator. The diagram below is the operating screen of the function. The third field is prompt field. If the source file does not exist, press **F3 Copy File** and the message **Copy file error!** Is Displayed.

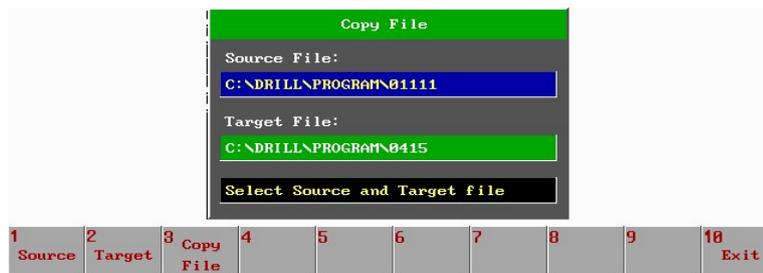


Figure 4-4-4 Copy File Screen

(6)DELETE FILE

The purpose of this function is to delete a file. Press **F5 Delete File** to start the function. The system requires the operator to input the file name which needs being deleted. The operating method is the same as that of open file. However, after selecting the file that needs to be deleted, press **ENTER** and the prompt window **Are you sure to delete the file?** to avoid deleting the wrong file. If you are sure of deleting it, please press **F1 YES**; otherwise, please press **F2 NO**.

(7) DXF Transfer

The purpose of this function is to transfer the DXF file to Drilling program. The detail is in 4.4.2.DXF.



(8) Wire Cute

The purpose of this function is to transfer the wiring hole of Wire Cut program to Drilling program. The detail is in 4.4.3.Wire Cut.

4.4.2. DXF File

DXF is a common-using file, and has the fixed structure. It is the communicating file between the different drawing software. The most application is to transfer to DXF after drawing in AutoCAD.

The purpose of the function is to transfer the point coordinate in DXF to the drilling point with GM Code. Beside, only the point in that layer can be transferred and other data is omitted.

In transferring the point coordinate, only the X、Y and Z coordinate of that point in that layer can be done. For example, the point coordinate is (10.0, 20.0, 30.0). After transfer, the instructions is as follows.

```
G00 X10.000 Y20.000 Z30.000;  
M36;
```

The meaning is moving to (10.0, 20.0, 30.0) in G00, and drilling in M36.

In the edit mode/file frame, press **F7 DXF**, then enter the DXF mode. The frame is as figure 4-4-5. The example in it transfer the point of C:\DXF\R14F.DXF in "0" layer. The explanation of the function key is as follows 下：

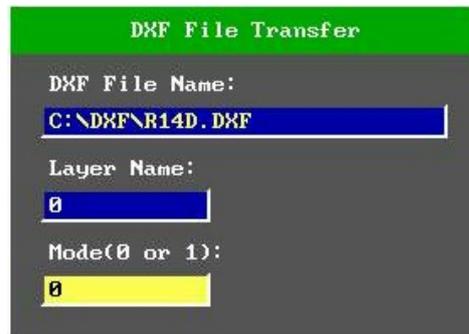


Figure 4-4-5 DXF frame

- ◆ **OK**: Press **F1 OK**, it is going to transferring the point coordinate of DXF file in that layer to drilling GM Code.
- ◆ **DXF File** : Press **F2 DXF File**, and give out the choosing window. And choose the transferred DXF file.
- ◆ **Layer** : Press **F3 Layer**, the cursor move to the layer field. Then key in the layer name with numeral or English word.
- ◆ **Mode** : Press **F4 Mode**, the cursor move to the mode field. Then key in the mode with numeral. There are two modes, 0 or 1. In mode 0, the normal situation, before starting Cycle Start, the electrode should be moved to the



origin of work coordinate. In some special case, the electrode can't be moved to the origin because the stroke is not large enough. The positions of the drilling holes are all inside the stroke. In this case, it must take mode 1, and before starting Cycle Start, please move the electrode above the first point

Mode 0 : When starting to execute program, the electrode should be in the origin of the work coordinate. Therefore, move the electrode to the origin before starting Cycle Start, if choosing Mode 0.

Mode 1 : When starting to execute program, the electrode should be above the first point. Therefore, move the electrode above the first point before starting Cycle Start, if choosing Mode 1.

Note : the programs are different in this two mode. Please notice to set the mode.

Operating instructions:

1. Choose DXF File : Press **F2 DXF File**, and it gives out the "choose file" window. Use direction key or key in the DXF file name.
2. Set Layer : Press **F3 Layer**, the cursor move to the layer field. Please set the layer name in English word or numeral.
3. Set Mode : Press **F4 Mode**, the cursor move to the mode field. Please set "0" or "1".
4. OK : Press **F1 OK**, then the point coordinate in that layer of the file will be transfer to the drilling program in GM Code.
5. Save File : Press **F3 Save File** to save the drilling program transferred from DXF as a file.
6. Execute the Program : Switch to the Multi-Hole mode, and load the program transferred from DXF file. According to the mode, move the electrode to the origin of work coordinate in mode 0 or move it to the point coordinate of the first hole. Then Press Cycle Start to execute the program.

Transferring Result :

1. If the transfer is finishing, the point coordinates in that layer are transferred to the drilling point with GM code instruction in the edit window. There are two examples to explain the meaning of the GM Code in mode 1 and mode 2.



Mode 0 :

M03;	→Spindle CW
M08;	→Pump ON
G92 X0.000 Y0.000 Z0.000 W0.000;	→Set the Origin of Work Coordinate
G00 X10.000 Y20.000 Z30.000;	→Absolute Position to the First Point
M36;	→Drilling
G00 X20.000 Y30.000 Z40.000;	→Absolute Position to the Second Point
M36;	→Drilling
.....	→Omitting
M02;	→Program End

Mode 1 :

/* 1ST POINT: X293.747 Y242.375 Z0.000 */	→Hint of the First Point Work Coordinate
M03;	→Spindle CW
M08;	→Pump ON
G92 X0.000 Y0.000 Z0.000 W0.000;	→Set the First Point as the Origin
G00 X0.000 Y0.000 Z0.000;	→Absolute Position to the First Point
M36;	→Drilling
G00 X10.000 Y10.000 Z10.000;	→Absolute Position to The Second Point
M36;	→Drilling
.....	→Omitting
M02;	→Program End

2. No setting in DXF file name, layer name or mode, then press **F1 OK**, it shows "file name, layer name and mode aren't completely!!" error message and back to edit window.
3. If there is no point in DXF file or in that layer, it give out "No Point Data or Not In This Layer!" error message.
4. DXF files are the fixed structure files. The wrong DXF file results in the wrong point coordinate. To avoid this error, it is going to stop the transfer, and show "This DXF file has errors!" error message.

4.4.3. Wire Cut Program

Before machining in wire cut machine, some mold need to machining the initial hole so that they can be through the wire. Some The application of the drilling EDM is machining the initial hole in wire cut machine.

User draws the drawing in software CAD, then transfer the drawing to the wire cut program. In drilling EDM machine, user can transfer the initial hole to drilling program with this function. The other codes in the wire cut program will be skipped.

In transferring, the system read the wire cut file. When it read the G92 command, the G92 commands are considered as initial hole command. So as it is transferring the XY value after G92 to the drilling coordinate. For example, there is a block as shows.

```
G92 X10.000 Y20.000
```

After transferring, the drilling code is this two block as

```
G00 X10.000 Y20.000
```

```
M36;
```

The meaning is running G00 and moving to (10.0, 20.0), and running M36 to drill at that point.

In the edit mode/file mode, press **F8 Wire Cut** to enter this mode. The frame shows as figure 4-4-6. The example in that figure transfer the wire cut file, C:\DRILL\PROGRAM\AUTO14. The function keys will be explained as follows.



Figure 4-4-6 the transferring Wire Cut Program Frame

- ◆ **Choose File** : Press **F1 File**, and it gives out the “choose file” window. Use direction key or key in the Wire Cut file name.
- ◆ **OK** : Press **F2 OK**, start to transfer the initial hole of the wire cut file to the



drilling hole in GM Code in drilling machine and show them in the edit window.

Operating instructions:

1. Choose File : Press **F1 File**, and it gives out the “choose file” window. Use direction key or key in the Wire Cut file name.
2. Set G92 Coordinate: move the cursor to the G92X Y Z fields with direction key, and set the value in number key.
3. Set Deepness Group : move the cursor to the D field, set the deepness group in number key.
4. Set Condition Group: move the cursor to the E field, set the condition group in number key.
5. OK : Press **F1 OK**, then the initial hole coordinate of the wire cut file will be transfer to the drilling program in GM Code.
6. Save File : Press **F3 Save File** to save the drilling program transferred from wire cut file as a file.
7. Execute the Program : Switch to the Multi-Hole mode, and load the program transferred from wire cut file. Then Press Cycle Start to execute the program.

Transferring Result :

If the transfer is correct, the coordinate of initial hole will be transferred to the drilling hole in GM Code, and the program shows in the edit window. The wire cut program and the drilling program transferred shows as follows.

```
%%  
(0273-DA)  
N1G92X-2.185Y9.201  
G41G90G01X-1.937Y8.6  
X-1.55Y8.76  
G03X-1.413Y9.228I-0.12J0.289  
G01X-2.533Y10.839  
G03X-3.098Y10.605I-0.257J-0.178  
G01X-2.751Y8.674  
G03X-2.323Y8.44I0.308J0.055  
M00  
G01X-1.937Y8.6  
M00
```



X-2.185Y9.201G40
M00
G00X-4.038Y7.963
M00

N2G92X-4.038Y7.963
G41G90G01X-3.578Y7.503
X-3.283Y7.799
G03X-3.335Y8.284I-0.221J0.221
G01X-4.986Y9.344
G03X-5.419Y8.911I-0.169J-0.264
G01X-4.359Y7.26
G03X-3.874Y7.207I0.263J0.169
M00
G01X-3.578Y7.503
M00
X-4.038Y7.963G40
M00
G00X-5.276Y6.11
M00

The drilling program is as follows after transferring.

/* Wire Cut file transfer program start */	→The Starting Hint of Transfer
G92 X10.000 Y20.000 Z-10.000;	→Set the Work Coordinate
M03;	→Spindle CW
M08;	→Pump ON
G00 X-2.185 Y9.201;	→Move to Initial Hole in Wire Cut File
M36 D1 E1;	→Drilling With D Code and E Code
G00 X-4.038 Y7.963;	→Move to Initial Hole in Wire Cut File
M36 D1 E1;	→Drilling With D Code and E Code
/* Wire Cut file transfer program end */	→The End Hint of Transfer

4.4.4. DIALOG EDIT:

Use the dialog window method to facilitate an operator in editing mach. prog. This is in order to save prog and editing time.

Under the edit mode, after pressing **F7 Dialog Edit**, enter the dialog edit mode. Under the dialog edit mode, there are various kinds of settings or the dialog windows of motions. When the dialog window is completed the input, press **ENTER** or **F1 OK** and then the system automatically produce the prog codes of the motion in the edit mode window. The dialog edit screen is as shown in Figure 4-4-7. The description of each function key in the dialog edit mode is as follows:

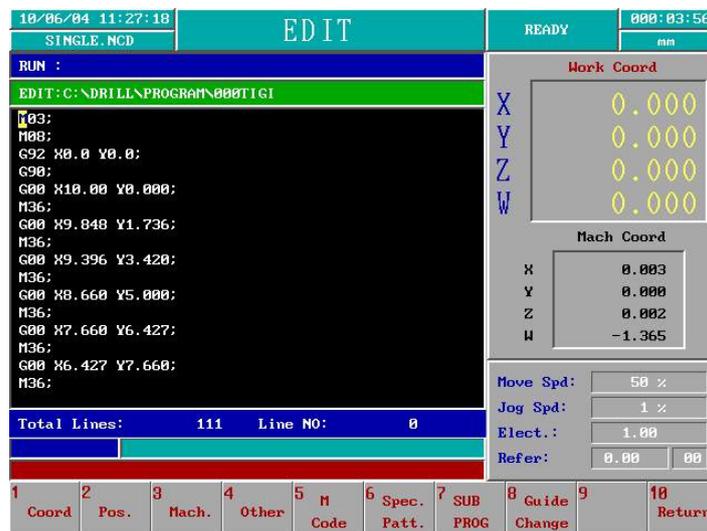


Figure 4-4-7 Dialog Edit Screen

4.4.4.1. COORDINATE

After pressing **F1 Coord.**, the dialog input box of generating **COORDINATION** provides an operator with setting the coordinate and its coord value in mach. prog. There are total four types: **Work Coord. Setting**, **Work Coord.**, **Rel Coord.** and **Check PT Setting**. When setting each field, use the up and down keys to move and select the desired field for setting. The respective descriptions of the four types are as follows:

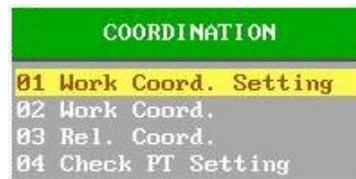


Figure 4-4-8 the Coordination dialog frame

Work Coord. Setting: Allow an operator to set the working coord value of the current position. The motion method is G92. The frame is as figure 4-4-9. For instance, when the input prog code is G92 X0.0 Y0.0 Z0.0 W0.0, the current working coord is set as (0.0, 0.0, 0.0, 0.0) so the function can be used to set the original point of the working coord. The set method is to input each spindle coord value of the desired coord in the X, Y, X, and W field.

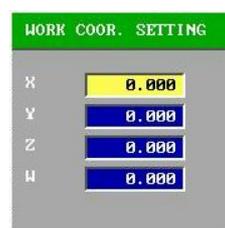


Figure 4-4-9 the Work Coord. Setting input frame

Work Coord: The mach. prog after being set the prog single block, its coordinate mode is a absolute coord until the coordinate is altered as the prog single block. The motion method is G90.

Rel. Coord: The mach. prog after being set the prog single block, its

coordinate mode is a relative coord until the coordinate is altered as the prog single block. The motion method is G91.

Check PT Setting: The function is to set the check point of the electrode. The check point coordinate is machine coordinate. The frame is as figure 4-4-10. The motion method is G06. If it is set G06 X1.0 Y1.0 Z1.0 in mach. prog, when the elect runs out, the electrode auto moves to the Pt. (1.0, 1.0, 1.0) and calibrate the electrode leng. The set method is to input each spindle coord value of the checking Pt. respectively in the coord field of X, Y and Z. Regarding the relevant function and setting of checking Pt., please see 9.2. Custom param. in Chapter 4 and 1. Machining manual replacement for the electrode is in Chapter Five.

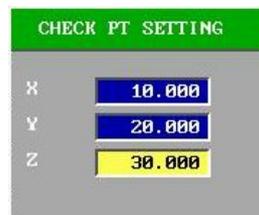


Figure 4-4-10 the Check PT Setting input frame

4.4.4.2.POSITION

After pressing **F2 Pos.**, the dialog choice window of **Pos.** appears. This allows an operator to select the desired function for position in the mach. prog. There are four modes, **Work Coord. Posi.**, **Mach. Coord. Posi.**, **Inner Center** and **Edge**. The followings are description of them.

