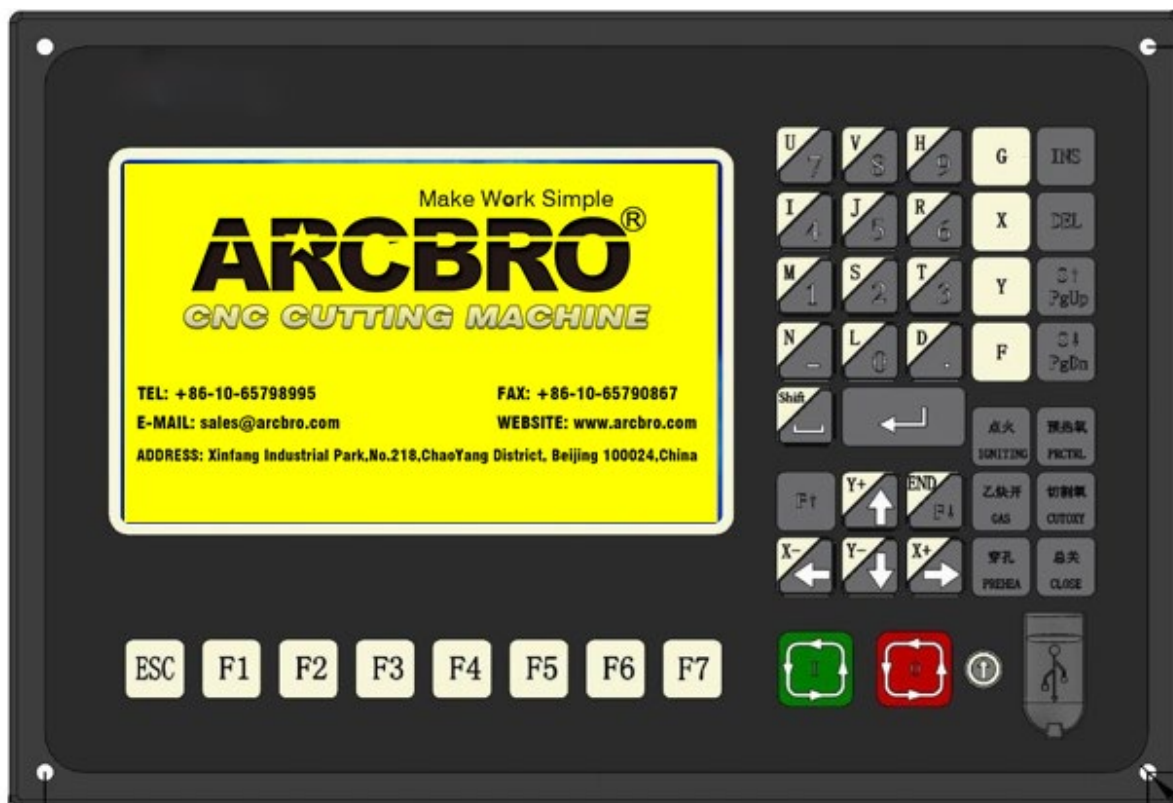


ARCBRO®

Stinger SERIES

CNC Plate Cutting Machine User's Manual



Operator Manual

ARCBRO | Revision 2 | English

Date of issue: August, 2019

Unique Solution

WWW.ARCBRO.COM

Contents

Technical Support.....	7
1. System Functions.....	8
1.1. Technical specifications.....	9
2. Main Menu.....	10
2.1. Menu Features.....	10
2.2. Description of Main Menu.....	10
3. Automatic Function(AUTO).....	12
3.1. Description of Automatic Mode Interface.....	13
3.1.1 SPEED.....	13
3.1.2 Program, Pierce and KERF.....	13
3.1.3 Work Mode--OPERATE.....	14
3.1.4 Input and output.....	14
3.1.5 Machining Parameters.....	14
3.1.6 Coordinate unit.....	14
3.1.7 The six control keys on the Keyboard.....	14
3.1.8 【1】 →ZOOMIN.....	15
3.1.9 【2】 →ZOOMOUT.....	15
3.1.10 【X】 →TEST.....	15
3.1.11 【Y】 →AU-SPD.....	15
3.2. Function selection in AUTO mode.....	15
3.2.1 【F1】 →SECTION.....	15

Service Support Spirit

3.2.2 【F2】 →MAN	16
3.2.3 【F3】 →RE-BREAK	16
3.2.4 【F4】 →VIEW.....	16
3.2.5 【F5】 KERF	17
3.2.7 【F8】 RETURN	17
3.2.6 【F6】 OTHERS	18
3.3. Automatic startup.....	22
3.3.1 Cutting speed	22
3.3.2 Automatic machining start.....	22
3.4. Adjustment of cutting position of automatic control.....	22
3.4.1 In the process of automatic processing, only the keystrokes effective:.....	22
3.4.2 Pause after the adjustment of position of the cutting	24
3.5. Original Path Return	25
3.5.1 Return along original path.....	25
3.5.2 Return at G00(when reach a piercing point)	26
3.5.3 Operation after returned.....	26
3.5.4 Repeat the above operation procedure until the desired effect is achieved.....	26
3.5.5 To exit machining mode	26
3.5.6 Total row number and beginning row in return process.....	26
3.6. Breakpoint Restoration Process	27
3.6.1 Breakpoint restoration.....	27
3.6.2 Attention:	28

Service Support Spirit

3.7. SECTION(section selection).....	28
3.7.1 Start SECTION function.....	28
3.7.2 Two cases for SECTION machining option:	29
3.8. MOVE HOLE for Thick Plate.....	30
3.8.1 HOLE	30
3.8.2 MOVE HOLE	30
3.8.3 NO HOLE	30
4. MAN(Manual Mode).....	31
4.1. Description of Manual Mode Window.....	32
4.1.1 [↑], [↓], [←], [→] directional control keys and [G] Continue Key	32
4.1.2 【F1】 AUTO	32
4.1.3 【F2】 STEP	32
4.1.4 【F4】 CORR.....	32
4.1.5 【F5】 H-SPED.....	33
4.1.6 【F6】 CLS-CO	33
4.1.7 【F8】 RETURN	33
4.1.8 【F3】 OTHERS	35
5. EDIT Mode	39
5.1. Description of EDIT Menu	39
5.1.1 【F1】 NEW.....	39
5.1.2 【F2】 LOAD	39
5.1.3 【F3】 SAVE.....	40

Service Support Spirit

5.1.4 【F4】 DELE	40
5.1.5 【F5】 DELL	40
5.1.6 【F6】 USB-	41
5.1.7 【F7】 VIEW	41
6. Command System	42
6.1. Description of Programing Symbols	42
6.2. Coordinate System	42
6.3. G: Basic Preparatory Commands	43
6.4. M Auxiliary Commands	48
7. SETUP (Parameter Setting)	51
7.1. Parameter Description	51
7.2. SETUP(parameter setting)	52
7.2.1 Speed parameters	52
7.2.2 System parameters	54
7.2.3 Flame Cutting Parameters	56
7.2.4 Plasma Parameters Setting	58
7.2.5 Control Parameters Setting	59
8. LIBMINIT (Library of Patterns)	63
8.1. LIBMINIT Setting	63
8.2. Selection of Pattern Elements	64
8.3. Arrangement and Layout of Pattern Element	64
9. Diagnose	66

Service Support Spirit

9.1. Check Input/output Ports.....	67
9.2. Output Check.....	67
9.3. Input Check	67
10. System input/output interface connected	68
10.1. The system input principle.....	68
10.2. The principle of the system output:	69
10.3. Input/output connect definition	69
10.4. 15 core motor port definitions	71
10.5. When using flame cutting typical wiring (DB15).....	72
Appendix I: Instructions for Software Upgrading Operation.....	73
Appendix II: Troubleshooting	74

Technical Support

Thank you very much to choose ARCBRO product, our whole engineer department work for you since the day you receive machine. When you have any questions during assembling or operating, it is free to contact us by Call, Email, Online help 7x24 hours.

Wish you enjoy a wonderful CNC cutting travel.

ARCBRO CNC Technical Support & Solutions

7x24 Hours

+86-10-65798995

support@arcbro.com

www.arcbro.com/about-arcbro-cnc-cutting-solutions/



MACHINE ▾ PLASMA/FLAME CUTTER ▾ BLOG ▾ SUPPORT ▾ ABOUT US ▾ 🔍

Contact

by arcbro / August 7, 2015

Contact Us for CNC solutions



E-mail: Please leave message to us by live chat —

>>

Contact ARCBRO Portable CNC Cutting Machine

We will reply as soon as possible



Address: Xinfang Industrial Park, No.218, ChaoYang District, Beijing 100024, China

Tel: 0086 10 65798995

Fax: 0086 10 65790867

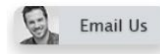
Share this:

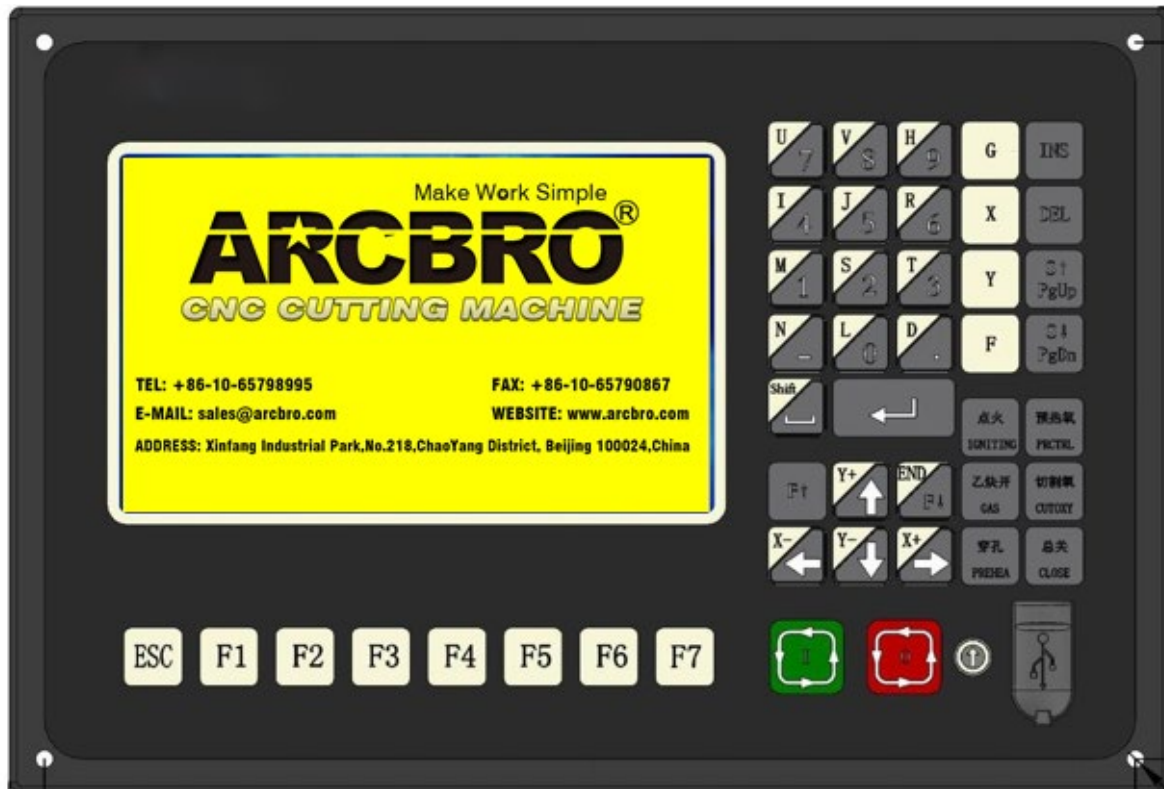


Our System

- 7/24 Hours Te
- Contact
- Any question, an
- Tel: +86-17610780555
- Email: info@arcbro.com
- Why ARCBRO

Follow Us





1. System Functions

- ◆ CNC System for Cutting is designed to work with torch/plasma, high-pressure water jet and laser cutting machines and extensively used in metal working, advertisement fabrication and stone machining businesses.
- ◆ High reliability as well as good resistance to plasma disturbance lightning and surge.
- ◆ Applied torch/plasma cutting technology, able to perform corner speed control and height control automatically;
- ◆ Kerf compensation, reasonability check and report for user's option;
- ◆ Breakpoint restoration, automatic power-back recovery and automatic breakpoint memory,
- ◆ Random section and piercing point selection,
- ◆ Extended piercing for thick plate and bridging feature for thin plate,
- ◆ Optional piercing position feature in mode of RETURN, SECTION(section selection) and RESBREK(breakpoint restoration), very convenient for user control,
- ◆ Ready for transitional cutting operation at any moment,

Service Support Spirit

- ◆ Special short line machining feature based on smooth travel, extensively applicable to metal blanking as well as advertisement and ironwork fabrication, etc.
- ◆ Parts library of 24-type patterns, extensible and customizable, including the common machining parts,
- ◆ Operation menu both in Chinese and English, dynamic graphic display, 8-times zoom, free point automatic tracking, USB disk program and timely software upgrade.

1.1. Technical specifications

- ◆ Processor: Industrial ARM9 CPU
- ◆ Display: 7" Color LCD
- ◆ Input/output: 13 channels of optical isolation input and 8 channels of optical isolation output
- ◆ Interlocked axles: 2 axles, extensible to 4 interlocked axles
- ◆ Maximum speed: <24m/min
- ◆ Pulse equivalency: variable, electronic gear numerator and optional denominator in the range from 1 to 65535
- ◆ Memory space: 32M-64M oversized memory capacity for user program and no restriction to machining program
- ◆ Machine case size: 298×202×95.2(mm)
- ◆ Operation temperature: 0 to +40°C; storage temperature: -40 to +60°C

2. Main Menu

2.1. Menu Features

The design of hierarchy-based functional windows is adopted for system operating display. If a function is invoked in the main menu window, the corresponding sub window menu will pop up. Press [F1] to [F7] to select the corresponding function according to window prompt. Press ESC to abandon the selection and return to the upper level of menu.



Fig.1.1 Main menu display of system after booting

2.2. Description of Main Menu

VER : The version information of software and hardware is indicated at the bottom left corner for your information.

- 【F1】 AUTO** : Programmed control of automatic machining.
- 【F2】 MAN** : To manually control the position of cutting nozzle.
- 【F3】 EDIT**: To edit/modify/input/output the machining program.
- 【F4】 SETUP** : To set system parameters.

Service Support Spirit

【F5】 DIAGNOSE: To check the input/output information of cutting machine.

【F6】 LIBMINIT: To set normal pattern and plan material.

[G] [G] [3] Initial Setting : A dialog box as follows will appear:



Fig.1.2

Parameter: To restore the factory parameter setting;

FORMAT FILES: To format user's program space;

LANGUAGE: Choose language as show

Press PgUp/PgDN on the keyboard to select the language then press Enter;



Fig.1.3 select LAN

3. Automatic Function(AUTO)

In the main menu, press [F1] to enter the automatic function window, as shown in the figure below.

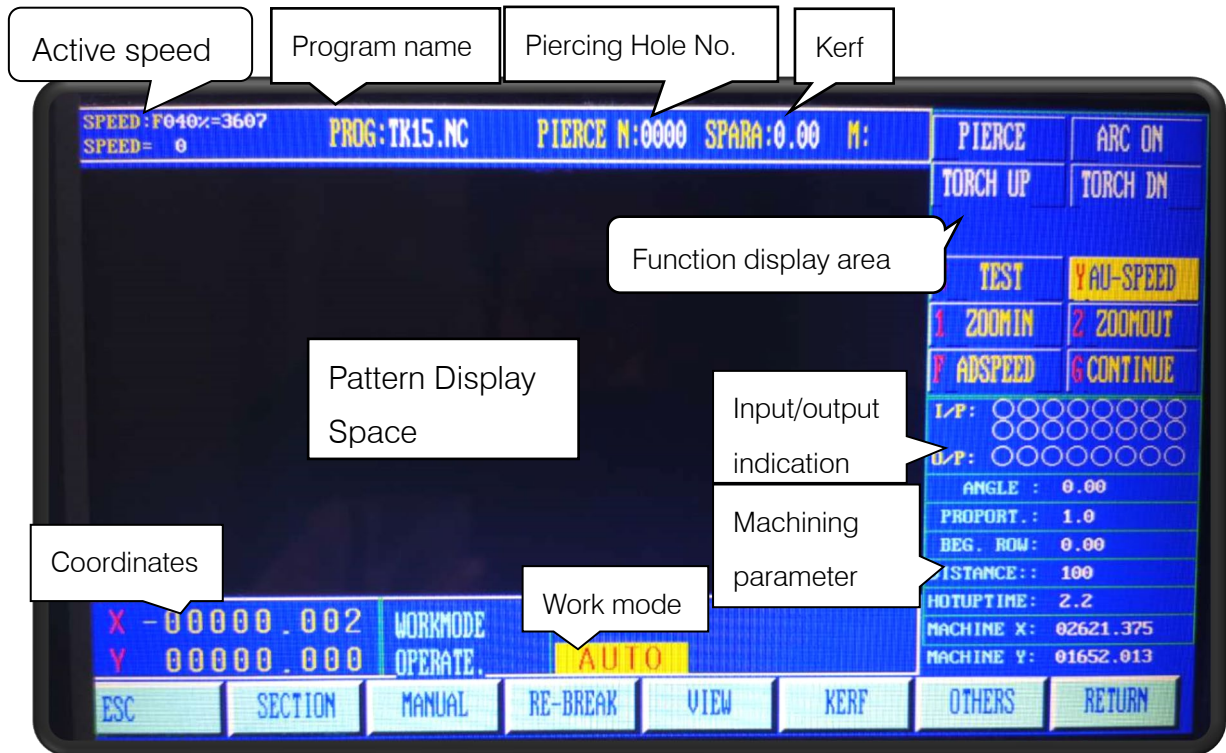
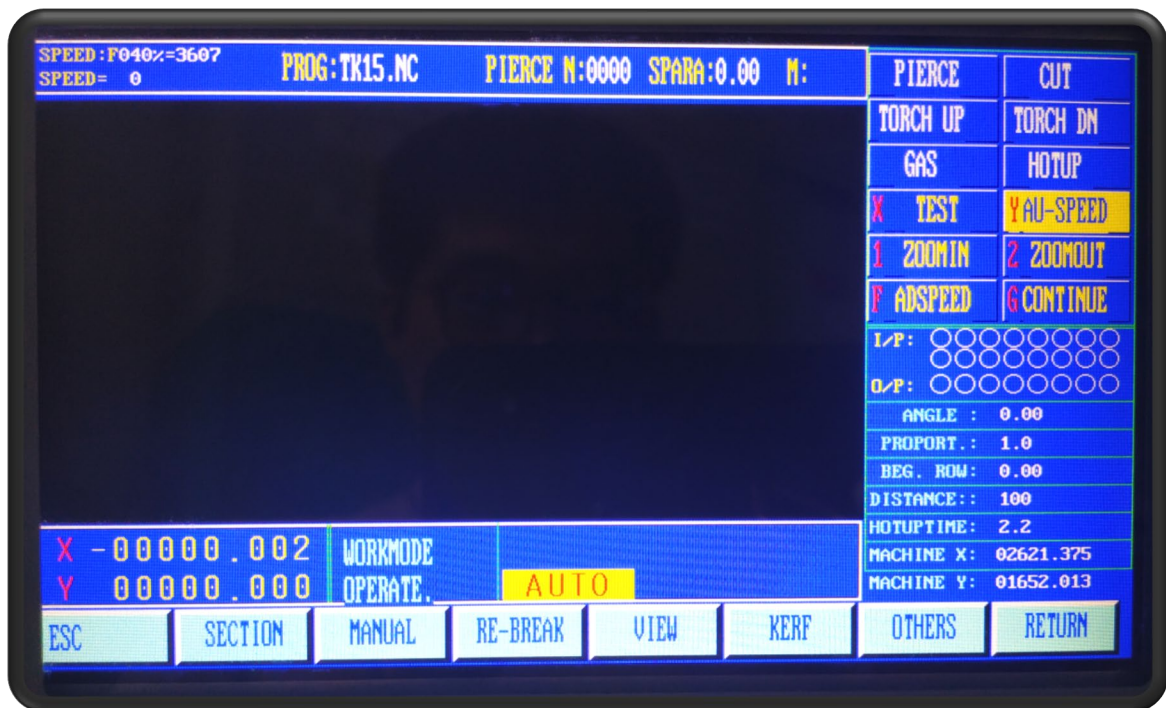


Fig. 3.1 Plasma Mode



Flame mode Fig. 3.2 Automatic Function (AUTO) Interface

Service Support Spirit

3.1. Description of Automatic Mode Interface

In the system work under the main menu, press [F1] into the function of automatic main picture:



3.1 interface automatic way specification

3.1.1 SPEED

In automatic mode, at the top left corner of the screen, it shows $F \times$ (multiplying factor of automatic machining speed) = Set machining speed.

In manual mode, at the top left corner of the screen, it shows $F \times$ (multiplying factor of manual speed) = manual speed.

SPEED indicates the actual speed while the active speed multiplying factor is adjustable with [F ↑] and [F ↓].

It is allowed to adjust the speed quickly by pressing [F] key in this Interface. And then, press ENTER key to confirm the selection.

Note: The speed may be indicated in **Metric** or **English** system depending on the Metric/English Selection in **SETUP** (Refer to **Parameter Control**).

3.1.2 Program, Pierce and KERF

They are used to indicate the name of machining program;

Service Support Spirit

Serial number of pierced hole(automatic clear in automatic machining mode)

The active compensating width of kerf respectively.

3.1.3 Work Mode--OPERATE

WORKMODE field indicates the active operating mode such as rotation selection, breakpoint recovery, mirroring and section selection.

OPERATE field indicates such information as machining/pause, limit alarms and time delay.

3.1.4 Input and output

Below the heavy-current switches, there are 3 rows×8 indicators ○.

The top indicators show the status of 13 input ports, ○ indicating no signal input and ● indicating signal input.