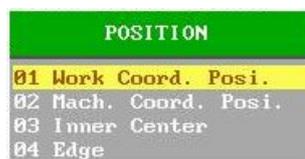
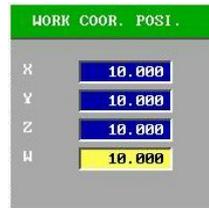


Figure 4-4-11 The Position Dialog Selecting Frame

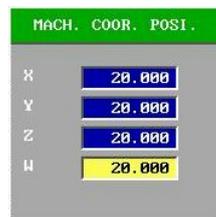
Work Coord. Posi. : The command is G00, provide operator to move to that point in work coordinate. The frame is as figure 4-4-12. The set method is to input each spindle coord value of the desired coord in the X, Y, X, and W field.



WORK COOR. POSI.	
X	10.000
Y	10.000
Z	10.000
W	10.000

Figure 4-4-12 the Work Coord. Posi. Input Frame

Mach. Coord. Posi. : The command is G53, provide operator to move to that point in machine coordinate. The frame is as figure 4-4-13. The set method is to input each spindle coord value of the desired coord in the X, Y, X, and W field.



MACH. COOR. POSI.	
X	20.000
Y	20.000
Z	20.000
W	20.000

Figure 4-4-13 the Mach. Coord. Posi. Input Frame

Edge : The command is G80, provide operator to edge the workpiece. The frame is as figure 4-4-14. The operator needs to choose the edging axis.



EDGE	
1	X+ edge
2	X- edge
3	Y+ edge
4	Y- edge
5	Z+ edge
6	Z- edge
7	W+ edge
8	W- edge

Figure 4-4-14 the Edge Choice Frame

Inner Center : The command is G33, provide operator to locate the center of the groove or hole. The frame is as figure 4-4-15. The operator needs to choose the locating method.



INNER CENTER	
01	In. Center X+X-
02	In. Center Y+Y-
03	In. Center X+X- Y+Y-

Figure 4-4-15 the Inner Center Frame

4.4.4.3. MACHINING

After pressing **F3 Mach.**, the dialog window of **Mach.** appears. This allows an operator to set the group no. of the loaded deep table and condition table. The D value stands for the deep table, and E value stands for the condition table.

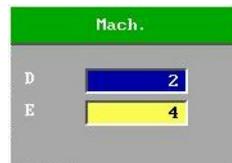


Figure 4-4-16 the Machining Dialog Frame

4.4.4.4. OTHER

Press **F4 Other**, Give out the **OTHER** choice window, including **Pause** , **Note** and **Change Cond.** .

Pause : The Command is G04, provide operator to pause for seconds. The frame is as figure 4-4-17. The operator needs to input the pause seconds.



Figure 4-4-17 the Pause Dialog Frame

Note : The Command is /* */ , provide operator to note the program. The frame is as figure 4-4-18. As the figure, the block command is /* CAR MOLD */. In executing, system will skip the block.



Figure 4-4-18 the Note Dialog Frame

Change Cond. : The Command is G38, provide operator to change the condition table file. The frame is as figure 4-4-19. As the figure, the block command is G38 P0011.



Figure 4-4-19 the Change Cond. Dialog Frame

4.4.4.5. M CODE

After pressing **F5 M Code**, the selection box of **M Code** appears. Use the up and down keys to move and select the desired motion. After pressing **ENTER**, auto-generate M code of the motion. Regarding M code, please refer to Table 4-1. The following is the relevant motions that M code selection box lists: **01 Program Hold**, **02 Program End**, **03 Spindle CW**, **04 Spindle CCW**, **05 Spindle Stop**, **07 Pump ON**, **08 Pump OFF**, **09 SubProgram end** and **10 New Elect. Change**.

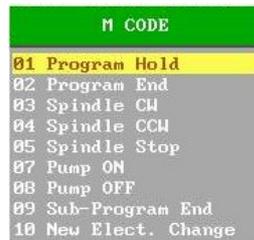


Figure 4-4-20 the M Code Selecting Frame

4.4.4.5. SPECIAL PATTERN

After pressing **F6 Spec. Patt.**, enter the dialog of a special pattern, including two modes: **Rect Pattern** and **Circle Pattern**. Press **F1 Rect Patt.** or **F2 Circle Patt.**, enter the dialog edit window respectively. The two types of dialog edit mode allow users to input by way of simple dialog and to edit the mach. prog code of the mach figure of rect patt. and circle patt. Regarding the detailed description about rect patt. and circle patt, please see 4.4.5. Rect. Pattern and 4.4.6. Circle Pattern below.

4.4.4.6. SUB PROGRAM

After pressing **F6 Sub Prog**, the dialog window of **Sub Prog** appears. This can allow the prog. to repeat calling and executing sub prog. **P** is the file name of sub prog, and **L** is the number of the execution of the sub program.



The motion method is M98.

Caution: The file name of sub prog needs to use o (Eng. letter o) as the beginning, but in P field, it does not need to input the beginning, o, of the file name.

4.4.4.7. CHANGE GUIDE

Press **F7 Change Guide**, give out the change guide dialog window. provide the operator to input the change guide command. The frame is as figure 4-4-21. As the figure, the block command is M06 T10. About the changing guide, please refer the 4.8.7. AGC.



Figure 4-4-21 the Change Guide Dialog Frame



G code	G00	Positioning
	G04	Hold for Seconds
	G05	Positioning with No-Guard
	G06	Measure Reference Point On a New Electrode
	G30	Block Skip Cancel
	G31	Block Skip Enable
	G38	Condition Table Change
	G53	Machine Coordinate position
	G80	Edge Searching
	G81	Mechanic Origin Searching
	G82	Locate Groove Center
	G83	Locate Inner Hole
	G84	Locate Outer Center
	G85	Locate Circular Cylinder Center
	G90	Absolute Coordinate
	G91	Relative Coordinate
G92	Work Coordinate Point Setting	
M code	M01	Program Hold
	M02	Machining END, Spindle Stop, Fluid Supply Stop, Return to Initial Program
	M03	Spindle CW
	M04	Spindle CCW
	M05	Spindle Stop
	M06	Guide Change
	M08	Fluid Supply Start
	M09	Fluid Supply Stop
	M20	Electrode Change
	M36	Machining a Hole
	M98	Calling sub-program
	M99	Sub-program end
D code	D0~ D99	Call the group of the deepness table
E code	E0~ E99	Call the group of the condition table

4-1 G M Code

4.4.5. RECT PATTERN

A user can edit a rect pattern composed of circle holes in simple dialog edit mode. The machined figure is as shown in Figure 4-4-22. Each solid and empty circle point stands for a machined fine hole. The screen of the dialog edit window is as shown in Figure 4-4-23. The following is the description aiming at the meaning that each input field in the dialog edit window stands for.

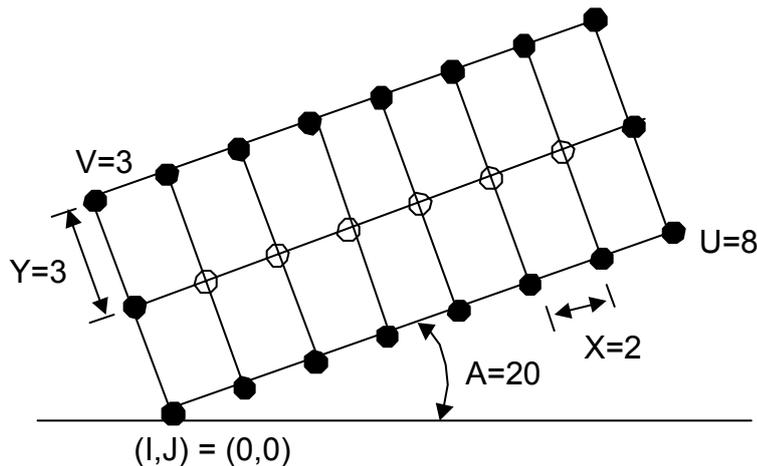


Figure 4-4-22 Rect Pattern Specification Illustration

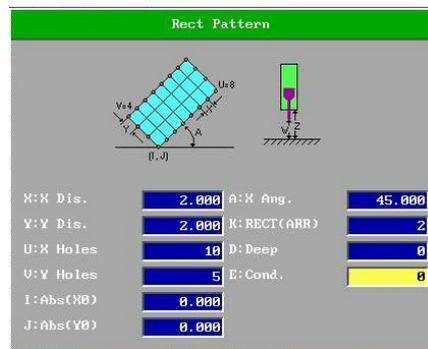


Figure 4-4-23 Rect Pattern Screen

- X: X Dis.** Set the distance between each hole in rect patt. X. In Figure 4-4-22, the distance of X is 2mm so X is 2.
- Y: Y Dis.** Set the distance between each hole in rect patt. X. In Figure 4-4-22, the distance of Y is 3mm so Y is 3.
- U: X Holes** Set the holes in rect patt. X. In Figure 4-4-22, the holes of X have 8 so U is 8.
- V: Y Holes** Set the holes in rect patt. Y. In Figure 4-22, the holes of Y



have 3 so V is 3.

I: Abs(X0) Set X coord of rect patt StPt. In Figure 4-4-22, the StPt of X coord is 0 so I is 0.

J: Abs(Y0) Set Y coord of rect patt StPt. In Figure 4-4-22, the StPt of Y coord is 0 so J is 0.

A: X Ang. Set X Ang between X in rect patt and X in work coord. In Figure 4-4-22, the angle between X in rect patt and X in work coord is 20° so A is 20.

K: rect (ARR) Set the type of rect (ARR). It can be set 0, 1 or 2. 0 stands for the empty array of rect pattern, and the completed figure only has solid circle points in Figure 4-4-22. 1 stands for the inner array of rect pattern and the completed figure only has empty circle points in Figure 4-4-22. 2 stands for setting a rect pattern as a solid rect pattern and the completed figure includes the solid circle points and empty circle points in Figure 4-4-22.

D: Set the group no. of used depth table.

E: Set the used group no. of used cond table.

When each field is input, and then **F1 OK** is pressed, the system auto-generates mach. prog of rect pattern in edit mode window.

The program as following is transferred by Rect. Pattern. The input data is showed in the second block G72. X Dis. is 5.0, Y Dis. is 5.0, X Holes U is 2, Y Holes V is 3, Abs(X0) I is 10.0, Abs(Y0) J is 10.0, X Ang. A is 45, rect (ARR) K is 2, D code is 0 and E code is 2.

<code>/* Rect Pattern Start */</code>	→Rect. Pattern Start
<code>/* G72 X5.000 Y5.000 U2 V3 A45.000 I10.000 J10.000 K2 D0 E2; */</code>	→Original Input Data
<code>G90 G00 X10.000 Y10.000;</code>	→Transferred Point
<code>M36 D0 E2;</code>	→Drilling
<code>G90 G00 X13.536 Y13.536;</code>	→Transferred Point
<code>M36 D0 E2;</code>	→Drilling
<code>G90 G00 X6.464 Y13.536;</code>	→Transferred Point
<code>M36 D0 E2;</code>	→Drilling
<code>G90 G00 X10.000 Y17.071;</code>	→Transferred Point
<code>M36 D0 E2;</code>	→Drilling
<code>G90 G00 X2.929 Y17.071;</code>	→Transferred Point
<code>M36 D0 E2;</code>	→Drilling
<code>G90 G00 X6.464 Y20.607;</code>	→Transferred Point
<code>M36 D0 E2;</code>	→Drilling



/* Rect Pattern End */

→Rect. Pattern End

4.4.6. CIRCLE PATTERN

A user can edit a circle pattern composed of circle holes in simple dialog edit mode. The machined figure is as shown in Figure 4-4-24. Each solid circle point stands for a machined fine hole, and the screen of dialog edit window is as shown in Figure 4-4-25. The following is the description of what each field in the dialog edit window stands for.

□

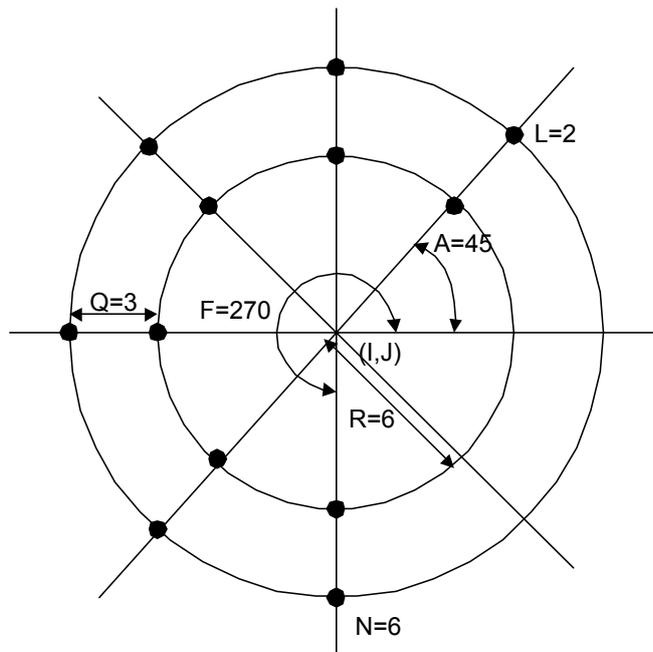


Figure 4-24 Circle Pattern Specification Illustration

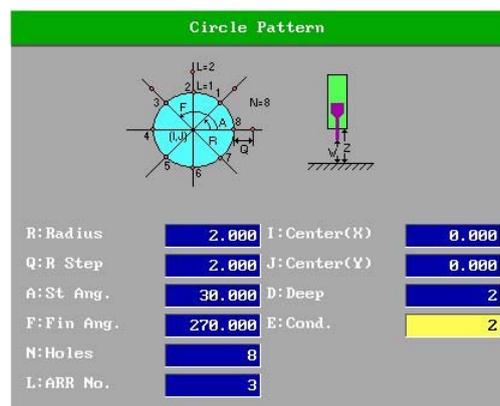


Figure 4-25 Circle Pattern Screen

R: Radius Set the radius of the inmost circle in the circle pattern. In Figure 4-4-24, the radius of the inmost circle in the array is 6mm so R is 6.

Q: R Step Set the distance between each circle in the circle pattern. In



Figure 4-4-24, the distance between the second circle and the inmost circle is 3mm so Q is 3.

A: St Ang. Set the angle (angle 0 stands for X direction of work coord) of the first hole in each circle in the circle pattern. In Figure 4-4-24, due to the angle of the first hole is 45° , A is 45. The input range is from 0 to 720.

F: Fin Ang. Set the angle of the last hole of each circle in the circle pattern. In Figure 4-4-24, due to the angle of the last hole is 270° , F is 270. The input range is larger than **A** and smaller than **A+360**.

N: Holes Set the total hole no. in each circle (including the first hole and the last hole). In Figure 4-4-24, due to the fact that each circle has 6 holes, N is 6.

L: ARR No. Set the total circle no. in the circle pattern. In Figure 4-4-24, due to the fact that there are total 2 circles, L is 2.

I: Abs(X0) Set X coord of the center in the circle pattern. In Figure 4-4-24, due to the fact that X coord of the center in the circle pattern is 0, I is 0.

J: Abs(Y0) Set Y coord of center in the circle pattern. In Figure 4-4-24, due to the fact that Y coord of the center in the circle pattern is 0, J is 0.

D: Set the group no. of used depth table.

E: Set the group no. of used cond table.

When each field is input and then **F1 OK** is pressed, the system auto-generates the prog of rect pattern or circle pattern in edit mode window.

Caution: The Fin Ang. of circle pattern must be more than the St Ang.