The bottom indicators show the status of 8 output ports, ○ indicating no signal output and ● indicating signal output.

See the description of **DIAGNOSE** function for the definitions of input/output ports.

3.1.5 Machining Parameters

This zone indicates various parameter values in the current machining mode.

3.1.6 Coordinate unit

The coordinates may be indicated in **Metric** (mm) or **English** system(inch) depending on the Metric/English Selection in SETUP(Refer to Parameter Control).

3.1.7 The six control keys on the Keyboard.

【IGNITION】 : Refer to M20 for ignition feature.

【PREHEA】 : To open the solenoid valve of preheating oxygen. Refer to M24for details.

【GAS】 : To open the solenoid valve of acetylene (gas). Refer to M10 for details.

【CUT】 : To open the solenoid valve of cutting oxygen; refer to M12 for details. To turn on the arcing switch if it is **plasma**.

【PIERCE】 heavy-current control key

A piercing process, the specific operation procedure is as follows:

Flame machining case: rise the cutting torch(M72), open the cutting oxygen(M12) and lower the cutting torch(M73).

Plasma machining case: execute M07 command.

Service Support Spirit

Note: This is a very important feature. It will be used repeatedly in pause, return and extend piercing operations. When preheat is done, pressing [PIERCE] key will start up piercing operation directly.

【OFF】 : To shut off all the heavy-current output.

【S ↑】 : Push it down to rise the cutting torch and release to stop the torch.

【S ↓】 : Push it down to lower the cutting torch and release to stop the torch.

3.1.8 **【1】** →ZOOMIN

The pattern will be magnified one times by one keystroke and it can be magnified to eight times maximum by three keystrokes. As show pic.

3.1.9 **【2】** →ZOOMOUT

To restore to normal pattern size. As show pic.

3.1.10 **【X】** →TEST

If **【X】** key is pressed, the system will run the program at top speed without executing M command. This feature is often used to position quickly or check the working size of steel plate. This operation can be paused at any moment or cancelled by pressing **【X】** key one more times.

3.1.11 **【Y】** →AU-SPD

Since the manual speed and automatic speed is separate in the system, this key is used to switched between automatic mode and manual mode to adjust the speed multiplying factor.



3.2. Function selection in AUTO mode

It is allowed to load a machining program by means of **EDIT** feature. If a program has been loaded with same name, it can run directly. If the program is invoked from an USB disk, especially a large program, it can be run directly with USB disk connected instead of saving it into the system.

3.2.1 **【F1】** →SECTION

To assign system to start up machining operation from any section or piercing point.

Service Support Spirit

It is often used when machining needs to start from a certain section or only a part needs being processed. Refer to **SECTION** feature for details.

3.2.2 【F2】 →MAN

To switch the system to manual mode.

3.2.3 【F3】 →RE-BREAK

After this function is selected, pressing **【START】** key to fulfil breakpoint restoration. Refer to **RE-BREAK** feature for details.

3.2.4 【F4】 →VIEW

It is used to test the program for any error. When this feature is selected, the pattern of machining program will be displayed with a cross cursor at the origin.

[S] key is used to magnify the pattern, one times by one keystroke and eight times maximum by three keystrokes. [Q] key is used to return the pattern to original size. The display position of pattern is moveable by pressing [↑], [↓], [←] and/or [→] key.

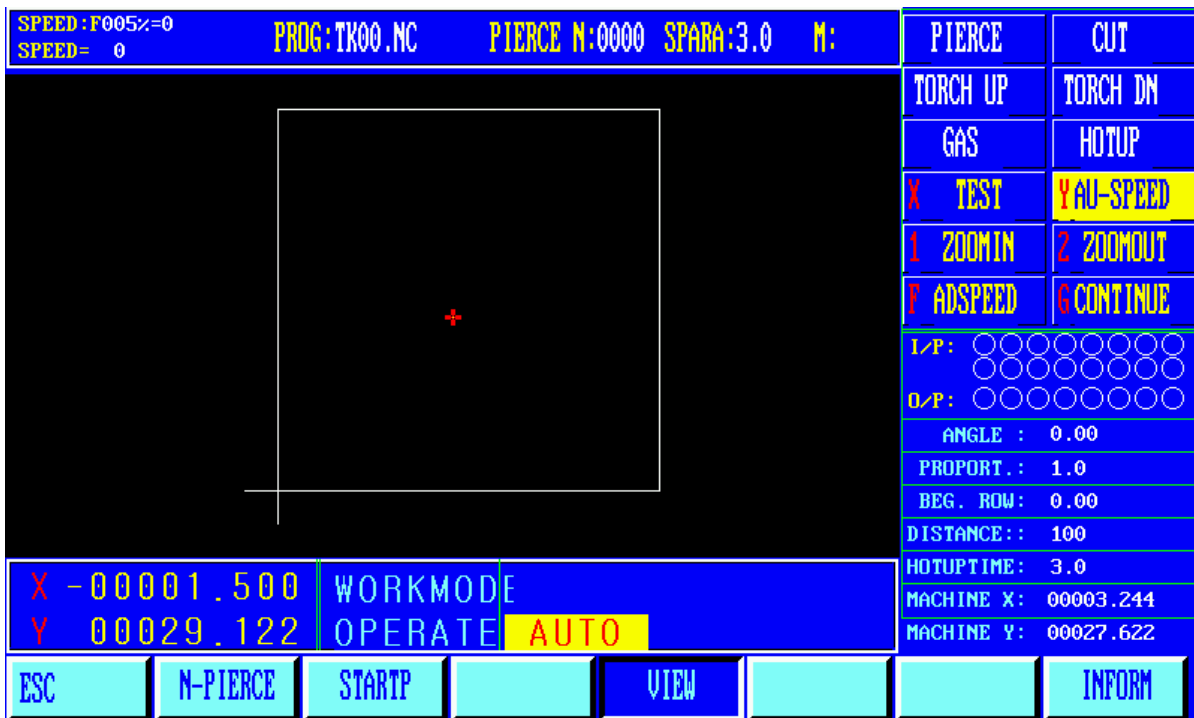


Fig. 3.3

Service Support Spirit

【F1】 →N-PIERCE

Select the perforation position. This graphic has 6 perforation positions. You can select the position according to the number.

【F2】 →STARTP

Starting position; A graphic has four starting positions, which are top left, bottom left, top right, bottom right;

Every time you press the STARTP key it changes a position



【F7】 →INFORM:

Information: Displays details of the current drawing



Fig. 3.5

3.2.5 【F5】 KERF

This key is used to reminder input of kerf compensating width; if no necessary to compensate (usually in blanking), simply enter 0.

3.2.7 【F8】 RETURN

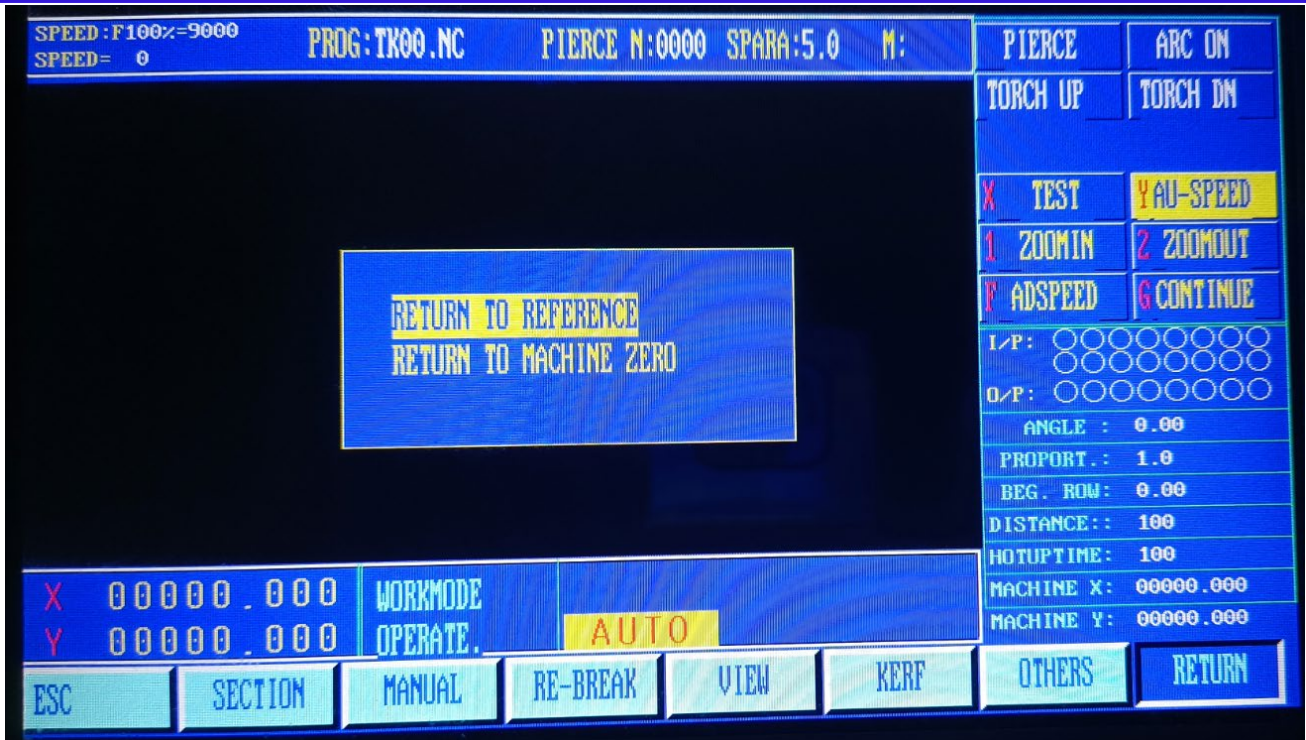
Two choices appear after pressing F8:

Choose the first one: RETURN TO REFERENCE;

【RETURN TO REFERENCE】 : This position is the coordinate reset position;

【RETURN TO MACHINE ZERO】 : It's a constant position, usually in the lower left corner of the machine

Service Support Spirit



3.2.6 【F6】 OTHERS

This key is used to enter the next interface as shown in Fig. 3.6below.

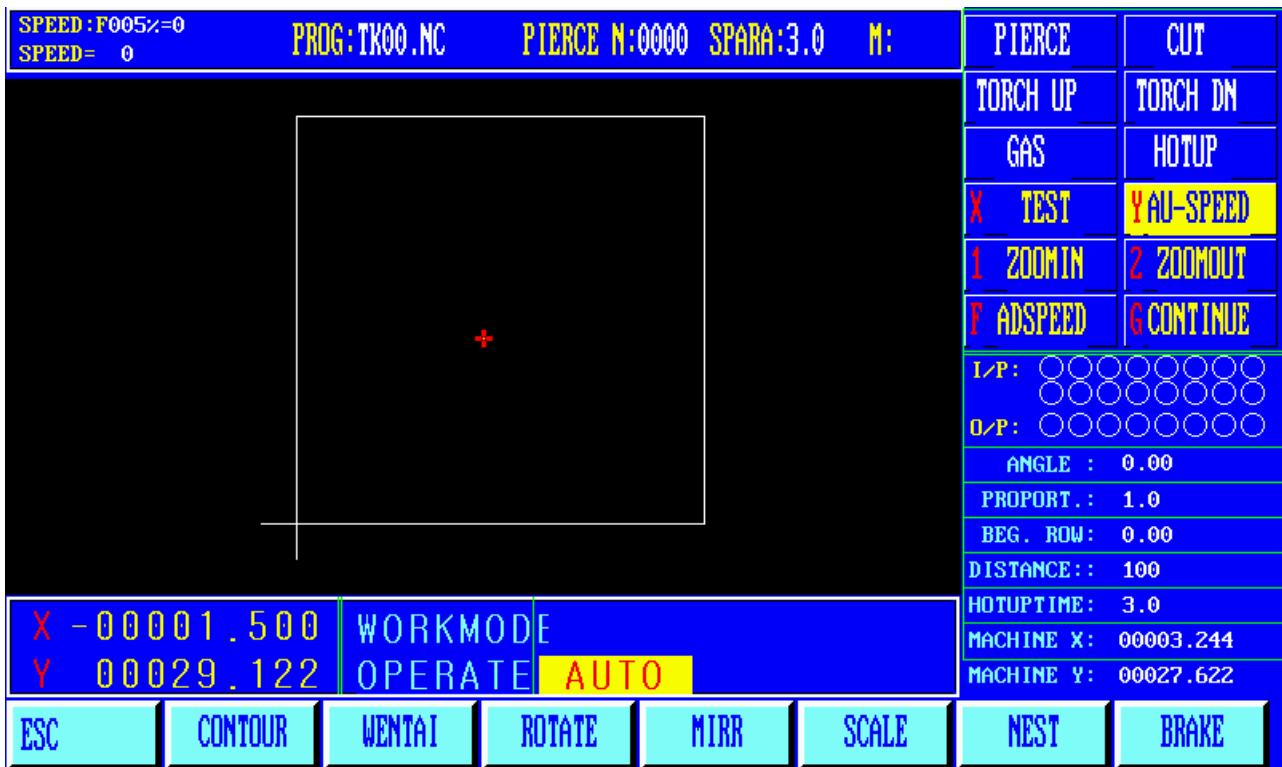


Fig. 3.6 OTHERS interface

Service Support Spirit

3.2.6.1. 【F1】 CONTOUR

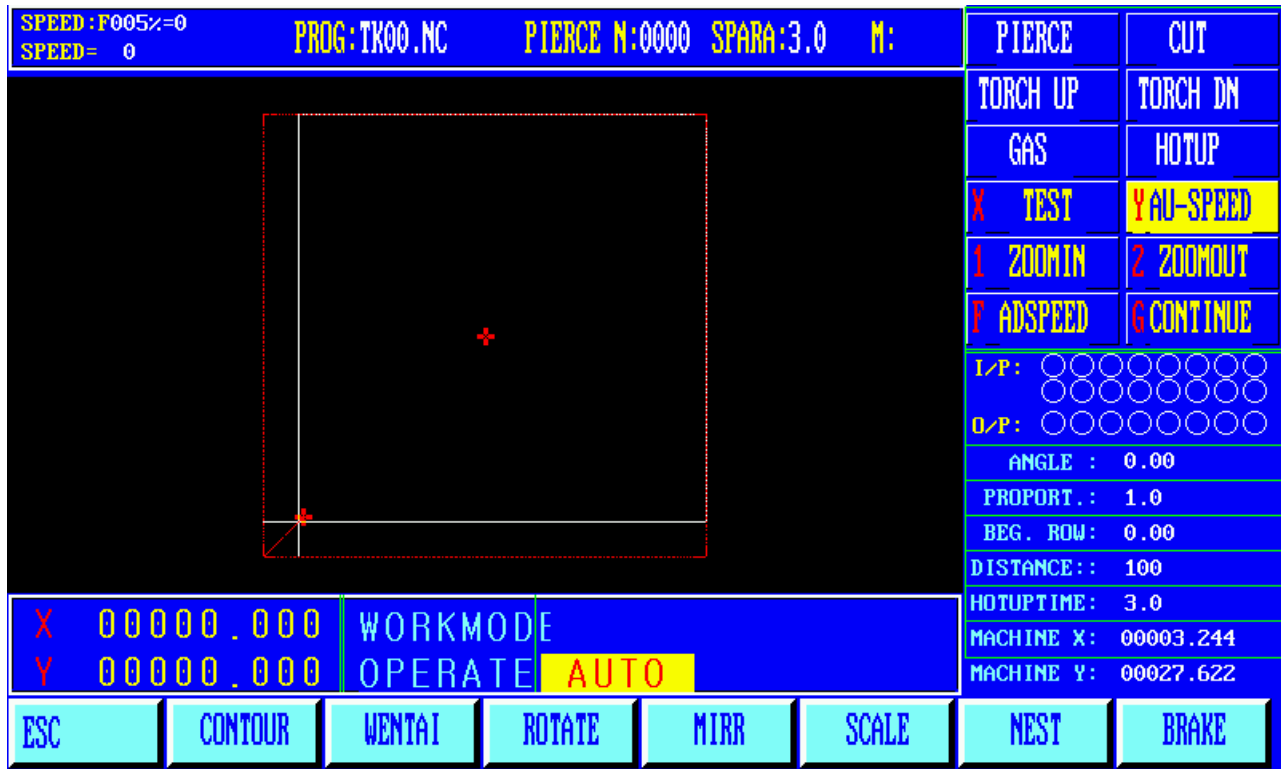


Fig. 3.6.1 CONTOUR interface

The user before processing, can be determined by taking the contour line, the machining path is beyond the scope of the steel sheet.

Press [F1] key to start walking contours, if not on the current torch reference point (coordinates not zero), the system will prompt:

The current point positioning --- will torch the current position as a reference point.

Reference point location --- torch to return to the reference point, and then began to walk contour.

When walking contours, if the torch over the range of steel, according to the [Pause] key to move the position of the torch to the plate edge, then press [Start] button,

Then the system prompt: Modify the reference point "EN / ESC?" Press Enter to confirm the reference point if changed, the current location will continue to operate as a contour line. Press [ESC] to abandon changes. This operation can be repeated many times do run until the position you prefer.

Service Support Spirit

3.2.6.2. 【F2】 WENTAI

The system will prompt to run WENTAI-based machining program if this key is pressed. WENTAI is a word machining software for CAD/CAM, widely used in cutting graphic characters in advertisement business. This feature is required to run a WENTAI-based program.

3.2.6.3. 【F3】 ROTAE { steel plate correcting feature }

3.2.6.3-1 Machining at rotation angle

This feature is used when the steel plate is not positioned perfectly or the steel plate needs to rotate an angle before machining. It is allowed to use **Rotate** function along with **BEG. PT.** and **END PT.** in **Manual** mode or input the expected angle directly. After acknowledged, the system will start the machining program at the appointed rotation angle.

Note: Counter clockwise direction is taken as positive angle.

3.2.6.3-2 Example

The system is able to identify and calculate the rotation angle by measuring the origin point and the end point of an edge of steel plate(a straight line). The procedure is as follows:

Determine the datum line. Take one borderline of steel plate as baseline and move the cutting torch to the origin point of this baseline. Press [F2] to set the beginning point of measurement. Control the cutting torch and let it move along the baseline to the end point. The larger distance between the beginning point and the end point, the more accurate of the measurement. Aim the cutting torch at the baseline and press [F3] to set the end point of measurement.

At this moment, the rotation angle of the datum line is calculated. After the rotate function is fulfilled, the rotation angle will appear in the OPERATE field, as shown in Fig. 3.2 below.

3.2.6.3-4 [F4] MIRR

It is possible to select **X Mirror**, **Y Mirror** and **No Mirror** if press [F5] key continually. When X Mirror is selected, the machining program will run along the symmetry direction of X-axis, as appears inverted horizontally.

When Y Mirror is selected, the machining program will run along the symmetry direction of Y-axis, as appears inverted vertically. When default No Mirror is selected, the machining program will run normally.

Service Support Spirit

3.2.6.3-5 [F5] SCALE

The system will reminder user to input scale if this key is pressed. When the machining program is executed, the work size will magnify or minify as per this scale. This feature is very useful in cutting graphic character.

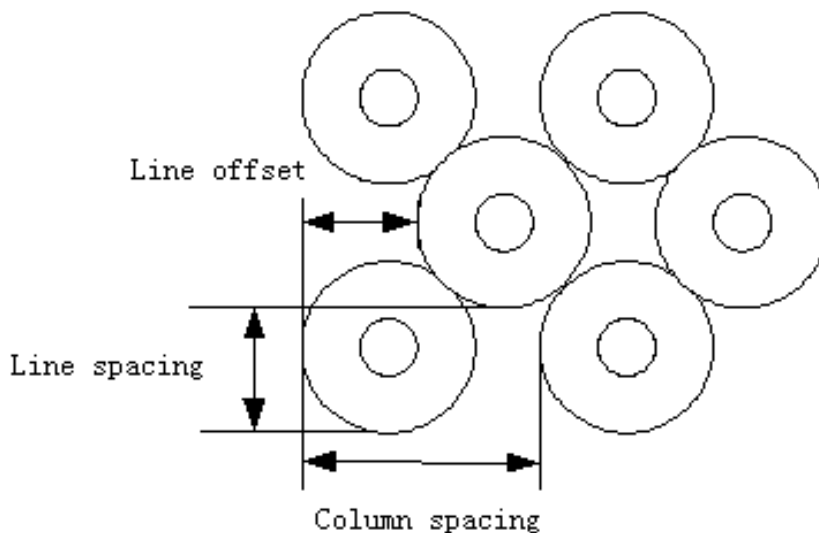
3.2.6.3-6 [F6] NEST

For a single process that has programmed, can through this function arrangement processing. System prompt:

Arrangement of rows - Y direction to arrange the number of rows Arrange the number of columns
- the column number of the X direction is arranged.

Line spacing - spacing between the X direction of the workpieces. The gap between the column
spacing -Y direction of the work pieces.

Line offset - the even lines, migration distance to the right.



3.2.6.3-7 [BRAKE] broken bridge function

Choose this feature, during processing, every cutting length (broken bridge point. Mm.

See [parameter] the [control]), will set up a bridge cut-off point, partition bridge length (MM)
(see [parameter] the [control]) controlled by the operator to continue after cutting.

Note: this function has remained until then click this button to cancel the function.

Service Support Spirit

3.3. Automatic startup

3.3.1 Cutting speed

When machining, the automatic speed ratio adjustment.

The speed of execution = machining speed limit * ratio, 【 F 】 , 【F ↑】 , 【F ↓】 is automatic rate adjustment. The two speed ratio once set, it is forever, is not affected by the shutdown.

3.3.2 Automatic machining start

1) Before automatic machining start

To choose the correct processing program, select the appropriate processing rate (ratio), the cutting torch on the cutting height (after the start process, automatically cutting torch lift (execution M70)), and other preparations in place, can start the process automatically.