The program as following is transferred by Circle Pattern. The input data is showed in the second block G71. Radius R is 3.0, R Step Q. is 4, St Ang. A is 0.0, Fin Ang. F is 180.0, Holes N is 4, ARR No. N is 2, Abs(X0) I is 2.0, Abs(Y0) J is 2.0, D code is 3 and E code is 5.

/* Circle Pattern Start */

/* G71 R3.000 Q4.000 A0.000 F180.000 N4 L2 I2.000 J2.000 D3 E5; */

M36 D3 E5;

G90 G00 X3.500 Y4.598;

M36 D3 E5;

G90 G00 X0.500 Y4.598;

→ Circle Pattern Start

→ Original Input Data

→ Transferred Point

→ Drilling

→ Transferred Point

→ Drilling



M36 D3 E5;
G90 G00 X-1.000 Y2.000;
M36 D3 E5;
G90 G00 X9.000 Y2.000;
M36 D3 E5;
G90 G00 X5.500 Y8.062;
M36 D3 E5;
G90 G00 X-1.500 Y8.062;
M36 D3 E5;
G90 G00 X-5.000 Y2.000;
M36 D3 E5;
/* Circle Pattern End */

→ Transferred Point
→ Drilling
→ Transferred Point
→ Circle Pattern End

4.4.7. SUB-PROGRAM

This mode provides the user call sub-program repeatedly.

In the dialog edition frame, press **F6 Sub Program**, it gives out the sub-program dialog box as figure 4-4-26, P field is the program name, and L field is the executing times. The Code is M98.

The name of sub-program is limited to started with "O", and the following name is 4 numbers, for example O0001,O0011,O0111.... In addition to that the sub-program is end with M99. If the sub-program is O0001, please key in 0001 in P field as figure 4-4-26.



Figure 4-4-26 sub-program dialog frame

Note 1 : The name of sub-program is limited to started with "O", and the following name is 4 numbers, for example O0001,O0011,O0111.... In addition to that the sub-program is end with M99.

Note 2 : When setting the P field, please ignore the "O", and key in the following 4 numbers. For example, calling O0001 sub-program, ignore the "O", enter "0001" in P field.

The following is a example that a program calls sub-program O0001. Figure 4-4-27 show the diagram of executing the program.

Main Program :

M03;	→ Spindle CW
M08;	→ Fluid ON
G92 X0.000 Y0.000 Z0.000 W0.000;	→ Set the Work Coordinate
G00 X-15.000;	→ Absolute Position to X-15.000
M98 P0001 L5;	→ Call Sub-Program O0001 5 Tmes
M02;	→ Program End

Sub-Program O0001 :

```
G91 G00 X0.000 Y0.000;  
M36 D5 E5;  
G91 G00 X2.000 Y0.000;  
M36 D5 E5;  
G91 G00 X-2.000 Y2.000;  
M36 D5 E5;  
G91 G00 X2.000 Y0.000;  
M36 D5 E5;  
G00 X5.000 Y-2.000;  
M99;
```

```
→ Relative Position  
→ Drilling  
→ Relative Position  
→ Sub-Program End
```



Figure 4-4-27 the result of calling sub-program

4.5. CONDITION TABLE

The condition table can save multiple sets of mach. parameters. When machining different materials, the mach. cond parameters of different groups are loaded. This can save an operator's setting time. The other method is to save the cond tables with different materials as individual file (ext. file name is .CON). When machining the different materials, load individual cond table file. The purpose of the cond table mode is to allow an operator to edit the environment of cond table mach. param.

In the main menu, press **F4 Cond. Table** to enter the cond table. The display screen is as shown in Figure 4-5-1. The upper part of the screen is the cond table general list which can review the setting of each set of cond values. Each cond table can totally save 100 sets of mach. param conditions. From 000 to 099, each set has 18 mach. parameters which can be set. **Current Cond. File:** in the bottom right of the screen is the file name of the current display cond table. After the system is turned on, the preset loading cond table is the Def. Cond COND.DAT.

The bottommost of the screen displays the mach. cond value param of **Current use condition param.** which is the same as the group no. displayed in the first field **NO.** The param group is one of 100 sets of cond tables. In addition, the cond value group in single mach and multi mach screens is that of **Current use condition param.** When the group or the param. is altered anywhere, save it and the other two places will be altered with it.

10/06/04 11:32:51		Condition Tab.						READY		000:03:56	
SINGLE.MCD		mm									
NO.	Hole	T_ON	T_OFF	I_P1	I_P2	C_P	UGap	U_S	S_F		
000	0.200	12	30	8	3	1	12	2	8		
001	1.000	40	20	2	2	1	12	3	9		
002	0.500	30	10	22	2	1	25	3	8		
003	0.200	5	50	8	2	3	5	2	7		
004	1.000	40	20	16	1	4	15	3	9		
005	0.500	40	10	15	3	1	10	3	8		
006	3.000	50	30	64	3	6	15	3	8		
007	0.300	12	6	20	1	1	15	2	8		
008	2.000	30	20	127	2	8	15	3	8		
009	3.000	60	30	63	3	9	15	3	10		
Current use condition parm.						Current Cond. File: COND.DAT					
NO.	Hole	T_ON	T_OFF	I_P1	I_P2	C_P	UGap	U_S	S_F		
000	0.200	12	30	8	3	1	12	2	8		
Range: [0.1~6.5 mm]							Refer:		0.00	00	
1	2	3	4	5	6	7	8	9	10		
Use	GO TO	Change Page	File				Page Up	Page Down	Exit		

Figure 4-5-1 Condition Tab. Screen

In condition tab., use up, down, left and right key to move to different groups or param. fields. And then input the set value of individual param. by numeric keys in accordance with the hint that the prompt field gives. The cond table has in total 18 mach param., but one page only can display 9 param. Use the left and right keys to move to different param. When the cursor is moved to the ninth param., press right key and switch to the second page to display another 9 set values of param. Or directly press **F3 Change Page** to change between the first and second page.

The Def. Cond is the mach. condition that the system presets. Thus the Def. Cond is further protected. If you would like to modify or save COND.DAT, it is necessary to enter **F6 Def. File**. Press **F1 UnLock**. After making sure unlock, modification or saving is allowed.

As shown in Figure 4-5-1, the cond table edit screen has in total seven function keys:

- ◆ Use: Press **F1 Use**, and reflect the condition param. of the group where the cursor places to **Current use condition param.** field. This means that the group with param. set of **Current use condition param.** be updated to the group of the cond table where the cursor places.
- ◆ GO TO: After pressing **F2 GO TO**, the system opens **Goto No.:** dialog box. After no. input, press **ENTER** or **F1 OK**. The cursor auto moves to the group of the cond table where input no. places.
- ◆ Change Page: The system has total 18 mach. param., but each page can only display 9 condition param. fields. Press **F3 Change Page** to change to different pages.
- ◆ File: An operator can open any condition table file, edit modify and copy, delete condition table in the condition database. After pressing **F4 File**, enter the file mode of condition table. The operating methods of open file, save file, another save file, copy file and delete file are the same as the description in 4.1 File in Chapter 4. The difference is that the operated file type is the condition table file with the ext. file name. CON. In addition, **F6 Def. File** it to load the system preset condition table COND.DAT. Once the system is turned on, it loads the condition table first. If after the system opens the other condition tables and you would like to open he present condition table, just press **F6 Def. File**. The Def. Cond provides the



functions of prevention from being modified and saved. After entering the Def. Cond, press **F1 UnLock**. After confirmation, the motions of modification and saving are allowed. **F7 Print to _A** is printing the current condition table to A:\Cond.txt. This file is text file.

- ◆ Page Up: Press **F8 Page Up** once, and the screen switches to last 10 sets of mach. conditions.
- ◆ Page Down: Press **F9 Page Down** once, and the screen switches to next 10 sets of mach. conditions.
- ◆ Exit: Press **F10 Exit** to exit the condition table mode.



The meaning of each mach. param.:

The following table is the meaning of the condition parameter. The input range and the column of parameter may be different, but the meaning is the same.

param.	meaning	Input value range
Hole	diameter size of electrode	0.01~10.00 mm
T_ON	spark time	0~99 us
T_OFF	stop time	0~99 us
I_P1	mach. peak value electric current 1	00~7 level
I_P2	mach. peak value electric current 2	00~15 level
C_P	mach. capacity	00~99 level currently not used
VGap	gap voltage during machining	00~99 level
V_S	work voltage selection	1~7 level
S_F	SPINDLE rotating speed	0~15 level
M_F	the max. speed setting of spark machining	0~100%
Stab	server sensitive degree setting during machining	00~100%
AUX	auxiliary param. (mach. at fixed speed)	0~1 level
Wa_P	water pressure setting of pump	00~99 level
Wa_Q	water quality setting of pump	00~99 level
W_H	records the mach. depth value of mach. material	0.000 ~999.999
Wear	records the wear value	0.000 ~999.999
W_M	records mach. material	0~99 types of material no.
E_M	records the electrode material	0~7 types of material no.

4.6. DEEP TABLE

A deep table can save multiple sets of deep param. When machining different depths or with different wears, load different sets of deep params. to facilitate an operator in operating, which can save the setting time. The purpose of a deep table is to allow an operator to edit the environment set by the depth from the deep table.

In the main menu, press **F5 Deep Table** to enter the mode. The display screen is as shown in Figure 4-6-1. The upper part of the screen is the general list of deep table that can review the setting of each set of depth value. A deep table can totally save 100 sets of depth and wear values. From 000 to 099, each set can set **Desired Deep**, **Wear** and **Mach. Type**. The bottommost part of the screen displays the group and its param. of **Current use deep param.** The used deep value group must be one of the 100 sets of deep tables. Additionally, the deep param. in the lower part of above-mentioned single mach. screen is the deep value group of **Current use deep param.** When the group or the param. is altered anywhere, save it and the other two places will be altered with it.



Figure 4-6-1 Deep Table Screen

As shown the figure above, the edit screen has in total five function keys:

- ◆ Use: Press **F1 Use**, and the deep param. where the cursor places reflect to **Current use deep param.** field. This means the group of **Current use**



deep param. is updated as the group where the cursor places.

- ◆ Print to _A: After pressing **F7 Print to _A** is printing the current condition table to A:\Deep.txt. This file is text file.
- ◆ Page Up: Press **F8 Page Up** once and the cursor moves upward to the last 10 deep conditions.
- ◆ Page Down: Press **F9 Page Down** once and the cursor moves downward to the next 10 deep conditions.
- ◆ Exit: Press **F10 Exit** to exit the deep table mode.

The meaning of each mach. deep param.:

param.	meaning	Input value range
Desired Deep	sets the desired real depth	0.000 ~999.999 -1: not use the deep setting
Wear	records wear value	0.000 ~999.999
Mach. Type	sets the mach. for blind hole or through hole	0: Blind hole; 1: Through hole

Caution: If the mach. type is blind hole, the total deep is “Desired Deep ” + “Wear”; if the mach. type is through hole, the total deep is ”Desired Deep”+ “Wear” + ”Through Hole Compensate”. The through hole compensate is set in the system param. of the custom param.. Please see 9.2 Custom param. in Chapter 4

4.7. MULTI HOLE MACHINE

The main function of this mode is to allow an operator to finish editing mach. prog in prog edit mode, and then enter multi hole mach. mode to execute the mach. prog.

The mode provides a real-time route display. Therefore, when entering the multi hole mode, it is necessary to set the start of the real-time route display and the display range. Thus press **F7 Multi Mach.** in the main menu and the system enters the setting window for the real-time route display range. The screen is as shown is Figure 4-7-1, total four param. **Range of X**, **Range of Y**, **Range of Z** and **Figure Setting Mode**. The description of each param. is as follows:

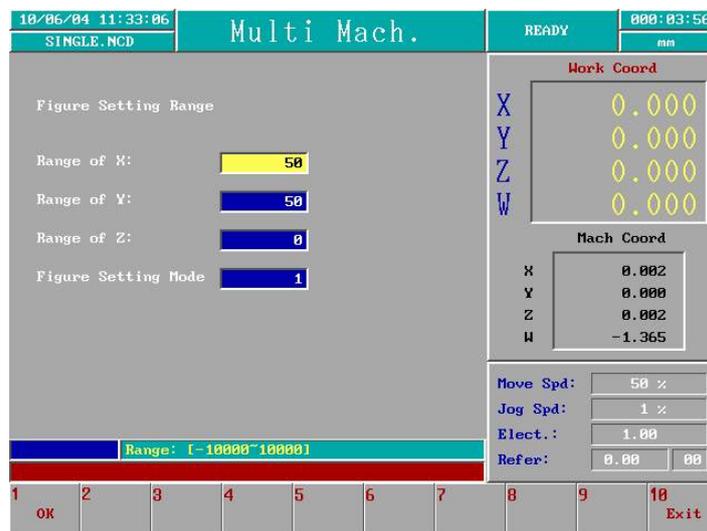


Figure 4-7-1 Multi Mach. Real-Time Display Range Setting Screen

Range of X, Y, Z:

These three fields respectively stand for the range of X, Y, and X that the real-time route displays. When the mach. route exceeds the display range, the route is not displayed.

Figure Setting Mode:

To set whether to start the real-time route display and the display mode, it can be set 0, 1 or 2. The description of the mode that each setting value stands for is as follows:



- (1) When it is set 0: This means that the real-time route is not displayed so after pressing **F1 OK**, enter mutli mach. window. The screen is as shown in Figure 4-7-2. It only has mach. prog display area with black background and no route display area. The prog block marked in yellow within the prog display area is the block that the system is currently executing.
- (2) When it is set 1: This means that half of the window is for displaying 2D real-time route. Part of the screen is as shown in Figure 4-7-3. The left half side with back background is to display the mach. prog, and the right half side is to display the real-time route of XY plane. The white real line is a route that the electrode has passed through. The yellow small box is the fine holes that are currently machined. The read small box stands for the machined fine holes.
- (3) When it is set 2: This means that most of the window is to display 3D real-time route and the mach. progress of each single hole. Part of the screen is as shown in Figure 4-7-4. Only the bottommost black background is to display the mach. prog. The other upper part with gray background is to display the real-time route and single mach. progress. The mach. progress of the machining single hole is displayed on the left side of the area. The display method is the same as that of the mach. progress of single mach, displayed in a bar chart with black background and red foreground. In the right side of the area is the real-time route display area, displayed in 3D way.

Once the system is turned on, the mach. prog is preset to automatically load TEMP file. When operating, edit the mach. prog file in edit mode first and then go to multi mach. mode to load the prog for starting the mach.

10/06/04 11:34:09		Multi Mach.				Machining		000:04:13																									
GENE																																	
Hole Total:0009		Mach Hole No:0003		Line NO: 0009																													
<pre> M03; M00; G92 X0.000 Y0.000 Z0.000 W0.000; G90; G00 X0.000 Y0.000; M36 D0 E0; G00 X10.000 Y0.000; M36 D0 E0; G00 X10.000 Y10.000; M36 D0 E0; G00 X0.000 Y10.000; M36 D0 E0; G00 X-10.000 Y10.000; M36 D0 E0; </pre>																																	
NO.		Desired Deep		Near		Mach. Type																											
D00		5.000		0.000		Blind																											
<table border="1"> <thead> <tr> <th>NO.</th> <th>Hole</th> <th>T_ON</th> <th>T_OFF</th> <th>I_P1</th> <th>I_P2</th> <th>C_P</th> </tr> </thead> <tbody> <tr> <td>000</td> <td>0.200</td> <td>12</td> <td>30</td> <td>0</td> <td>3</td> <td>1</td> </tr> </tbody> </table>										NO.	Hole	T_ON	T_OFF	I_P1	I_P2	C_P	000	0.200	12	30	0	3	1										
NO.	Hole	T_ON	T_OFF	I_P1	I_P2	C_P																											
000	0.200	12	30	0	3	1																											
Deep : [-1, 0~999.999]																																	
<table border="1"> <tr> <td>1 Spark</td> <td>2 Pause</td> <td>3 Reset</td> <td>4 To StarPt</td> <td>5 To BreakPt</td> <td>6</td> <td>7 Save Param</td> <td>8</td> <td>9</td> <td>10 Return</td> </tr> </table>										1 Spark	2 Pause	3 Reset	4 To StarPt	5 To BreakPt	6	7 Save Param	8	9	10 Return														
1 Spark	2 Pause	3 Reset	4 To StarPt	5 To BreakPt	6	7 Save Param	8	9	10 Return																								
						<table border="1"> <tr> <td colspan="4">Work Coord</td> </tr> <tr> <td>X</td> <td>10.000</td> <td>Y</td> <td>10.000</td> </tr> <tr> <td>Z</td> <td>0.000</td> <td>W</td> <td>-4.180</td> </tr> <tr> <td colspan="4">Mach Coord</td> </tr> <tr> <td>X</td> <td>10.002</td> <td>Y</td> <td>10.000</td> </tr> <tr> <td>Z</td> <td>0.002</td> <td>W</td> <td>-5.622</td> </tr> </table>				Work Coord				X	10.000	Y	10.000	Z	0.000	W	-4.180	Mach Coord				X	10.002	Y	10.000	Z	0.002	W	-5.622
Work Coord																																	
X	10.000	Y	10.000																														
Z	0.000	W	-4.180																														
Mach Coord																																	
X	10.002	Y	10.000																														
Z	0.002	W	-5.622																														
						Move Spd: 50 % Jog Spd: 1 % Elect.: 1.00 Refer: 0.00 00																											

Figure 4-7-2 Multi Mach. Screen of Mode 0

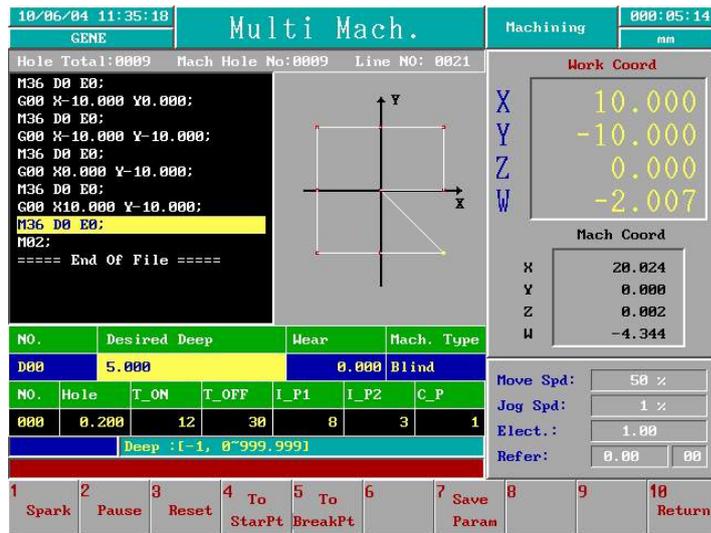


Figure 4-7-3 Multi Mach. Screen of Mode 1

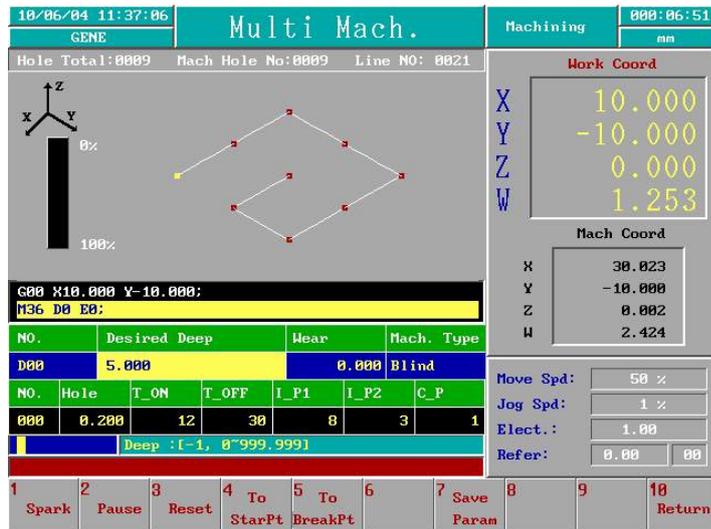


Figure 4-7-4 Multi Mach. Part Screen of Mode 2

In multi mach screen, **Hole Total** displays the total holes required by the mach. prog. **Mach Hole No.** is the machining hole no. However, for the mach prog edited by rect pattern and circle pattern, the system cannot calculate the total holes. The lower part of the screen displays the currently used cond value and deep value param. The first two rows are **Current use deep param.**, and the last two rows are **Current use cond param.** The **NO.** file of deep param. is the group no. of the currently loaded deep table and the **NO.** field of cond param. is the group no. of the currently loaded cond table. Regarding the description of cond table, please see 4.5. Condition Table and

4.6. Deep Table in Chapter4

In multi mach. mode, it is the same as the single mach that can directly modify and save the setting of the currently used cond table and deep table. But if during the machining, the depth setting cannot be altered. In addition, if the current cond table file is COND.DAT, the cond setting cannot be saved to COND.DAT. Regarding the modification of the Def. Cond COND.DAT, please see 5. Condition Table in Chapter 4.

When the soft-limit function starts but the home search motion is not started yet, the function key **Mutli Mach.** will lose its effectiveness and cannot enter the multi mach. mode. At this time, as long as the home search motion is completed or the software function is canceled, the function key of entering multi mach. mode operates normally.

Operating instructions:

1. Load mach. prog: Use **F1 Load File** to load the prog file that is edited completely.

Caution: In prog edit mode, the prog file loaded for editing cannot be opened and loaded again in the multi mach. mode. It is necessary to go to the prog edit mode to load the other files and then enter the multi mach. mode to open the file.

2. Set Mach. Deep: Press **F4 Deep Select** to set **Current use deep param.** Regarding the setting method, please refer to 2.6 Single Mach. Operating Step 1. in Chapter 4. Save the param. at last.

3. Set Mach. Cond.: Set **Current use cond param.** Regarding the setting method, please refer to 2.6 Single Mach. Operating Step 2. in Chapter 4. Save the param. at last. But if the current condition table file is the Def. Cond COND.DAT, saving is not allowed.

4. Mach. Set: Press **F3 Mach. Set** to generate the window of **Program Start Setting**. This can limit the system in only executing the certain blokc in the prog, including setting **Program run START no.** and **Program run End no.**

5. Clear Mach. Time: Press **F6 Clear M. Time** to reset the total mach. time as

0.

6. Start Mach.: Press **F5 Start Mach.** to unfold the function key of **Start Mach.** The function key includes spark, pause, reset, save param., etc. The respective description is as follows:

- ◆ Spark: The function is to start the mach. prog or restore to continue executing the mach. prog which stays in **Pause** status, to start the spark for machining the workpiece. Press **F1 Spark** to start the function. At this time, the status field displays **Spark**.
- ◆ Pause: The purpose of the function is to pause executing the workpiece prog. Press **F3 Reset** to start the function. At this time, the mach. status field displays **Pause**. When the function is started, the system pauses executing the workpiece prog until the function key **F1 Spar** is pressed.
- ◆ Reset: The purpose of the function is to allow an operator to stop executing the mach. prog. and exit the mach. sub mode. Press **F3 Reset** to start the function. At this time, the mach. status field displays **Reset**.
- ◆ Save Param.: Press **F7 Save Param.** to start the function. The purpose of the function is to allow an operator to update and save the param. of the currently used cond table and deep table respectively to the cond table and deep table files. If the cond table file is the Def. Cond COND.DAT, saving is not allowed.

7. Modify Param.: During the process of spark mach., the spark param. of the condition table can be altered at any time, but the mach. deep setting of the deep table cannot be altered. Press **F7 Save Param.** to save the cond value and deep value of the group. If the cond table file is COND.DAT, the cond table cannot be saved.

Assuming the machining has not arrived at the machining deepness in drilling, but the operator does not want to machine this hole. Operator can press **F6 Hole OK**, and it will move the electrode upward to safe position above the workpiece and continue to execute the next block of the program.

4.7.1. SET

This function set to execute some blocks. For example, there is a program with 10 blocks. If it is only to run the block 3 to block 7, please use this function. If the all blocks of program are executed, don't run this function. The default setting is the all block are executed.

Press **F3 Set** in the Multi Hole frame and the system enters the setting window for blocks setting frame. The executing frame is as figure 4-7-5 after setting the executed blocks.

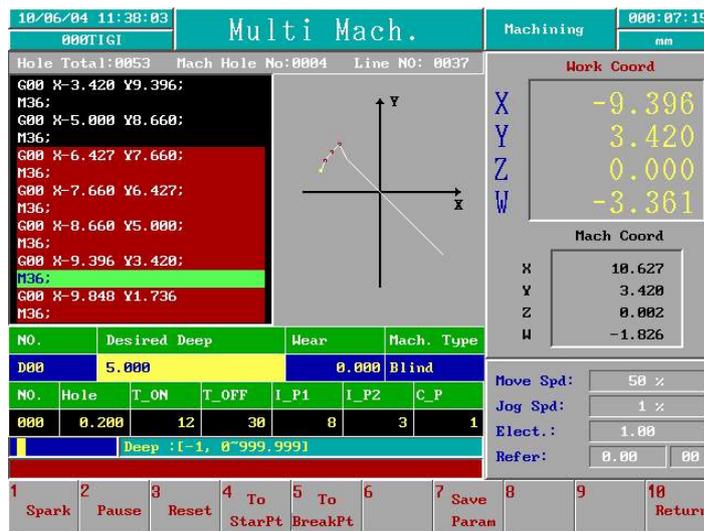


Figure 4-7-5 the executing frame of blocks set

After setting the block set, system is going to mark the blocks with red background color. It remind user of the executed blocks. If the block set is canceled, the marked red background is returned to black and all the blocks is going to execute.

As shows in figure 4.7.5., there are two function keys in this mode. :

- ◆ **Set :** Press **F1 Set**, and give out the block set window. In this window, there are two fields. The first is the **Program run START no.**, the second is **Program run END no.**. After finishing setting, the program is going to be executed from Start Block NO. to End Block NO. User must notice that the Start Block NO is set to be large than the End Block NO.
- ◆ **Clear Set :** Press **F2 Clear Set**, the marked red background is returned to black and all the blocks is going to execute.

4.7.2. Start Point

In Machining, if the program is reset, it is powerful function returning to the start point and executing the program from the first block. It can be done after resetting the program, rebooting up the system and homing. But it need to load the same program, before executing the Start Point.

In multi-hole frame, press **F5 Start Mach.**, then press **F4 To StartPt**, the frame is showing as figure 4-7-6. It is executing To Start Point function. It is noticed that **F4 To StartPt** is gray and inactive before finishing homing. After homing, it is normal and active.

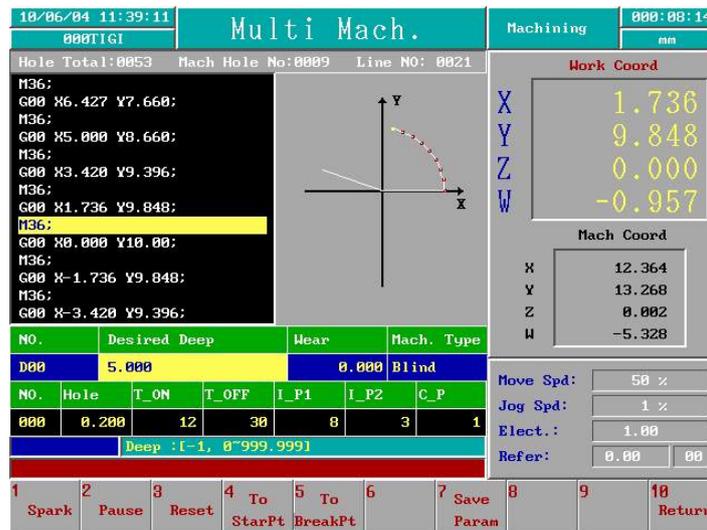


Figure 4-7-6 To Start Point frame

Operating instructions:

1. reset the program : In machining of multi-hole mode, reset the program. If it isn't to be machining anymore, shutdown the controller.
2. reboot up the controller : It is necessary to reboot up the controller and do homing, if switching off it previously.
3. load the program : load the program that is reset previously in multi-hole mode.
4. Press **F4 To StartPt** : in multi-hole mode, press **F5 Start Mach.**, then press **F4 To StartPt**. It is going to start the To Start Point function. It shows the Move W&Z to safe pos., press BasePoint! in hint box.
5. JOG the Z & W axis : It shows the Move W&Z to safe pos., press BasePoint! in hint box. User is noticed that the electrode is going to move to the original position, after pressing the BasePoint key. In order to avoid



the crash, please raise the W and Z axis to safety height, then press **BasePoint** key.

6. Press **BasePoint** key : Press **BasePoint** key, the electrode is going to move to the original position and It shows **Moving to initial point...** in hint box. After moving. It shows **Move W&Z to mach pos., press BasePoint!** ◦
7. JOG the Z & W axis : Because Z and W axis have be raised to safety height, it is necessary to drop off the Z and W axis and to make the electrode is above the workpiece. Then Press **BasePoint** key.
8. Press **BasePoint** key : Press **BasePoint** key, then it shows **Please press F1 to start machinig!** in hint box.
9. Press **F1 Spark** : Press **F1 Spark**, then the program is going to be executed from the first block.

4.7.3. Break Point

In Machining, if the program is reset, it is powerful function returning to the Break Point, status and executing the program from the reset block. For example, when executing the 21th blocks and the work coordinate is (1.736,9.848), the program is reset. After doing the To BreakPt function, the system can return to the work coordinate (1.736,9.848) and execute it from 21th block as figure 4-7-7. It can be done after resetting the program, rebooting up the system and homing. But it need to load the same program, before executing the Break Point.

In multi-hole frame, press **F5 Start Mach.**, then press **F5 To BreakPt**, the frame is showing as figure 4-7-7. It is executing To Break Point function. It is noticed that **F5 To BreakPt** is gray and inactive before finishing homing. After homing, it is normal and active.