2) There are two ways of automatic processing start:

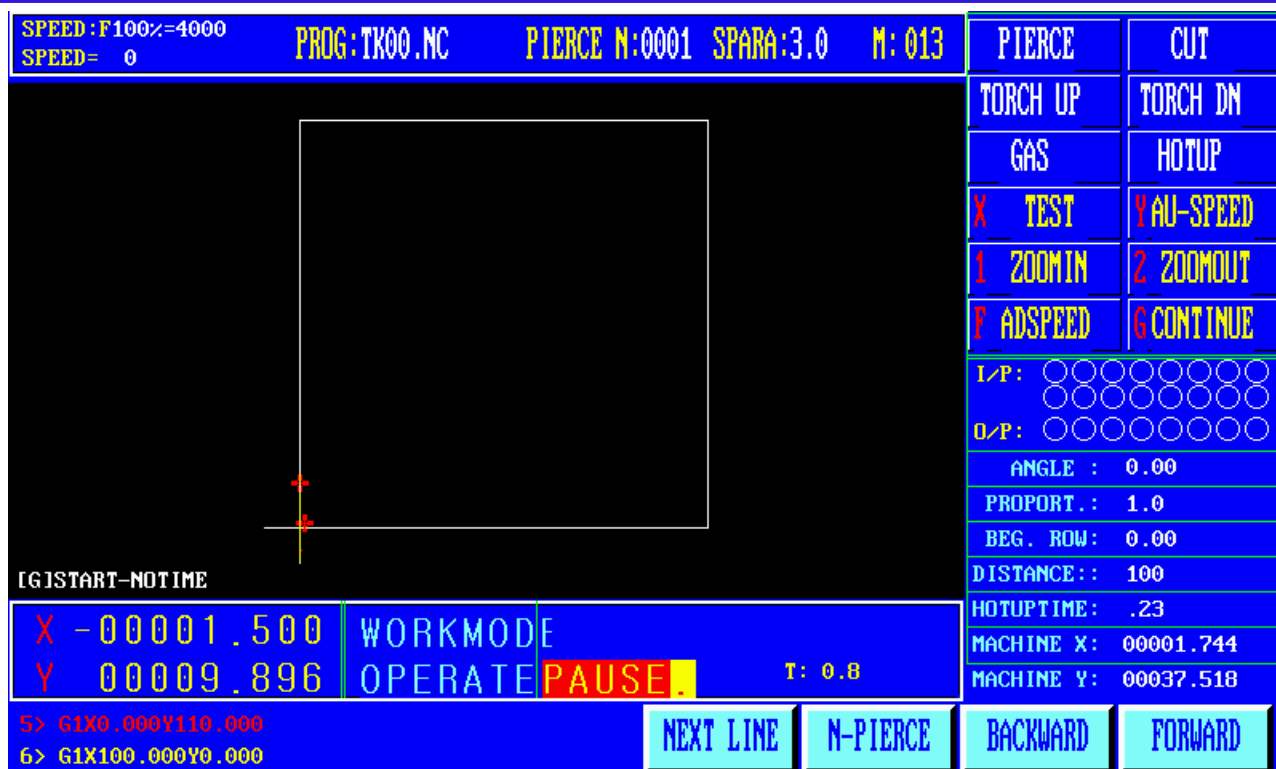
According to the green "start" button on the panel.

According to external "start" button (see "input and output port definitions")

3.4. Adjustment of cutting position of automatic control

3.4.1 In the process of automatic processing, only the keystrokes effective:

[Pause]: press this button, the system movement slowing down to stop, close cutting oxygen (plasma processing, shut off the ignition switch), closed higher controller (M39), keep the currently displayed image. If you choose in the parameter is set to "suspended after raise gun (1)" (see control parameters), suspended after cutting torch will lift (M70).



After suspended can be the following:

- A. Select new piercing point, press "F5" key to enter the new perforation after a period, the system automatically go to the new perforation, waiting for the punch operation.
- B. The original track back.
- C. Press "F4" button, select the line escapement operation, the system prompt: Fallback, forward and choose program line, according to mark new selected row cursor position operation, press the "start" key, the machine go to the new line escapement position, waiting for the punch instruction to continue processing.
- D. Adjust the position and so on.
- E. Exit processing.
- F. [Start]button: systems continue to run.
- G. [ESC] key: Exit the machining program, return to the automatic image.
- H. 【F ↑】, 【F ↓】 speed adjusting button: increase or decrease feed speed ratio.
- I. 【S ↑】, 【S ↓】 control cutting torch up and down. Press the corresponding key, Cutting up or down. Raise my hand cutting movement to stop.
- J. [Stop key]: Scram button for external key (see the [external input interface]), the signal from the input port access. When stop is valid, and stop all movement, output shut down, for unexpected emergencies. Due to the conditions of the suspension

System and the following points:

Service Support Spirit

- 1) The external pause button press.
- 2) Plasma processing and choose the "arc pressure test (1)", if the broken arc will be suspended.
- 3) If you choose "" bump shot detection (1)", efficiently gun will be suspended.

3.4.2 Pause after the adjustment of position of the cutting

3.4.2.1 the following several ways of cutting torch position need to adjust:

- 1) Cutting torch is blocked, or need to change, often move the cutting torch to a safe location, after processing to return to the starting point.
- 2) Need outer perforation, don't want to put the punch points on the outer limits of the machining. In the workpiece outside to look for a suitable position (usually the edges of the plate), perforation and then cut along a straight line to the starting point (pause) continue to normal processing.
- 3) Transfer of cut, work piece is bigger, wide is bigger, need to change the local cutting.

3.4.2.2 several operating can adjust the cutting position of the following:

1. Suspension
2. The fallback
3. Punch
4. Parts processing
5. Machining
6. Breakpoint recovery

In the state, if want to change the cutting position, can be directly according to the **【 ↑ 】** **【 ↓ 】** **【 ← 】** **【 → 】** key to adjust the position of the cutting torch (the system ratio for manual rate can be adjusted). After adjustment in place, press the [start] key, the following dialog:

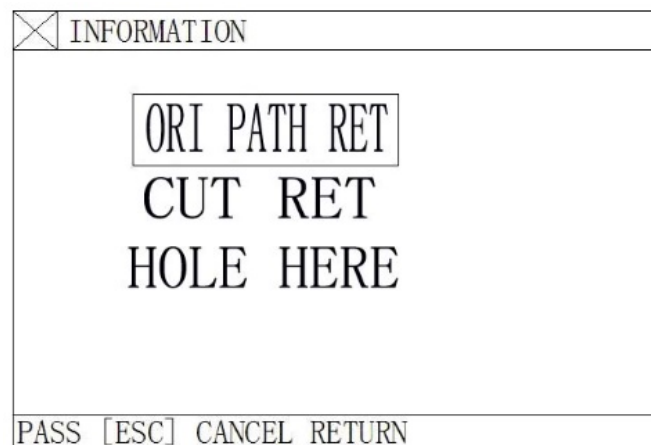


Fig. 3.3 Dialog box of position adjustment options

Service Support Spirit

1) ORI PATH RET(original path return)

Return the cutting torch to the adjustment beginning point at speed G00 and wait for other operation; at this step, it is allowed to press an heavy-current function key such as IGN(ignition), PREHEA(piercing preheat), CUT(cutting oxygen), etc. Tip: The system will continue to process from the breakpoint location if [PIERCE] key is pressed after preheating.

2) CUT RET(cut return)

After piercing, the cutting torch returns from the current position to the adjustment beginning point along a straight line at cutting speed and continue machining along the original path without stop, which is somewhat like extended pierced, making the piercing point more smooth. ;

3) HOLE HERE(piercing here)

After piercing, set the coordinates at current point as coordinates of "adjustment beginning point", and continue machining along the original path to fulfill transitional piercing function.

4) Attention:

Be sure to have the system preheated sufficiently(for flame cutting) before taking steps 2) and 3) , For piercing operation will start right away once the operation is selected.

It is recommended to preheat the system(for flame cutting) and then press [START] key to select appropriate operation.

3.5. Original Path Return

When it is necessary to return the cutting torch along the original path due to failure to cut through in machining operation, handle it in the following ways:

3.5.1 Return along original path

Press [PAUSE] key to suspend the running system. A "pause" mark will appear on the system display in addition to prompt as shown in the figure below.

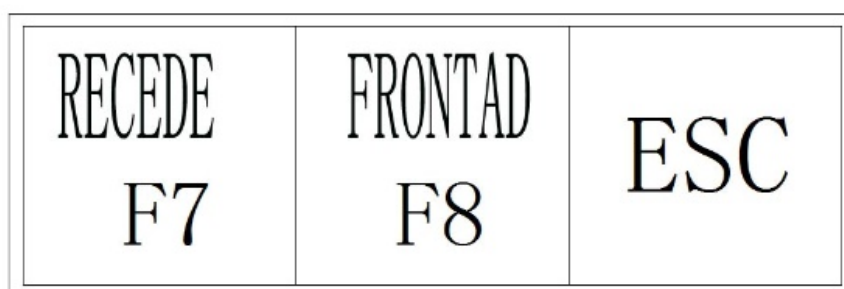


Fig. 3.4 Machining pause display

Service Support Spirit

Press [F6] key to return the cutting torch along the original path.

Press [F7] key to advance along the original path after returning. In the process of return, if the cutting torch fails to reach the required position, press [PAUSE] key again and repeat the above procedure until it reaches the right position.

3.5.2 Return at G00(when reach a piercing point)

In the process of return, the cutting torch will stop at G00 or a piercing point for operator to select going on return or advancing.

3.5.3 Operation after returned

After the cutting torch returns to the designated position, it is allowed to select position adjustment of cutting torch (refer to 3.4) or piercing directly by pressing the appropriate heavy-current function key such as PREHEA(piercing preheat) and CUT(cutting oxygen).

It is common practice to--

Press [PIERCE] key after preheating operation is done:

For flame cutting operation, press CUT key(open the cutting oxygen) if the cutting torch lifts or continue machining if the cutting torch lowers; or

For plasma cutting operation, ignite arc and do not go on machining until the arc ignition is completed.

3.5.4 Repeat the above operation procedure until the desired effect is achieved.

3.5.5 To exit machining mode

In pause state, press [ESC] key to exit machining mode.

3.5.6 Total row number and beginning row in return process

The program segment for return allows 300 rows maximum. In case of breakpoint restoration(RESBREK) or machining section selection(SECTION), the return beginning row is the row containing the current breakpoint or the section selecting row, upon which however the cutting torch is not allowed to return along the original path for machining.

3.6. Breakpoint Restoration Process

3.6.1 Breakpoint restoration

In case of manual pause or power outage in machining operation, the system will automatically save the current position of cutting torch as a breakpoint, which will be kept permanently in the system, whether power the system off or not.

In automatic mode, press [F2] key to select RESBREK function and then [START] key to start breakpoint restoration as long as there is no change with the current program.

If the position of cutting torch does not change, the system will prompt "Breakpoint" when a breakpoint is found and wait for subsequent command. User may select piercing directly or position adjustment of cutting torch. Refer to 3.4 for details.

If the position of cutting torch changed (not at the breakpoint any more), the system will provide three options when a program breakpoint is found, which is virtually position adjustment of cutting torch.