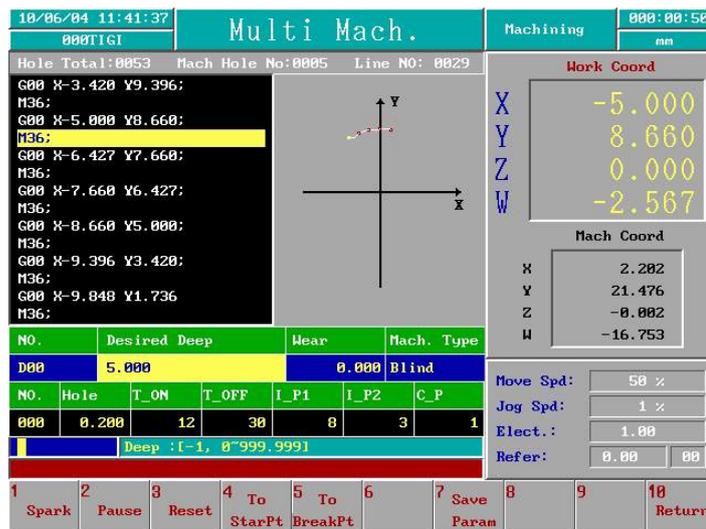


Figure 4-7-7 To Break Point frame

Operating instructions:

1. reset the program : In machining of multi-hole mode, reset the program. If it isn't to be machining anymore, shutdown the controller.
2. reboot up the controller : It is necessary to reboot up the controller and do homing, if switching off it previously.
3. load the program : load the program that is reset previously in multi-hole mode.



4. Press **F5 To BreakPt** : in multi-hole mode, press **F5 Start Mach.**, then press **F To BreakPt**. It is going to start the To Break Point function. It shows the **Move W&Z to safe pos.**, press **BasePoint!** in hint box.
5. JOG the Z & W axis : It shows the **Move W&Z to safe pos.**, press **BasePoint!** in hint box. User is noticed that the electrode is going to move to the reset position, after pressing the **BasePoint** key. In order to avoid the crash, please raise the W and Z axis to safety height, then press **BasePoint** key.
6. Press **BasePoint** key : Press **BasePoint** key, the electrode is going to move to the reset position and It shows **Moving to breakl point...** in hint box. After moving. It shows **Move W&Z to mach pos.**, press **BasePoint!** ◦
7. JOG the Z & W axis : Because Z and W axis have be raised to safety height, it is necessary to drop off the Z and W axis and to make the electrode is above the workpiece. Then Press **BasePoint** key.
8. Press **BasePoint** key : Press **BasePoint** key, then it shows **Please press F1 to start machinig!** in hint box.
9. Press **F1 Spark** : Press **F1 Spark**, then the program is going to be executed from the reset block.

4.8. SYSTEM PARAMETER

The purpose of this mode is to allow an operator to plan the specification and feature of the system, to set the custom param. and to display the relevant functions to the maintenance and diagnosis information.

The mode includes the sub modes: MCS, Custom, Alarm List, Global Data, Command Data, PITCH, AGC, Para Backup and MCL. MLC is display the status of MLC, include IO, C, S, A, Timer, Counter and Ladder. Only MSC, Custom, PITCH and AGC can be modified. The remaining only provides the display function and cannot be modified. If you would like to modify MSC, PITCH or AGC Parameter, it is necessary to pass the password confirmation and then the parameter can be set. This is to avoid the param. of the system feature being modified casually.

Operating method:

Sub Mode of MCS, Custom, PITCH and AGC: Use the up, down, left or right keys to move the cursor, and key in the number data, then press **ENTER** key to accept the input data and update it to the corresponding work field. But in MCS, PITCH and AGC sub mode, it is necessary to pass the password confirmation and then the parameter can be set.

Other Sub Modes (apart from password mode): Use the up and down keys to switch to each screen of the mode.

4.8.1. MCS Parameter

This mode is to allow an operator to check or set the feature constant of the system machine and server. Under the system param., use the function key **F1 MCS** to enter the mode. At this time, the system mode filed displays **System Param.** The operating screen is as shown in Figure 4-8-1.

General operators can check the system param. setting, but only the users who are authorized for modification (needs to pass the password confirmation) can modify the system param. of MCS.

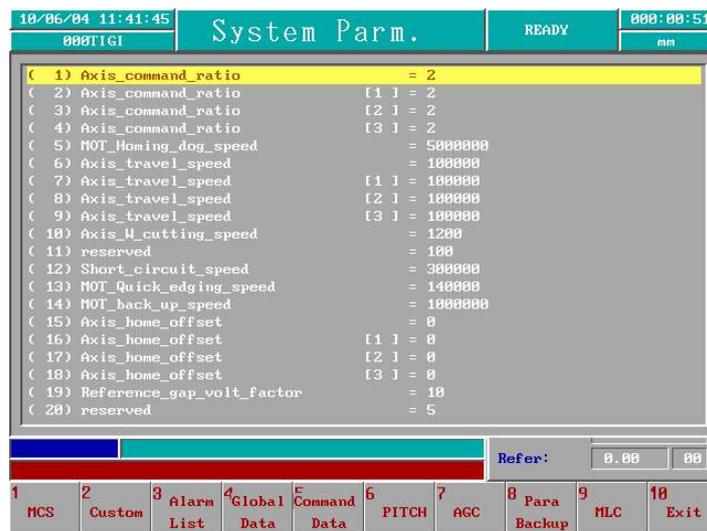


Figure 4-8-1 MCS parameter Screen

Function key explanation :

1. Save file : press **F1 Save file**, save MCS parameter to MCS.DAT. This file is binary file. It is dangerous to delete it.
2. Print to _A : press **F2 Print to _A**, save MCS parameter to MCS.TXT. This file is txt file.
3. MCS modified : press **F3 MCS modified**, It can work, no need to reboot up.

4.8.2. Custom Parameter

This mode is to allow an operator to set the custom param..

Under the system param. mode, use the key function **F2 Custom param.** to enter the mode. At this time, the system status field displays **Custom Param.** Figure 4-8-2 below is customer param. setting screen.

There are two custom files according to inch or mm unit. It is Custom.dat for mm unit ; It is Custom.inh for inch. When booting up, It can load that file according to mm or inch unit.



Figure 4-8-2 Custom Parameter Screen

Function key explanation :

1. Save file : press **F1 Save file**, save Custom parameter to CUSTOM.DAT or CUSTOM.INH. This file is binary file. It is dangerous to delete it.
2. Print to_A : press **F2 Print to_A**, save Custom parameter to CUSTOM.TXT. This file is txt file.
3. Set Time : press **F3 Set Time**, key in the setting time, and press **ENTER**. The system time is going to change to your setting time.
3. Set Date : press **F4 Set Date**, key in the setting date, and press **ENTER**. The system date is going to change to your setting date.

The meaning of each param.:

(1) Unit:

The unit is for setting the system coord. There are two types: inch and

mm. Setting 0 mean mm unit, it load Custom.dat and refresh this frame; Setting 1 mean inch unit, it load Custom.inh and refresh this frame. After setting the new unit, Please restart the computer for the newly set unit to come into effect.

(2) Lang.:

The language mode that the system displays. At present, it only supports Chinese and English mode. Setting 0 mean the Chinese mode ; Setting 1 mean the English mode. The newly set language will come into effect when the computer starts next time.

(3) Software Protect:

When it is set ON and the mach. prog is executing, the system checks whether the mach. route enters the forbidden distance zone of the machine coord (please see the distance limit of each spindle below). If yes, it sends out the error message **Warning! Soft Limit Switch +X ON!**, stops the mach and enters the reset status. If not, execute the mach. normally. When the system is started with function set ON, it reminds a user to start the home search motion to make sure that the function of soft-limit can operate normally. Or set the function OFF. When it is set OFF, the system cannot provide the function of soft-limit protect.

(4) X+, X-, Y+, Y-, Z+, Z-,W+,W- distance limit:

This is in order to define the moveable area of each spindle. The other zones not allowed to enter are called “forbidden zone”. The setting needs to coordinate with that the param. setting of **Software Protect** is set Yes. Figure 4-8-3 below is for simple illustration. The orange zone is the range that the cutter can move.

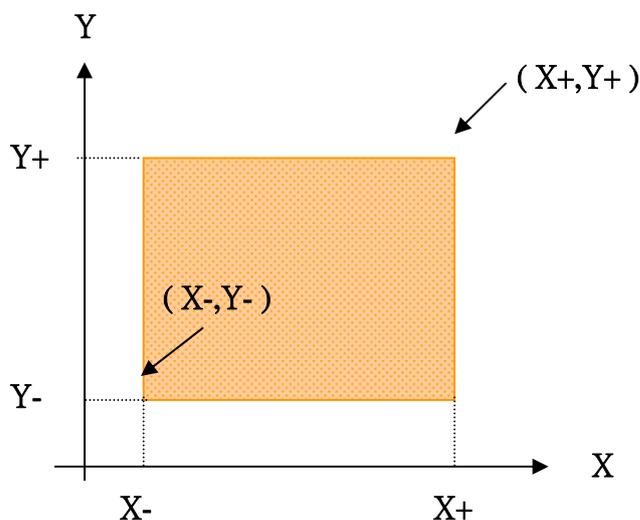


Figure 4-8-3 Illustration of the Range That the Cutter Can Move

For instance, set $X+ = 200, Y+ = 300, X- = -200$ and $Y- = -300$. When machining, if the movement range of the work coord in X is more than 200 or less than -200 , the system will display the error message. The same reason, if the move range of the work coord in Y is more than 300 or less than -300 , the system will also display the error message.

(5) Reserve

(6) Safety Height:

This is for the time when the system completes one hole spark and mach. It refers to the height being increased after the electrode moves out the workpiece surface. Please set positive value, and the input unit is mm or inch.

(7) Through Hole Compensate:

When the mach. type is though hole, the total mach. depth is "Desired Depth"+"Wear"+"Through Hole Compensate". The purpose is to make sure the completion of through hole motion. Please set positive value, and the input unit is mm or inch.

(8) Distance back to edge:

This is for the time when the sytem executes edge searching. It refers to the distance that moves backward when the electrode contacts the workpiece. Please set positive value, and the input unit is mm or inch.

(9) Short Reset Height in Mach.

If the gap voltage is low than the setting voltage in hole machining, the electrode is going to back. If the back position in machining reach the setting height, it is going to reset and give out the alarm message, **ALARM! MOT 006:Short In MACHINING!**. How to calculate the height? Count up From the surface of starting machining point. For example 5mm, When electrode back to 5mm above the upper surface of workpiecem, it is going to reset and give out the alarm message. This parameter need to set more than **(8) Distance back to edge**. Please set positive value, and the input unit is mm or inch.

(10) Check Distance:

The param. only needs being set when the electrode wear is measured. Place the chunk rule on the workpiece. When the electrode wear is measured, move the electrode to under the chunk rule and then make the electrode search the upper side of the chunk rule to get the coord of the chunk



rule. The electrode wear value=chunk rule coord – the coord that touches the workpiece surface when starting the mach. – chunk rule thickness. The thickness of the chunk rule is touch distance.

(11) Touch pt Enable:

The param. is to set whether to change the setting of the electrode length touch pt. When the param. is 0, it means machining. When the electrode is run out and needs being replaced, the setting of the electrode touch pt. needs to use manual method to move each spindle to touch pt. by pressing JOG key of each spindle. When the param. is 1 and the mach. prog is no G06 setting, the touch pt is (Touch pt X, Touch pt Y, Touch pt Z). (the description of Touch pt X, Y, Z, please refer to (11) as below). In addition, when the mach. prog has the setting of G06, the touch pt. coord is that of the prog single block G06. Regarding more detail about the touch pt. setting of replacing the electrode, please refer to 1. Machining Manually Replacement for the Electrode Step 1 is in Chapter 5.

(12) Touch pt X, Y, Z:

When the param of Touch pt Enable is 1 and the mach. prog does not have G06 setting, the three params. can function. The setting allows a user to preset the touch pt coord of touch electrode length when the electrode is being replaced and machining. When machining, if the electrode runs out and needs being replaced, the electrode needs to be moved to the touch pt. to touch the electrode length. The X, Y, Z coord value of the touch pt. are respectively the X, Y, Z param. of touch pt.

(13) In Move, Z home:

It move to check point or machining point in changing electrode, running the initial point or the break point. It move Z axis to home, then move XY axis. It is avoiding the crash of electrode and workpiece. Setting 1 means that it move Z axis to home at first; Setting 0 means that Z axis is hold and move X Y axis, then move Z axis to goal point.

(14) Min. Unit::

This is setting the meaning of instruction without decimal point. Setting 1 means that the unit is mm or inch.; Setting 0 means 0.001mm or 0.0001inch. For example, the instruction is G00 X10.; If setting 1, it move to X10.0; Setting 0, it move to X0.01.

(15) ATC Enable::



If the machine is 4-axis closed loop controller with ATC instrument. This parameter is set to switch on auto-change-electrode. Setting 1 means it is auto-change; Setting 0 means manual-change.

Note : If it is 3-axis close loop controller, Please set to 0.

(16) Dist. of Low Sp. :

It is not stable in starting of machining. Slowing down is better for the machining efficiency. It can set to slow down in starting machining. After reaching the setting deepness, it is speed up. This parameter is setting the deepness. The calculating method is distance counting down from the upper surface. (17) Percent. of Low Sp. is the setting of slowing down. For example, setting 2mm means it is machining with low speed except reach the 2mm position. Please set positive value, and the input unit is mm or inch.

(17) Percent. of Low Sp. :

Reference to (16) **Dist. of Low Sp.**, this is setting the percent of slowing down. Setting 50 means the low speed is 50 percent of the original speed. Please set positive value.

(18) Dist. Down to Edge(CheckPt):

It need to move the check point to measure the length after changing electrode in hole machining. But because the electrode is above the guide, it expend a lot of time to do edge procedure. In order to save time, before edging, let electrode move down a distance. Setting negative value means moving down; Positive value means moving up. The input unit is mm or inch.

4.8.3. Alarm List

This mode is to display the system alarms and warnings. Through the display of the alarms and warnings, the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F3 Alarm List** to enter the mode. At this time, the system mode field displays **Alarm List**. Figure 4-8-4 below is alarms list display screen. Use the up and down keys to change the page.

It can save as a file, press **F1 Print to_A** save to A:\Errlist.txt as a txt file.

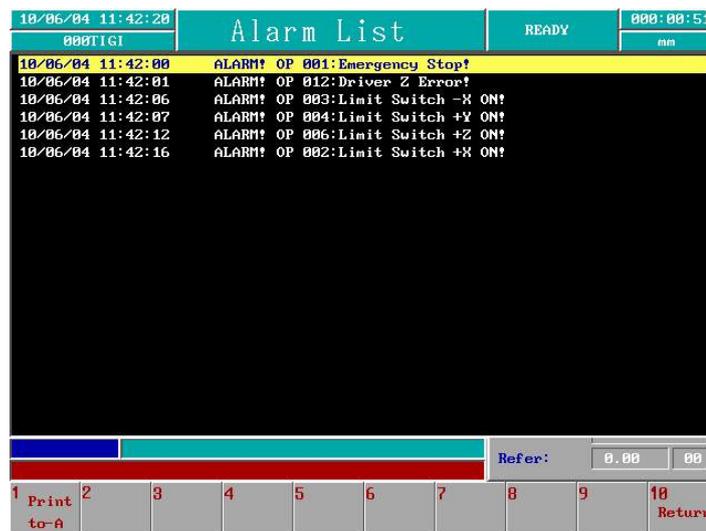


Figure 4-8-4 the frame of alarm list

Function key explanation :

1. Print to_A : press **F1 Print to_A**, save Alarm to Errlist.txt. This file is txt file.

4.8.4. Global Data

This mode is to display the system param. of Global Data (System Variable). Through the display of the system variable param., the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F4 Global Data** to enter the mode. At this time, the system mode field displays **Global Val.** Figure 4-8-5 below is Global Data display screen. Use the up and down keys to change the page. The left side of the equal sign is the param. name, and the right side of the equal sign is the current value of the param.