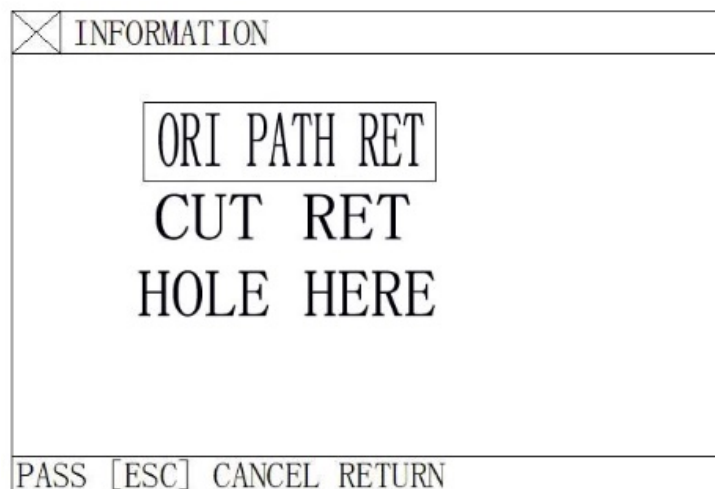


Fig.3.5 Dialog box of restart options after system pause and breakpoint restoration

ORI PATH RET: To return to the breakpoint at speed G00, which is often used for the breakpoint set for cutting torch replacement;

CUT RET: little bit off from the breakpoint after breakpoint restoration, which is somewhat like piercing along the exterior margin, making the breakpoint smoother;

HOLE HERE: the same operation as previous, which can also be used for transitional cutting. Now, it is allowed to press a heavy-current function key such as IGN (ignition), PREHEA (piercing preheat), CUT(cutting oxygen), etc.

Service Support Spirit

Tip: The system will continue to process from the breakpoint location if [PIERCE] key is pressed after preheating.

If press [ESC] key when a breakpoint is found, the system will exit machining mode.

3.6.2 Attention:

Never try to change the machining program, rotation angle and scaling proportion whether it is breakpoint restoration or power-off restoration, which will be automatically recorded by the system and free from affect by power-off operation; otherwise, the system can not find the breakpoint.

3.7. SECTION(section selection)

3.7.1 Start SECTION function

SECTION: to assign system to start up machining operation from any section or piercing point. Press [F1] to select SECTION function and the system will show a menu as shown in the figure below:

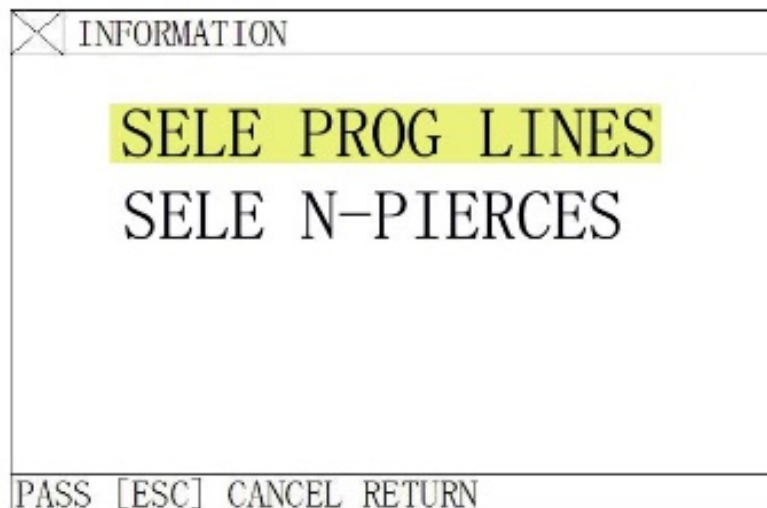


Fig. 3.6 SECTION function menu

Move the cursor with [↑] or [↓] key and make one choice from two machining options. According to your option, the system will prompt to input the serial number of the option (program line number or the serial number of piercing point).

3.7.2 Two cases for SECTION machining option:

3.7.2.1 Transitional machining: to start machining somewhere different from the start position of the program;

3.7.2.2 Repeat the machining operation starting from a certain section of the program.

1. For the former case, a scrap work piece is often used for aiming at the piercing point and machining directly (**HERE POSITION** option);
2. For the latter case, the reference point is used for positioning (**REF POSITION** option).
3. For these two options, the system will prompt an option menu as shown below after start-up:

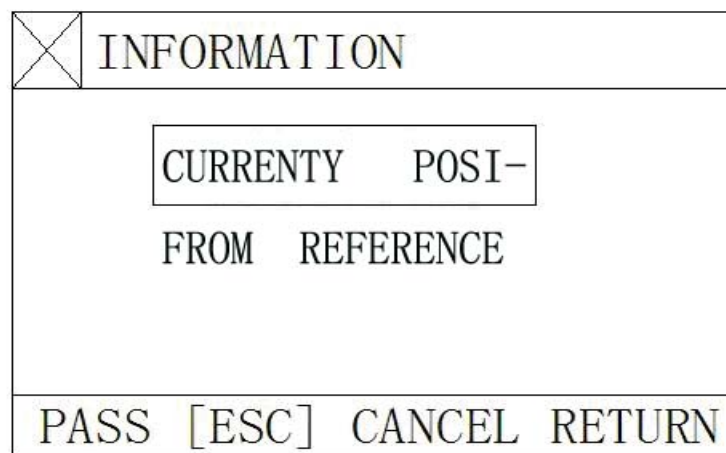


Fig. 3.7 Option dialog box prompted by the system after machining start if SECTION machining is selected

- A. If "**HERE POSITION**" option is selected, the system will draw the full pattern first and then draw a large cross cursor at the piercing point. Operator may press [S] key to zoom in the pattern and observe if it is the expected piercing position. If not satisfied, operator may press [ESC] to exit the machining mode and take the option again.
- B. If it is the expected piercing point, use heavy-current control switch for ignition and preheat before press [PIERCE] key to start operation.
- C. If "**REF POSITION**" option (Reference Point Positioning) is selected, operator should aim the cutting torch at the reference point. After started up, the system will control the cutting torch to reach the piercing point and then perform the operations as above.

3.8. MOVE HOLE for Thick Plate

1. In automatic mode, **MOVE HOLE** option is taken for thick plate.
2. For edge piercing method, it is important to **move the cutting torch to the nearest edge of steel plate before piercing operation.**
3. Preheat the system and press **[START]** key at the end of preheat. The cutting torch will cut the steel plate along a straight line to the piercing point at selected cutting speed and then go on machining.
4. When MOVE HOLE option is taken, it needs to change the **MOVE HOLE** option to 1 in the **parameter control** menu, thus to enables the option. In this way, whenever it is time for piercing operation, the system will prompt an option menu as shown below:

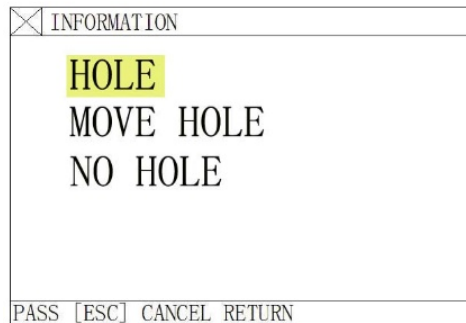


Fig. 3.8 Option dialog box prompted by the system after reach piercing point if edge piercing method is selected

3.8.1 HOLE

To pierce at the original position, which is often used for boring.

3.8.2 MOVE HOLE

Operator may press [↑], [↓], [←] and/or [→] key to move the cutting torch to the edge of steel plate; in the meantime, the speed multiplying factor is adjusted to 5% automatically. The system is preheated.

Press [START] key at the end of preheat. The cutting torch will reach the piercing point along a straight line at selected cutting speed and then go on machining.

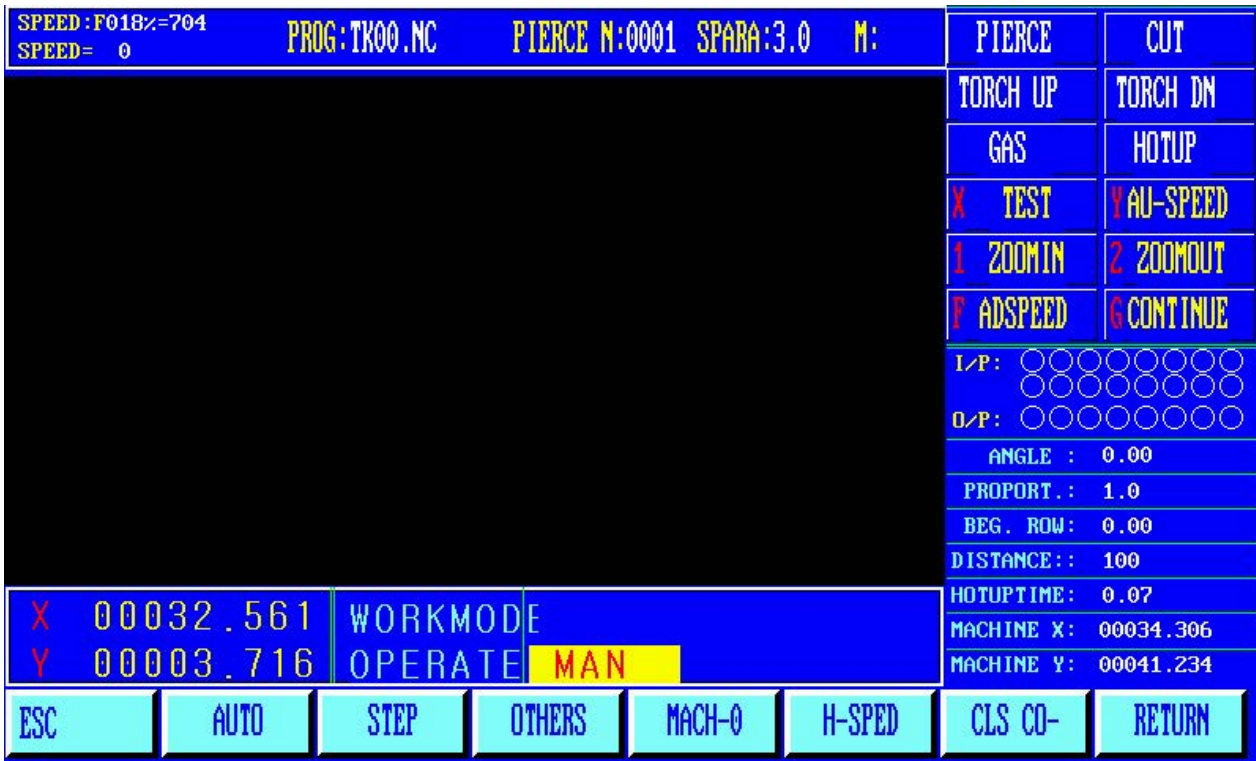
3.8.3 NO HOLE

Instead of piercing, the system moves through the current piercing position to next, and new piercing prompt appears.