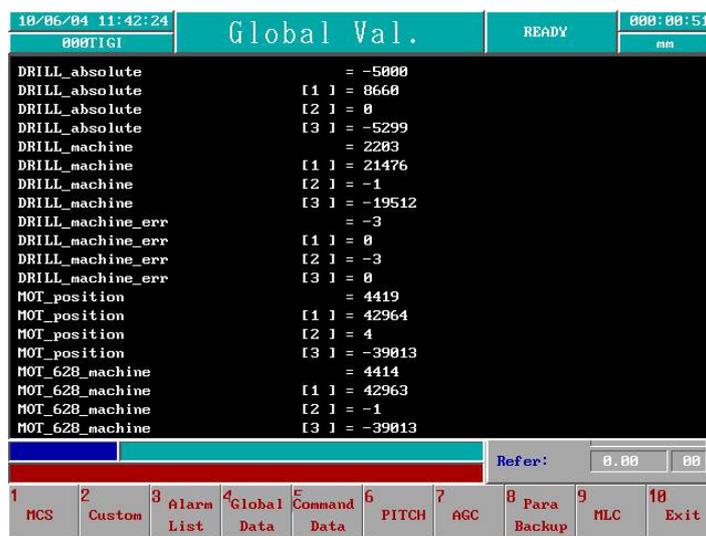


Figure 4-8-5 Global Val. Screen

4.8.5. Command Data:

This mode is to display the situation of the system command param. Through the display of the system Command Data, the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F5 Command Data** to enter the mode. At this time, the system mode field displays **Command Data**. Figure 4-8-6 below is Command Data display screen. Use the up and down keys to change the page.



Figure 4-8-6 Command Data Screen

4.8.6. PITCH Data:

This mode is to display the setting situation of the system pitch error compensation data. Through the display of the system pitch data, the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F6 PITCH** to enter the mode. At this time, the system mode field displays **PITCH DATA**. Figure 4-8-7 below is Pitch Data display screen. Use the up, down, left and right keys to move the cursor. Use page up and page down keys to change the page.

In this mode, it is necessary to pass the password confirmation and then the parameter can be set.

10/06/04 11:42:47		PITCH DATA		READY	000:00:53
000TIGI					mm
1. Comp. Enable	0	21. X Comp. Amount[11]			0
2. Comp. Dir	0	22. X Comp. Amount[12]			0
3. X Comp. Section	50	23. X Comp. Amount[13]			0
4. Y Comp. Section	100	24. X Comp. Amount[14]			0
5. Z Comp. Section	150	25. X Comp. Amount[15]			0
6. H Comp. Section	50	26. X Comp. Amount[16]			0
7. Reserve	0	27. X Comp. Amount[17]			0
8. Reserve	0	28. X Comp. Amount[18]			0
9. Reserve	0	29. X Comp. Amount[19]			0
10. Reserve	0	30. X Comp. Amount[20]			0
11. X Comp. Amount[11]	10000	31. X Comp. Amount[21]			0
12. X Comp. Amount[21]	10000	32. X Comp. Amount[22]			0
13. X Comp. Amount[31]	10000	33. X Comp. Amount[23]			0
14. X Comp. Amount[41]	10000	34. X Comp. Amount[24]			0
15. X Comp. Amount[51]	5	35. X Comp. Amount[25]			0
16. X Comp. Amount[61]	6	36. X Comp. Amount[26]			0
17. X Comp. Amount[71]	7	37. X Comp. Amount[27]			0
18. X Comp. Amount[81]	8	38. X Comp. Amount[28]			0
19. X Comp. Amount[91]	9	39. X Comp. Amount[29]			0
20. X Comp. Amount[101]	0	40. X Comp. Amount[30]			0

				Refer:	0.00	00
1 Save File	2 Print to-A	3 MCS Modify	4	5	6	7
			8	9	10 Return	

Figure 4-8-7 Pitch Data Screen

Function key explanation :

1. Save File : press **F1 Save File**, save pitch parameter to Pitch.bin. This file is binary file. It is dangerous to delete it.
2. Print to_A : press **F2 Print to_A**, save pitch parameter to Pitch.txt. This file is txt file.
3. Modify : press **F3 Modify**, It can work, no need to reboot up.

4.8.7. AGC:

If there is no AGC instrument, this frame does not display.

This mode is to display the setting situation of the system AGC data. The T code of AGC instruction “M06 T**” stand for real size of guide. The “**” code is the number from 1 to 64. Therefore, this frame is setting the real size of the T code. For example, maybe the T01 means the size 0.9mm electrode, and key in 0.9, press **ENTER**. The following figure 4-8-8 is AGC frame. As the above description, T01 stand for the guide in a size 0.9mm. When executing M06 T01, it is going to changing the size 0.9mm guide.

Under the system param. mode, use the function key **F7 AGC** to AGC mode. At this time, the system mode field displays **ACG**. Figure 4-8-8 below is AGC display screen. Use the up, down, left and right keys to move the cursor.

In this mode, it is necessary to pass the password confirmation and then the parameter can be set.

10/06/04 11:42:53		AGC DATA				READY	000:00:53	
000TIG1							mm	
T	Diameter	T	Diameter	T	Diameter	T	Diameter	
1	0.90	17	No Use	33	No Use	49	No Use	
2	0.80	18	No Use	34	No Use	50	No Use	
3	0.70	19	No Use	35	No Use	51	No Use	
4	0.60	20	No Use	36	No Use	52	No Use	
5	0.50	21	No Use	37	No Use	53	No Use	
6	0.40	22	No Use	38	No Use	54	No Use	
7	0.30	23	No Use	39	No Use	55	No Use	
8	0.20	24	No Use	40	No Use	56	No Use	
9	0.10	25	No Use	41	No Use	57	No Use	
10	6.00	26	No Use	42	No Use	58	No Use	
11	No Use	27	No Use	43	No Use	59	No Use	
12	No Use	28	No Use	44	No Use	60	No Use	
13	No Use	29	No Use	45	No Use	61	No Use	
14	No Use	30	No Use	46	No Use	62	No Use	
15	No Use	31	No Use	47	No Use	63	No Use	
16	No Use	32	No Use	48	No Use	64	No Use	

Range: [0.1~6.5 mm]					Refer:	0.00	00		
1 Save File	2	3	4	5	6	7	8	9	10 Return

Figure 4-8-8 AGC Screen

Function key explanation :

1. Save File : press **F1 Save File**, save pitch parameter to Guidesiz.dat. This file is binary file. It is dangerous to delete it.



4.8.8. MLC

This mode displays the diagnosis and maintenance information. There are I/O bit、C bit、S bit、A bit、Timer、Counter and Ladder sub modes. It display the status of I/O bit、C bit、S bit、A bit、Timer、Counter and Ladder.

Operating Method :

In MLC mode, you can press any function key to inter the sub-mode. If there are more than two pages, you can press Page Up or Page Down to page up or page down.

Introduce the sub mode as following.

4.8.8.2. C BIT

This mode is to display C status of the system. Through the display of system C status, the diagnosis and maintenance of the system can be carried out.

In the system param. mode, use the function key **F2 C** to enter the mode. At this time, the system mode field displays **C**. Figure 4-8-11 below is C status display screen. Use the up and down keys to change the display page. The status display method is the same as that of 4.8.9.1. I/O.



Figure 4-8-11 C Screen

4.8.8.3. S BIT

This mode is to display S status of the system. Through the display of system S status, the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F3 S** to enter the mode. At this time, the system mode field displays **S**. Figure 4-8-12 below is S status display screen. Use the up and down keys to change the display page. The status display method is the same as that of 4.8.9.1. I/O.



Figure 4-8-12 S Screen

4.8.8.4. A BIT

This mode is to display A BIT status of the system. Through the display of system A bit status, the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F4 ABIT** to enter the mode. At this time, the system mode field displays **A**. Figure 4-8-13 below is A bit status display screen. Use the up and down keys to change the display page. The status display method is the same as that of 4.8.9.1. I/O.



Figure 4-8-13 A Bit Screen

4.8.8.5. Timer

This mode is to display Timer status of the system. Through the display of system Timer status, the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F5 Timer** to enter the mode. At this time, the system mode field displays **TIMER**. Figure 4-8-14 below is Timer status display screen. Use the up and down keys to change the display page.

10/06/04 11:43:16		TIMER						READY	000:00:53	
000TIGI									mm	
1 / 2										
NO.	S.Value	N.Value	NO.	S.Value	N.Value	NO.	S.Value	N.Value		
00	5	5	12	2	3	24	0	0		
01	10	10	13	8	9	25	0	0		
02	10	10	14	2	3	26	0	0		
03	8	9	15	8	9	27	0	0		
04	2	3	16	2	3	28	0	0		
05	8	9	17	8	9	29	0	0		
06	2	3	18	2	6	30	20	0		
07	8	9	19	8	11	31	20	0		
08	2	3	20	2	3	32	100	0		
09	8	9	21	0	0	33	0	0		
10	2	3	22	0	0	34	0	0		
11	8	9	23	0	0	35	30	31		

								Refer:	0.00	00
1	2	3	4	5	6	7	8	9	10	
I/O	C	S	A	TIMER	COUNTER			LADDER	Exit	

Figure 4-8-14 Timer Screen

4.8.8.6. Counter

This mode is to display Counter status of the system. Through the display of system Counter status, the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F5 Counter** to enter the mode. At this time, the system mode field displays **COUNTER**. Figure 4-8-15 below is Counter status display screen. Use the up and down keys to change the display page.

10/06/04 11:43:16		TIMER			READY		000:00:53	
000TIGI							mm	
1 / 2								
NO.	S.Value	N.Value	NO.	S.Value	N.Value	NO.	S.Value	N.Value
00	5	5	12	2	3	24	0	0
01	10	10	13	0	9	25	0	0
02	10	10	14	2	3	26	0	0
03	8	9	15	8	9	27	0	0
04	2	3	16	2	3	28	0	0
05	8	9	17	0	9	29	0	0
06	2	3	18	2	6	30	20	0
07	8	9	19	8	11	31	20	0
08	2	3	20	2	3	32	100	0
09	8	9	21	0	0	33	0	0
10	2	3	22	0	0	34	0	0
11	8	9	23	0	0	35	30	31

								Refer:	0.00	00
1	2	3	4	5	6	7	8	9	10	
I/O	C	S	A	TIMER	COUNTER			LADDER	Exit	

Figure 4-8-15 Timer Screen

4.8.8.7. Ladder

This mode is to display Ladder status of the system. Through the display of system Ladder status, the diagnosis and maintenance of the system can be carried out.

Under the system param. mode, use the function key **F9 Ladder** to enter the mode. At this time, the system mode field displays **LADDER**. Figure 4-8-16 below is Timer status display screen. Use the up and down keys to change the display page.

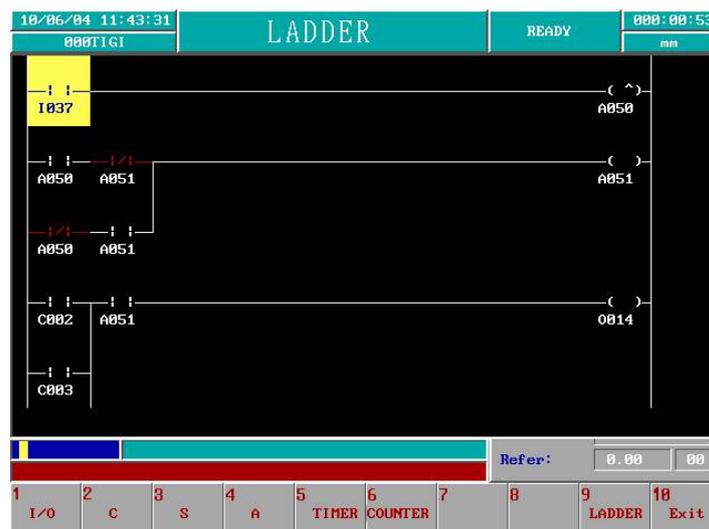


Figure 4-8-16 Timer Screen

4.8.9. Password

This mode is to allow an operator to confirm the function of authorizing password. Through the successful confirmation of authorizing password, MCS system param. can be modified. If the confirmation is wrong, MCS system param. only can be observed. The operating screen of password is as shown in Figure 4-8-17 below.

In the screen, there are 2 function keys for use:

- ◆ OK: After pressing **F1 OK**, it carry out the confirmation whether in the input password by the operator is correct.
- ◆ Cancel: After pressing **F10 Cancel**, exit the password input window.



Figure 4-8-17 Password Screen

4.9. RS232

This mode allows an operator to use RS232 connection port to transmit the mach. prog codes. The system can be the transmission end to send out the prog code to other computers. It also can be the reception terminal to receive the prog codes sent by other computers. The mode is divided into two function keys: **F1 RS232** and **F2 RS232 Port**. The following will introduce respectively.

4.9.1. RS232

This mode is the operating screen of RS232 connection port receiving and transmitting mach. prog codes. The operating screen is as shown in Figure 4-9-1 below.

In RS232 window, press **F1 RS232** to enter the mode. When the system is the transmission terminal, the prog codes are loaded into the window. After pressing **F1 Send**, send out the prog codes; when the system is the reception terminal, the prog codes, through the transmission of RS232, are received one line after another and display in the window. After the reception is completed, press **F2 SaveAs** and save all the prog codes as another file. In addition, press **F3 Clear** to clear the prog codes in the window. Or during the process that the prog codes receive or transmit, press **F4 Reset** to reset the motion. To exit RS232 sub mode, press **ESC** to enter the main menu.



Figure 4-9-1 RS232 Screen

4.9.2. RS232 Port

This mode is to allow an operator to set the transmission protocol of RS232 connection port. The operating screen is as shown in Figure 4-9-2 below.

In RS232 window, press **F1 RS232 Port** to enter the mode. In the setting window, if Mode field is set Terminal, this stands for the reception terminal. If it is set Host, this stands for transmission terminal.

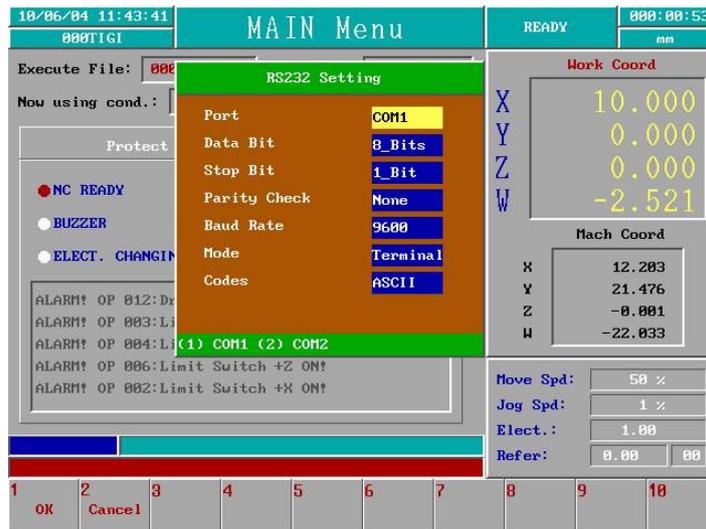


Figure 4-9-2 RS232 Port Screen

Chapter 5 Operating Example Description

The chapter illustrates that during the process of spark mach., when the electrode runs out, how to replace the electrode in a manual method.

Machining manually replace electrode

During the process of spark mach. in single mach. or multi mach., when the electrode runs out and needs being replaced, follow the steps about electrode manual replacement below to replace the electrode

Step One: The purpose of the step is to set the touch pt of the electrode length. In the system, there are three types of setting modes. The respective description is as follows:

Mode 1: When the param. of **Touch pt Enable** in 4.8.2 Custom Param in Chapter 4 is set 0 and the mach prog does not have G06 setting, enter the mode. During the process of spark mach., if the electrode runs out, the machine auto pause and it displays **Move elect. to detect point & press Base Point**. At this time, use JOG button of **X+**, **X-**, **Y+**, **Y-** and **W+** to move the electrode to the position with the same height as that of the mach. StPt and keep the proper height. And then press **Base Point**, and the touch pt coord is the coord of the current electrode

Mode 2: When the param. of **Touch pt Enable** in custom param. is set 1 and the mach. prog does not have G06 setting, enter the mode. At this time, the param. of **Touch pt X**, **Touch pt Y**, **Touch pt Z** in custom param. stands for the coord value of the touch pt. Therefore, during the process of spark mach., if the electrode runs out, the machine automatically pauses, and moves the elect to the coord (Touch pt X, Touch pt Y, Touch pt Z).

Mode 3: When the mach. prog has G06 setting, enter the mode. When the mach. prog has the relevant setting to G06, the setting of **Touch pt X**, **Y**, **Z** in custom param cannot function and the touch pt coord is the set coord for G06. For instance, the mach. prog single block is G06 X1 Y1 Z1; during the process of spark mach., if the electrode runs out, the machine automatically pauses spark mach., and moves the electrode to the coord (1,1,1) to carry out the touch



of the electrode length. Regarding the editing of G06, use the **coord** of **dialog edit** to edit

Step Two: After executing the step, the machine displays **Detecting**. At this time, after the computer records the coord of base point, move W axis to the positive direction limit and displays **W axis is moving to W+ limit!**. Then remove the old electrode and replace with a new one.

Step Three: After executing Step Two, the machine displays **Press FTC button!**. It is recommended to move the W axis inside the Guider and move the electrode to the proper distance near the workpiece first. And then press the **FTC** button again. (shorten the edging time)

Step Four: After executing Step Three, the machine displays **W axis is edging!**. Afterward, it displays **W&Z axis move to safe pos. press Base Point**. Press **Base Point** button.

Step Five: After executing Step Four, **Move W&Z axis to Guider press Base Point**. Press **Base Point** once more time to continue the mach.

Caution One: When single mach., after replacing the electrode, the prog will automatically execute the fluid supply and rotation. In multi mach. mode, if the prog does not add the prog code of M08, M03 (see Table 4-1), after executing Step Four, be sure to press **Fluid Supply** and **Rotate** button.

Caution Two: If the desired workpiece is not flat, after executing Step Four, use the **W+** button to adjust the safety distance.

Chapter 6 Program Code

6.1. G Code Explanation

G00 : Fast Positioning

Format : G00 IP____ ;

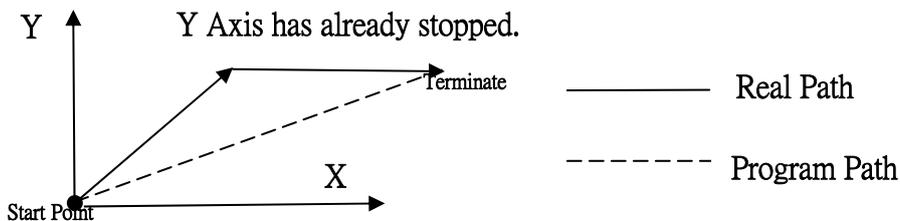
Explanation : IP is identical to X、Y、Z and W the coordinate system values target. As per its absolute values or augment values, it is determined by the status of G90 or G91 (This manual will adopt this interpretation hereafter.). Symbol";" means the ending of single block (This manual will adopt this interpretation hereafter.).

Note:

If using G00, the moving speed for each axis is determined by the parameter system NO.6~9.

Example: G00 X10.0 Y20.0 Z30.0 Z40.0;

Figure:





G04 : Pause

Format : G04 X_____ ;

Explanation :

Pause the auto-machining for seconds. X value is the holding time. When time is up, It is going to run the next block.

Note:

The unit of X value is second. The setting range is 0.001~9999.999 seconds.

Example : G04 X1.0;

Pause for 1 second

G05 : No-guard Fast Positioning

Format : G05 IP_____ ;

Explanation :

IP is identical to X、Y、Z and W the coordinate system values target. As per its absolute values or augment values, it is determined by the status of G90 or G91 (This manual will adopt this interpretation hereafter.).

Note:

1. The largest difference between G00 and G05 is that G05 command will omit the short protection when tools and workpieces colliding (When short happening, the feed-in rate is 0).
2. When using G00, the moving speed for each axis is determined by the setting values of parameter system NO.6~9.

Example : G05 X10.0 Y20.0 Z30.0 Z40.0;



G06 : Set Check Point

Format : G06 X___ Y___ Z___ ;

Explanation :

Set the check point coordinate. The X, Y and Z value is machine coordinate of the check point. When the electrode is used up in machining, it is going to change electrode. At first, it is going to the check point and detect the wear of electrode. °

Example : G06 X10.0 Y20.0 Z30.0;

Set the check point at(10.0, 20.0 30.0)



G30 G31 : block skip setting

Format : G30 ;

G31 ;

G30 : block is not skipped

G31 : block is not skipped

Explanation :

1. "/" is the skip code, but the code works with G30 and G31. G31 switch on the block-skip, and G30 switch off the block-skip. G30 is system default for booting up.
2. When executing G31, the following block with "/" at block beginning is going to be skipped. But after G30, although the block is beginning with "/", it is not skipped.

Example:

```
/* SKIP TEST*/  
G00 X0. Y0.;  
/G00 X10. Y10.; → not skipped in system default  
G31;  
G00 X20. Y20.;  
/G00 X30. Y30.; → skipped  
G30;  
G00 X40. Y40.;  
/G00 X50. Y50.; → not skipped  
....  
M02;
```



G38 : Condition Table Change

Format : G38 P___ ;

Explanation :

The range of P value is number from 0000 to 9999, and the changed condition file name is 0000.con~9999.con.

Note : If the changing condition file name do not exist, system is going to shoot the Alarm message and reset the system. Please save the changing file.

Example: Now, the condition table file is default file, Cond.dat, the changing file is 0002.con ◦

```
/* TABLE CHANGE TEST*/
```

```
G38 P0002;
```

```
G00 X0. Y0. Z0.;
```

```
....
```

```
M02;
```



G53 : Mechanic Coordinate System

Format : G53 IP___ ;

Explanation : When G53 command designated, the coordinate system will change to work coordinate system and move to the designated mechanic position of IP at the speed of G00.

Example: G53 X10.0 Y10.;

Move to machine coordinate(10., 10.)

G80 : Edge Searching

Format : G80 X+1/X-1/Y+1/Y-1/Z+1/Z-1/W+1/W-1 ;

Explanation : Locate X、Y、Z or W axis, on the workpiece's location of positive or negative axial directions.

Example: G80 X+1;

Edge searching in positive direction of X axis.

G81 : Mechanic Origin Searching

Format : G81 IP___ ;

Explanation : Locate X、Y、Z or W axis to the mechanic origin.

Example 1 : G81 X1.0 ;

Search the mechanic Origin of X axis.

Example 2 : G81 X1.0 Y1.0 Z1.0 W1.0 ;

Search the mechanic Origin of X,Y,Z and W axis.

Example 3 : G81 W1.0 ;

Search the mechanic Origin of W axis.



G82 : Locate Groove Center

Format : G82 P* ;

Explanation: P1 is to locate the groove center of X axis.

P2 is to locate the groove center of Y axis.

Example: G82 P1;

Locate the groove center of X axis.

G83 : Locate Inner Hole Center

Format : G83 P* ;

Explanation : P1 is to locate XY plane's inner hole center.

Example: G83 P1;

Search the XY plane's inner hole center.

G84 : Locate Outer Hole Center

Format:G84 P* IP___ ;

Explanation : P1 is to locate XY planes' axial outer center.

Example : G84 P1 X5.0 Z-5.0 ;

Locate XY plane's axial outer center.

G84 P1 Y5.0 Z-5.0 ;

Locate XY plane's Y axial outer center.



G85 : Locate Circular (Square) Cylinder Center

Format :G85 P* IP___ ;

Explanation: P1 is to locate XY plane's circular (square) cylinder center.

Example:G85 P1 X5.0 Y5.0 Z-5.0 ;

Locate XY plane's circular (square) cylinder center.

G90、G91 : Absolute and Augment Coordinate Setting

Format : G90 ;

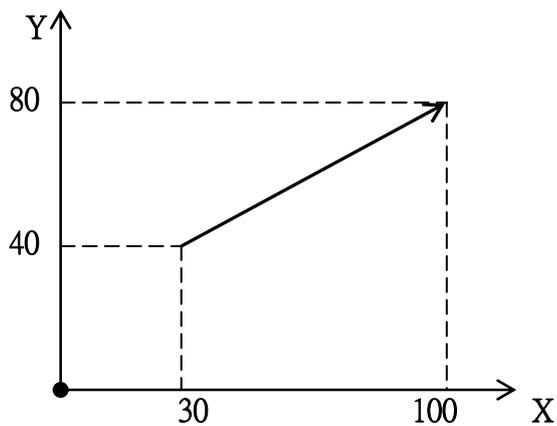
G91 ;

Explanation : G90 is absolute coordinate setting.

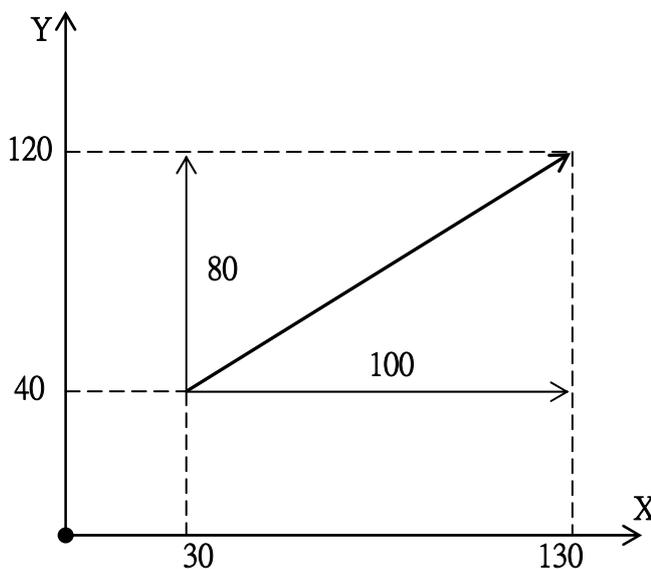
G91 is augment coordinate setting.

Note : G90、G91 can be claimed in the same single block, but take care to recognize the difference caused by its order.

Example : G90 G00 X100.0 Y80.0 ;



G91 G00 X100.0 Y80.0 ;



G92 : Absolute Origin Setting

Format : G92 IP___ ;

Explanation : This command is to set a certain point of absolute coordinate system as the origin of its coordinate system. This new set origin is to include new work coordinate system and treat it as new work coordinate system. After the setting for this new work coordinate system, the absolute coordinate system will be calculated based on this coordinate system.

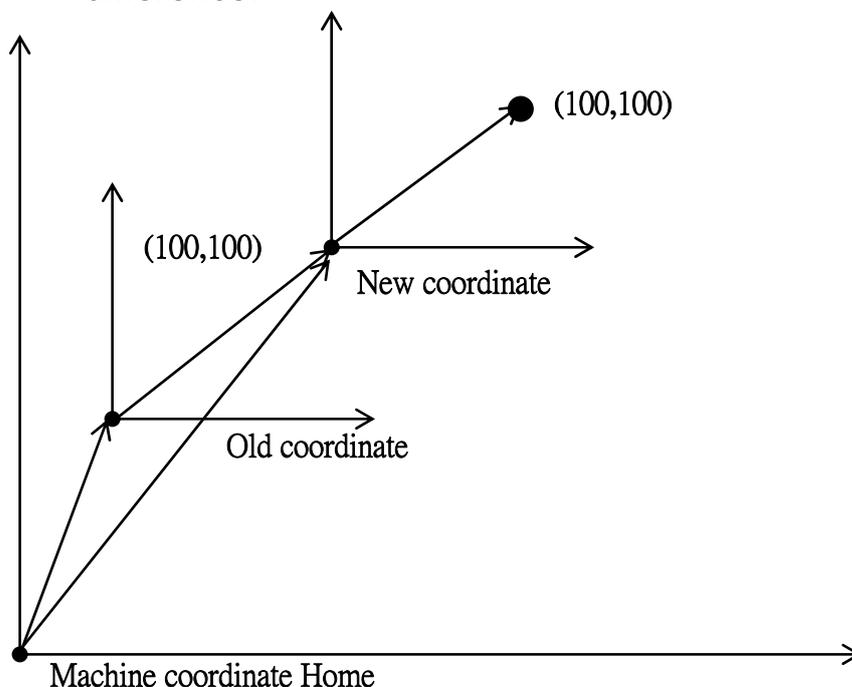
Note : Using G92 command, the work coordinate system of corresponding axis will be changed, and the rest un-designated axis will be kept "as is" for tier previous origin setting.

Example : G00 X100.0 Y100.0;

G92 X0.0 Y0.0 Z0.0 ;

G00 X100.0 Y100.0;

Move to (100,100), and use G92 to set the new work coordinate origins. Reference to the following figure, the coordinate is changed. Then move to (100,100). It is obvious that the machine coordinate is difference.





G Code List

G Code	Function Explanation
G00	Fast Positioning
G04	Hold for Seconds
G05	No Guard Fast Positioning
G06	Check Point Set
G30	Block Skip Cancel
G31	Block Skip Active
G38	Condition Table Change
G53	Machine Coordinate Position
G80	Edge Searching
G81	Mechanic Origin Searching
G82	Locate Groove Center
G83	Locate Inner Hole
G84	Locate Outer Center
G85	Locate Circular Cylinder Center
G90	Absolute Coordinate
G91	Relative Coordinate
G92	Work Coordinate Point Setting

6.2. Auxiliary Function (M Code)

Auxiliary function is used to control the ON and Off of mechanic function and the command format is immediately following after M code with a number of 1 or 2 digits. Auxiliary can not be written into the same block as G code.

M01 : Program Pause

Explanation : When system executes automatic work till the M01 code, it will pause the program execution for the convenience to inspect the proceeding work status. If required to continue work program, just press START button.

M02 : Program End

Explanation : The general command M02 is written in the ending of main program. When system executing till this command, the system will view the program is executed to the end and make the cursor stay at the beginning block of program.

M03 : Spindle CW

M04 : Spindle CW

M05 : Spindle Stop

M06 : Automatic Guide Exchange

Format : M06 T* *

Explanation : This only works at the machine with 4 axis and Auto-Guide-Change(AGC) instrument. The "***" is the guide number. The guide number is corresponding to real size of guide, so as to change the right size guide. Therefore, it is need to set the corresponding data for guide number and real size in AGC frame of system parameter sub-mode. When executing this instruction, it is first to change the guide and changing the electrode.