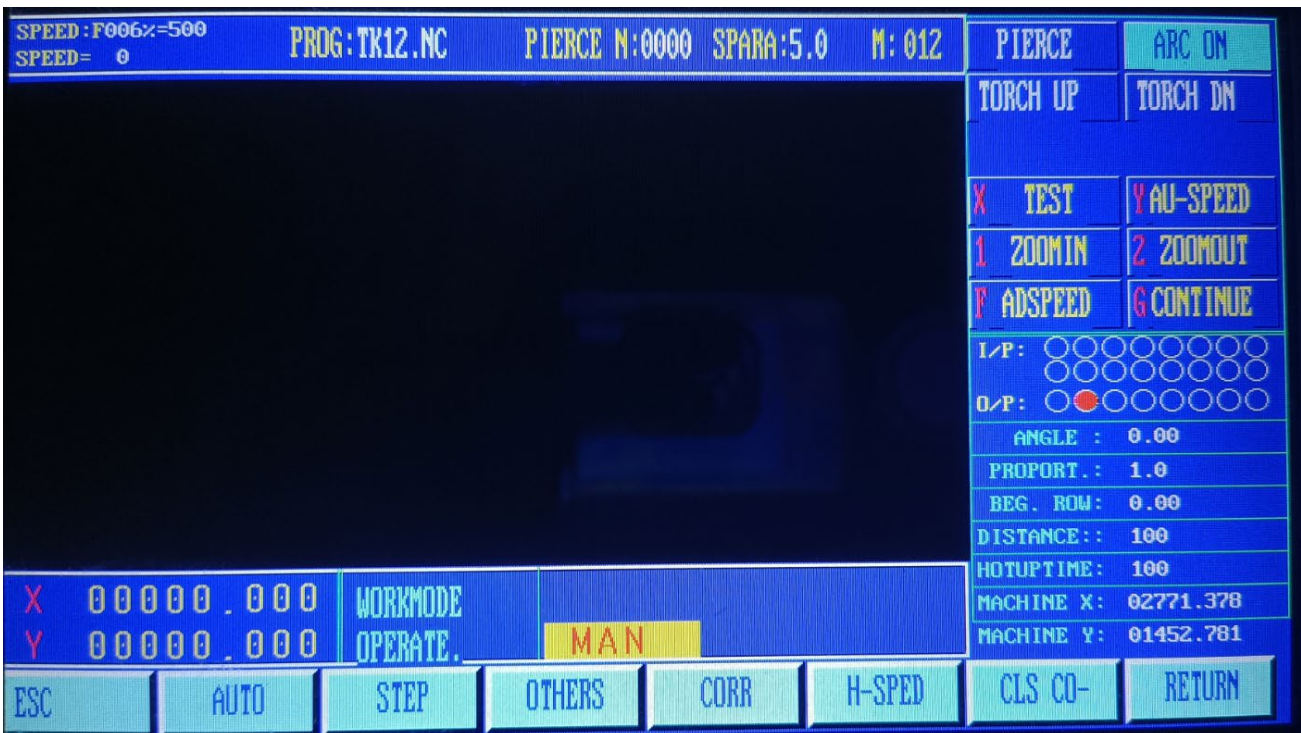
Service Support Spirit

4. MAN(Manual Mode)

In main menu, press [F2] to enter MAN (manual) window, as shown in figure below:



Manual mode window (flame cutting operation)



Manual mode window (plasma cutting operation)

Fig.4.1 Manual mode window

Service Support Spirit

4.1. Description of Manual Mode Window

The window display in manual mode is the same as it in automatic mode except that the multiplying factor is manually adjustable, which affects manual operation, return speed to reference point, move speed, etc. There are some special operations in manual mode as follows:

4.1.1 [↑], [↓], [←], [→] directional control keys and [G] Continue Key

Generally, if any of the four direction keys is pressed and held, appropriate axial movement will carry on until the key is released. However, if [G] Continue key is pressed and highlighted before any direction key is pressed, the cutting torch will keep moving even if the key is released. The movement will not stop until the direction key is pressed again. If two axial movements are required, let one axial movement start first and then press the direction key for the other axial movement. Two axial movement will go on simultaneously. In this case, the axial movement of cutting torch will stop if the corresponding direction key is pressed while the other axial movement will go on until the corresponding direction key is pressed too. [PAUSE] key will stop the movement as well.

4.1.2 【F1】 AUTO

The system is switched to automatic mode.

4.1.3 【F2】 STEP

If STEP function (highlighted) is selected, the system will prompt to input STEP INCREMENT: 0050.000 (previous input value by default). In STEP mode, if direction key is pressed once, the cutting torch will move an increment at the speed of the current maximum speed limit multiplied by multiplying factor.

4.1.4 【F4】 CORR

When the steel plate is placed on the material table, the steel plate may be inclined or the steel plate needs to be rotated by an angle due to other reasons, and then this function can be selected during processing.

Operation method: First move the torch to a certain position on the side of the steel plate, Press number 1 START;

Service Support Spirit

Press the arrow keys to move the torch to another position on the side of the steel plate; Press number 2 END.

Note: The two positions are as far apart as possible, so that the accuracy of the correction is higher.

The system will automatically calculate the angle of rotation. Then select the rotation function in automatic processing and enter the angle. After confirmation, the system will rotate the processing program according to the specified angle.

Note: The angle is positive counterclockwise.

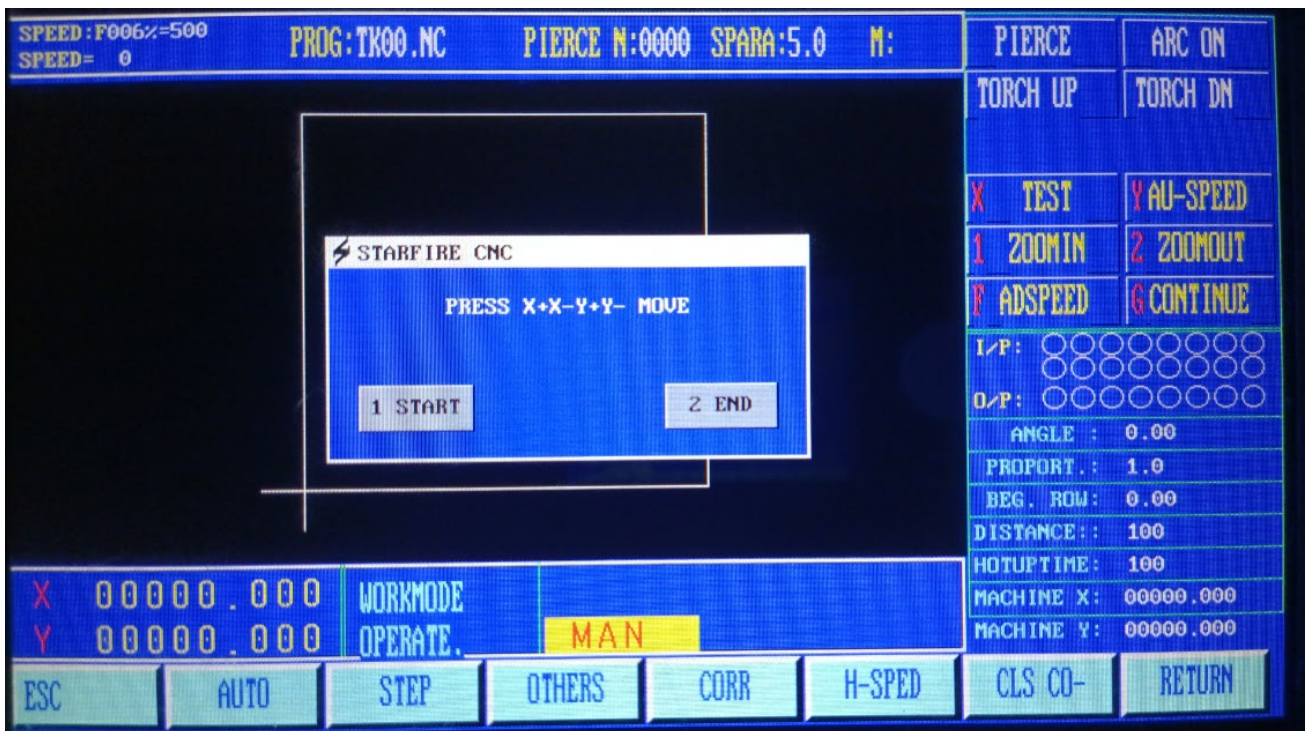


Fig.4.2 Steel plate calibration function

4.1.5 【F5】 H-SPED

Quick select manual ratio, press F5 high-speed (highlighted) ratio is 80%, then 10% at low speed.

4.1.6 【F6】 CLS-CO

Set the coordinate value to zero

4.1.7 【F8】 RETURN

Two choices appear after pressing F8:

Service Support Spirit

Choose the first one: RETURN TO REFERENCE;

【RETURN TO REFERENCE】 : This position is the coordinate reset position;

【RETURN TO MACHINE ZERO】 : It's a constant position, usually in the lower left corner of the machine

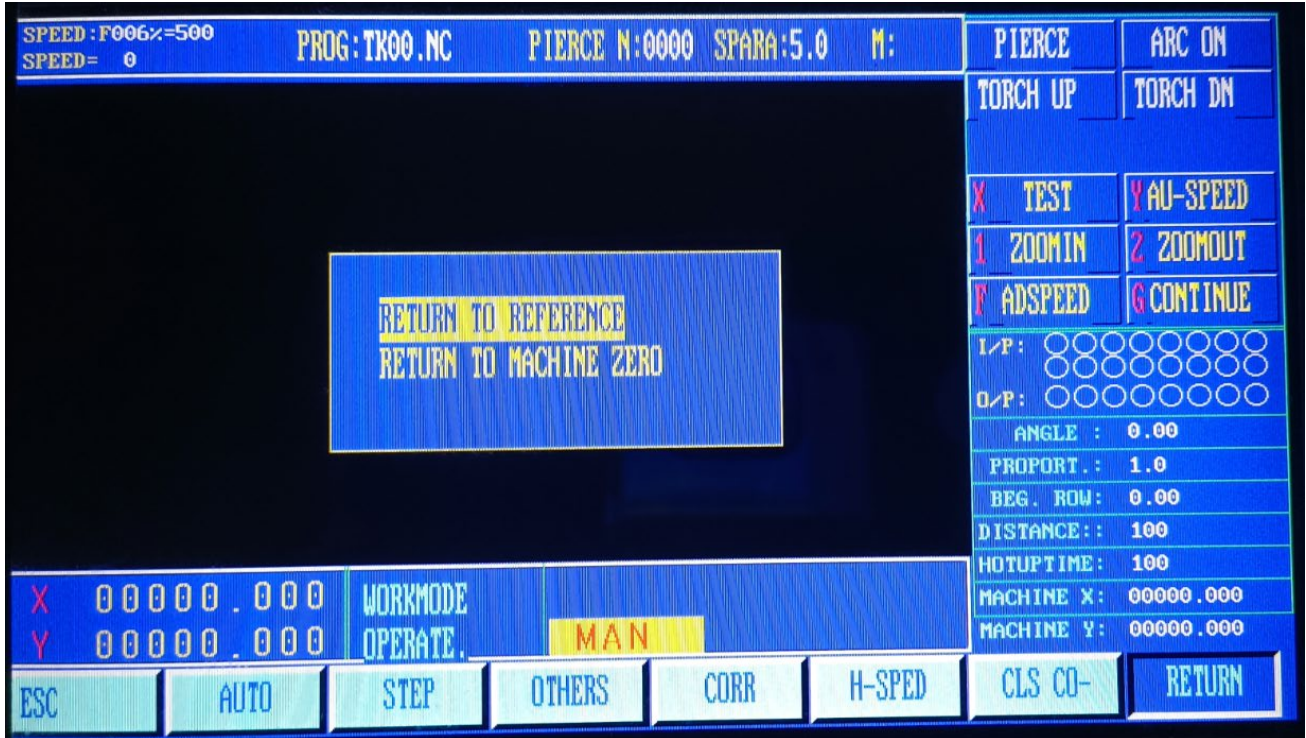


Fig.4.3 RETURN Interface

4.1.8 【F3】 OTHERS



Fig.4.4 OTHERS Interface

4.1.8.1 【F1】 MACH-0

【F1】 MACH-0: 【RETURN TO MACHINE ZERO】 : It's a constant position, usually in the lower left corner of the machine

4.1.8.2 【F2】 BRAKE

When this function is selected, during the machining process, each cut length (set breakage point. mm. See [Control] in [Parameter]), set a breakage point,

The length of the partition bridge (mm) (see [Control] in [Parameter]) is controlled by the operator to continue cutting. Note: This function has been maintained until Press this key to cancel the function.