M08 : Fluid Supply Start

M09 : Fluid Supply Stop

M20 : Automatic Electrode Exchange

Format : M20

Explanation : Running the changing Electrode procedure. In the machine with the Auto-Tool-Change(ATC) instrument, the changing the same size electrode procedure is auto-run. In the machine with no ATC instrument, changing electrode is executed by hand.

M36 : Hole machining

Format : M36 D__E__ ;

Explanation :

D E code may be used or not.

D code is the group number of deepness table, the setting range is D00 ~D99 °

E code is the group number of condition table, the setting range is E00~ E99 °

The total deepness in hole machining is difference according to the machining type, and list at the following

Machining type	Total machining deepness
Blind	Desired Deep + Wear
Through	Desired Deep + Wear + Through Hole Compensation



M98 : Call sub-program

Format : M98 P_____ L_____ ;

Explanation : P* * * *, and * * * * is sub-program number
0000~9999

The format of sub-program is O* * * *

L is the times for repeated calling.

Example : M98 P0011 L2;

Call program O0011, and execute it twice.

M99 : Sub-program ends, and back to main program.

Explanation: When sub-program executes to the end, it must end with
M99, and make the cursor back to main program to continue its
execution. If main program executes till M99, it will back to the first part
of main program to continue the program execution.



M Code List

M Code	Function Description
M01	Program Hold
M02	Program End
M03	Spindle CW
M04	Spindle CCW
M05	Spindle Stop
M06	Guide Change
M08	Fluid Supply Start
M09	Fluid Supply Stop
M20	Electrode Change
M36	Machine a Hole
M98	Call Sub-Program
M99	Sub-Program End



Chapter 7 System Alarm Message

System alarm message is divided into 2 types: ALARM and WARNING. When the system is giving out alarm in work execution, the work will be forced to reset. At this moment, the operator should, if available for problem solving, inspect the machines according to the message, and just press **RESET** button to clear the situation, reset to normal operation. But in some situation, it can not reset the alarm. It is need to close the system, re-opening the machines. When warning messages happen to system, the work will not be forced to pause. At this moment, the operator can, if available for problem solving, inspect the machines according to the messages. The messages will be cleared automatically, no need to press **RESET** button.

7.1. Alarm

7.1.1 MOT Alarm

MOT 001 : X AXIS FOLLOWING ERROR!

- (1)The following error of X axis is more than the values set by system parameter NO.42.
- (2) Inspect if the speed is too large.
- (3) Inspect if normal in server drivers
- (4) Inspect if machines operating normally
- (5) Inspect if boards are normal.
- (6) Press RESET and availably continue the operation.

MOT 002 : Y AXIS FOLLOWING ERROR!

- (1)The following error of Y axis is more than the values set by system parameter NO.43.
- (2) Inspect if the speed is too large.
- (3) Inspect if normal in server drivers
- (4) Inspect if machines operating normally
- (5) Inspect if boards are normal.
- (6) Press RESET and availably continue the operation.



MOT 003 : Z AXIS FOLLOWING ERROR!

- (1)The following error of Z axis is more than the values set by MCS parameter NO.44.
- (2) Inspect if the speed is too large.
- (3) Inspect if normal in server drivers
- (4)Inspect if machines operating normally
- (5) Inspect if boards are normal.
- (6) Press RESET and availably continue the operation.

MOT 004 : WAXIS FOLLOWING ERROR!

- (1)The following error of W axis is more than the values set by MCS parameter NO.45.
- (2) Inspect if the speed is too large.
- (3) Inspect if normal in server drivers
- (4)Inspect if machines operating normally
- (5) Inspect if boards are normal.
- (6) Press RESET and availably continue the operation.

MOT 005 : Short IN MOVE!

- (1)The short status is happening in executing G00, G53 or G81
- (2)Press RESET to reset the alarm message
- (3)Solve the short problem, and then cycle start again

MOT 006 : Short In MACHINING!

- (1)Gap voltage is low than the setting voltage and the electrode is going to back.
When the position in back is reach the setting height.
- (2)the reasons are the setting voltage is too high, machine doesn't transfer to machining voltage or it is short for some times in machining.
- (3)Solve the above reason, and then cycle start again.

MOT 011 : Soft Limit Switch +X ON!

MOT 012 : Soft Limit Switch -X ON!

MOT 013 : Soft Limit Switch +Y ON!

MOT 014 : Soft Limit Switch -Y ON!

MOT 015 : Soft Limit Switch +Z ON!

MOT 016 : Soft Limit Switch -Z ON!

MOT 017 : Soft Limit Switch +W ON!

MOT 018 : Soft Limit Switch -W ON!

- (1) Move outside the soft stroke.
- (2) Press RESET, and the availably reset parameters.
- (3) Press JOG to move inside the soft stroke, or enlarge the range of soft stroke.

1.1.2 OP Alarm

OP 001 : Emergency Stop!



- (1) E-Stop gives out alarm
- (2) If there are any emergency, please solve it. Pull up E-Stop, and press Machine Ready key in operating panel °

OP 002 : Limit Switch +X ON!

- (1) X axis reaches stroke limitation (+).
- (2) Press RESET, and then use JOG X- to move machines away from stroke limitation.

OP 003 : Limit Switch -X ON!

- (1) X axis reaches stroke limitation (-).
- (2) Press RESET, and then use JOG X+ to move machines away from stroke limitation.

OP 004 : Limit Switch +Y ON!

- (1) Y axis reaches stroke limitation (+).
- (2) Press RESET, and then use JOG Y- to move machines away from stroke limitation.

OP 005 : Limit Switch -Y ON!

- (1) Y axis reaches stroke limitation (-).
- (2) Press RESET, and then use JOG Y+ to move machines away from stroke limitation.

OP 006 : Limit Switch +Z ON!

- (1) Z axis reaches stroke limitation (+).
- (2) Press RESET, and then use JOG Z- to move machines away from stroke limitation.

OP 007 : Limit Switch -Z ON!

- (1) axis reaches stroke limitation (-).
- (2) Press RESET, and then use JOG Z+ to move machines away from stroke limitation.

OP 008 : Limit Switch +W ON!

- (1) W axis reaches stroke limitation (+).
- (2) Press RESET, and then use JOG W- to move machines away from stroke limitation.

OP 009 : Limit Switch -W ON!

- (1) W axis reaches stroke limitation (-).
- (2) Press RESET, and then use JOG W+ to move machines away from stroke limitation.

OP 010 : X SERVO ALARM

- (1) X axis' servo giving out ALARM.
- (2) Please inspect the error message, so as to find out causes.



(3) Re-open machines.

OP 011 : Y SERVO ALARM

- (1) Y axis' servo giving out ALARM.
- (2) Please inspect the error message, so as to find out causes.
- (3) Re-open machines.

OP 012 : Z SERVO ALARM

- (1) Z axis' servo giving out ALARM.
- (2) Please inspect the error message, so as to find out causes.
- (3) Re-open machines.

OP 013 : W SERVO ALARM

- (1) W axis' servo giving out ALARM.
- (2) Inspect system parameter
- (3) Re-open machines.



1.1.3 INT Alarm

INT 001 : NO SUCH TOKEN

- (1) There are symbols and texts forbidden by workpiece program inputted into data by workpiece program.
- (2) Amend the program errors.
- (3) Pres RESET to release alarm signals.

INT 002 : GRAMMAR ERROR

- (1) There are syntax error within the input data of workpiece program.
- (2) Amend the program errors.
- (3) Pres RESET to release alarm signals.

INT 003 : OUT OF RANGE

- (1) There are MACRO syntax of much complexity, such too many “()” marks.
- (2) Simplify the complex degree or divide into 2 BLOCKS to execute.
- (3) Pres RESET to release alarm signals.

INT 004 : EXECUTE NODE ERROR

- (1) Within system, there are operating representation forbidden to execute.
- (2) Pres RESET to release alarm signals.

INT 005 : FUNCTION ERROR

- (1) Within system, there are functions forbidden to execute. (No happening in normal system)
- (2) System error, please contact the supplier.

INT 006 : DIVIDED BY ZERO

- (1) Within MACRO syntax, there are denominators of 0.
- (2) Amend the denominators without 0 existing.
- (3) Pres RESET to release alarm signals.

INT 007 : VARIABLE OVER RANGE

- (1) Zone variables, common variables, system variable's serial number exceeding available number of arrangement order.
- (2) Amend the serial number exceeding range of variable's serial number.
- (3) Pres RESET to release alarm signals.



INT 008 : DOMAIN ERROR

- (1) Within MACRO syntax, the function definition domain is wrong, such as the negative value of index in square root, or both zero for 2 index of Anti-tangent.
- (2) Change the value of definition domain.
- (3) Pres RESET to release alarm signals.

INT 010 : NOT ALLOWABLE DECIMAL POINT

- (1) The number inputted in addresses, there are numbers forbidden to exist with decimal point.
- (2) Change the value of addresses.
- (3) Pres RESET to release alarm signals.

INT 011 : WORD DATA OVER RANGE

- (1) The number inputted in addresses, there are numbers exceeding available range.
- (2) Change the value of addresses.
- (3) Pres RESET to release alarm signals.

INT 101 : ILLEGAL G CODE

- (1) The input data exist with the forbidden G code.
- (2) Delete the forbidden G code.
- (3) Pres RESET to release alarm signals.

INT 107 : LACK OF FILENAME

- (1) Within the input data, it lacks the file name desired call. (No input on P address)
- (2) Add the desired file name.
- (3) Pres RESET to release alarm signals.

INT 108 : ILLEGAL FILENAME

- (1) Within the input data, there are illegal file names.
- (2) Amend file names.
- (3) Pres RESET to release alarm signals.

INT 109 : FILE NOT FOUND

- (1) The desired to execute is not in system.
- (2) Create the file desired to execute or amend it.
- (3) Pres RESET to release alarm signals.

INT 121 : LACK OF SUB RETURN

- (1) Sub-program of no main program's command
- (2) Add main program's command into sub-program.
- (3) Pres RESET to release alarm signals.

INT 122 : PROGRAM OVERFLOW

- (1) The total of sub-program and MACRO exceed the limitation (8).
- (2) Reduce the number of called tiers.



(3) Pres RESET to release alarm signals.

INT 123 : MACRO OVERFLOW

- (1) The call of MACRO exceeds the limitation (4).
- (2) Reduce the number of called tiers.
- (3) Pres RESET to release alarm signals.

INT 124 : MACRO UNDERFLOW

- (1) Deficient in MACRO Mode Call's STACK
- (2) Confirm the accuracy of variable usage for MACRO Mode Call
- (3) Pres RESET to release alarm signals.

INT 125 : WITHOUT LABEL

- (1) LABEL name does not exist.
- (2) Please confirm the LABEL name.
- (3) Pres RESET to release alarm signals.

INT 126 : BLOCK NOT FOUND

- (1) BLOCK indicated by serial number does not exist.
- (2) Inspection if the BLOCK indicated by serial number exists.
- (3) Pres RESET to release alarm signals.

INT 150 : INSUFFICIENT DATA

- (1) The G code desired to execute is deficient .(For example: deficient in P 、 R 、 Z within G10)
- (2) Supplement the deficient information.
- (3) Pres RESET to release alarm signals.

INT 152 : ILLEGAL IN CAN CYCLE

- (1) Within CAN CYCLE, there are forbidden executions.
- (2) Cancel CAN CYCLE first and execute it.
- (3) Pres RESET to release alarm signals.

INT 170 : ILLEGAL PATTERN

- (1) PATTERN setting is not reasonable.
- (2) Please input reasonable information.
- (3) Pres RESET to release alarm signals.

INT 171 : ILLEGAL DATA

- (1) The inputted information is wrong.
- (2) Please input reasonable information.
- (3) Pres RESET to release alarm signals.

INT 180 : LACK TOOL NUMBER

- (1)Changing guide instructions M06 has no T code
- (2)Press RESET to clear the alarm message
- (3)check and modify the program



INT 185 : NO THE SIZE GUIDE

- (1) Changing guide instructions T code is corresponding to no real size setting
- (2) Press RESET to clear the alarm message
- (3) Check and modify the program, or set the real size with T code in AGC frame

INT 186 : CONDITION FILE NO FOUND!

- (1) Changing condition table, but no that file
- (2) Press RESET to clear the alarm message
- (3) Check and modify the program, or save that condition file



7.2. Warning Message

WARNING! 001 : Air Pressure is too Low

- (1) Air pressure is not enough. It can work partly but the efficiency is bad, auto changing tool and guide can't work.
- (2) Check the machine and solve the air system.

WARNING! 002 : Getting Fire!

- (1) The machine is getting fire. If it is machining, it is going to pause.
- (2) After solving the fire, you can press the CYCLE START key and continue the machining or reset it.

WARNING! 003 : Oil Temp. is too High

- (1) Oil temperature is too high. It can work, but the efficiency is low, maybe it result in the emergency.
- (2) Check the machine and solve this problem.

WARNING! 004 : Oil Level is too Low

- (1) Oil level is too low. It can work, but the efficiency is low, maybe it result in the emergency.
- (2) Check the machine and solve this problem.

WARNING! 002 : Electrode Bank Warning!

- (1) No new electrode warning message from PLC in ATC procedure. The Machine stop, put new electrode into the bank, and set the PLC

WARNING! 003 : Guide Bank Warning!

- (1) No such size guide warning message from PLC in AGC procedure. The Machine stop, put that size guide into the bank, and set the PLC



Anotronic-SKM EDMs
Manual, ZNC, CNC



Anotronic-Ocean
EDM Drilling Machines
Manual, ZNC, CNC



CNC Wire EDM Sub-Contract
On The Latest Technology
Machines



Anotronic Electrochemical
Deburring Machines (ECD)



Multi axis CNC Turning



Fully Automated 5axis cnc Milling

Perhaps You Did Not Realise How Much **ANOTRONIC** Has To Offer!

- Full High capacity Machine Shop including fully Automated 5axis cnc Milling, Multi axis CNC Turning, Grinding etc.
- Laser Part marking
- CNC CMM inspection
- Sale of Standard Electric Discharge Machines (Manual, ZNC & CNC) to take components up to 2500mm x 1200mm x 700mm.
- Design, Manufacture & Sale of Electrochemical Deburring Machines (ECM)
- Comprehensive After Sales Service.
- Design, Manufacture & Sales of EDM & ECM Tooling and Electrodes.
- Multi axis CNC EDM & ECM Sub-Contract Service.
- ECM Sub-Contract Service.
- Wire EDM Sub-Contract Service with cutting areas up to 500mm x 320mm x 420mm HIGH.
- EDM small hole drilling Machines and Sub-Contract Service.
- EDM/ECM Consultancy Service.



CNC CMM inspection



5axis cnc Milling



ANOTRONIC LTD.

Unit 3, Hollingdon Depot, Stewkley Road, Soulbury,

Nr. Leighton Buzzard, Beds., LU7 0DF. England.

Telephone +44 (01525) 270261 Fax +44 (01525) 270235

E-Mail sales@anotronic.com Internet <http://www.anotronic.com>

Anotronic Ltd is a company registered in England & Wales with company number 1658055
Registered at the above address. • VAT number 382-1697-32 • © copyright Anotronic Ltd. 1999-2013