4.1.8.3 【F3】 P-START and 【F4】 P-END

[F3] starting point, "F4" measuring the finish line, steel sheet processing calibration function

When the steel plate is placed on the material table, the steel plate may be inclined or the steel plate needs to be rotated by an angle due to other reasons, and then this function can be selected during processing.

Service Support Spirit

Operation method: First move the torch to a certain position on the side of the steel plate,

Press number 1 START;

Press the arrow keys to move the torch to another position on the side of the steel plate; Press number 2 END.

Note: The two positions are as far apart as possible, so that the accuracy of the correction is higher.

The system will automatically calculate the angle of rotation. Then select the rotation function in automatic processing and enter the angle. After confirmation, the system will rotate the processing program according to the specified angle.

Note: The angle is positive counterclockwise

4.1.8.4 【F5】 COORDI-

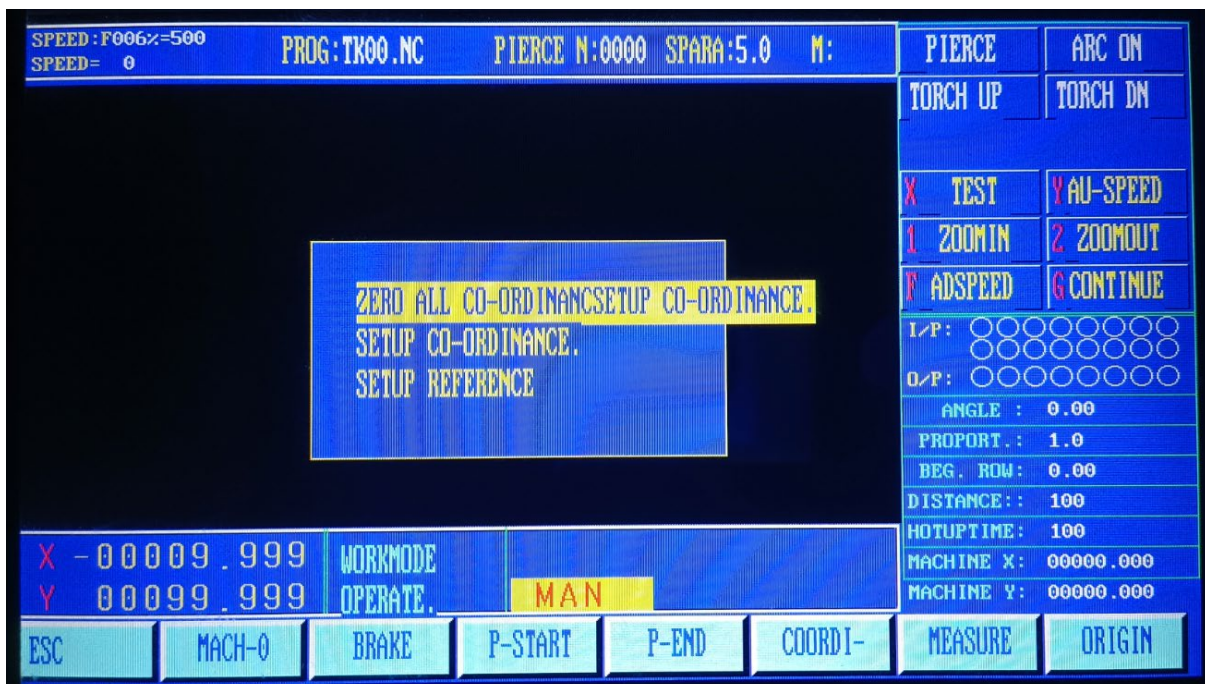
F5 Set coordinates

There are three options: click this button reset all coordinates, set the current coordinates, it is set to the reference point coordinates.

All coordinates are zero

Set the current coordinates

Set for the current coordinates



Service Support Spirit

4.1.8.5 【F6】 MEASURE and 【F7】 ORIGN

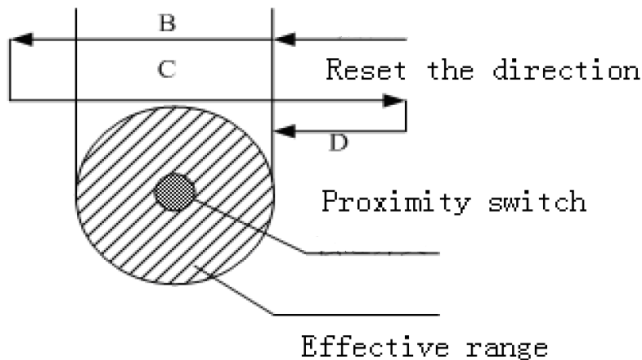
Machine back to the origin can be divided into "measuring the origin" and "reset" two parts.

Machine back to the origin, is the process of return machine mechanical zero point, back to zero biaxial running at the same time, independently back to zero. Reset (before back to zero):

(a) Mechanical origin should choose NPN hall proximity switch (normally open, the signal is low), and install in a suitable position.

(b) Set parameters (see parameter F3) reset speed - see "parameters" - "speed", reset when the feed rate, the unit is mm/min. reset direction - see [parameter] - [system], 0 - the shaft no longer, 1 - reverse, 1 - positive.

(c) Machine back to zero point process is as follows:



Section A: to reset speed fast forward.

Section B: slow down stop

Section C: reverse crawling to the effective area.

Section D: forward, to stop the valid area (the end).

(d) To measure the origin, to determine the mechanical origin of the work piece coordinate system. Methods:

Cutting torch will be moved to the reference point in the work piece coordinate system(a bit). set the current work piece coordinates.

[manual] - [supplementary] - [origin] measurement system to complete the mechanical origin, and the current coordinate values into the origin of the machine tool value, parameter Settings 【system】 option in the origin of the machine tool change accordingly.

(e) Reset - cutting back mechanical zero, and set up the work piece coordinate system.

Method: [manual] (or [automatic]) - auxiliary - reset, origin system to complete the mechanical action, and the origin of the machine tool is set to the current coordinates of the machine tool coordinate set to 0.

【note】 : If two axes in the parameter reset direction from 0, namely no reset action, execute reset after operation, the current value is equal to the origin of the machine tool. Machine tool coordinate is equal to zero, this is important, because the program limit is machine tool coordinate as a benchmark. Cutting torch will be moved to the first mechanical origin (the origin

Service Support Spirit

is not necessarily true), select reset function, can determine the current machine tool coordinate (zero), then according to the distance from the effective on both sides of the mechanical origin, fill out the positive/negative limit value of the software.

5. EDIT Mode

In the system main menu, press [F3] to enter EDIT menu as shown in the figure below:

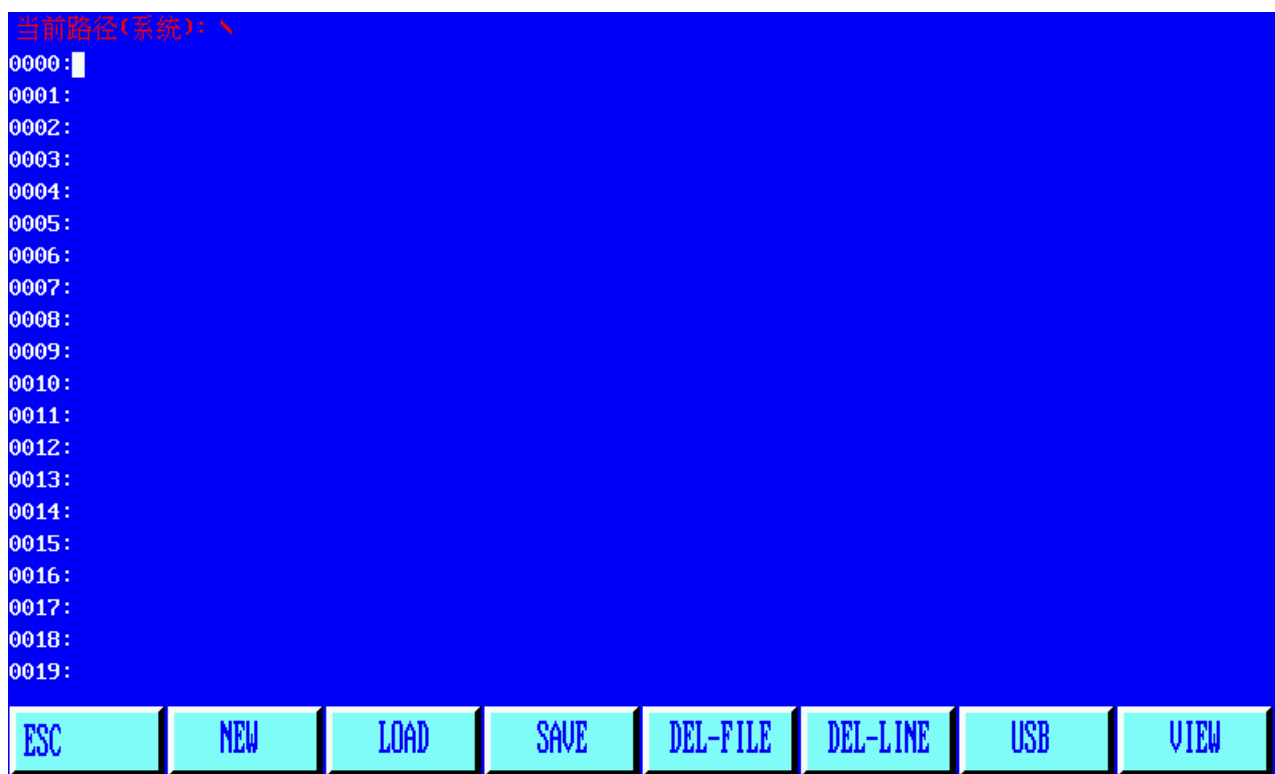


Fig. 5.1 EDIT menu window

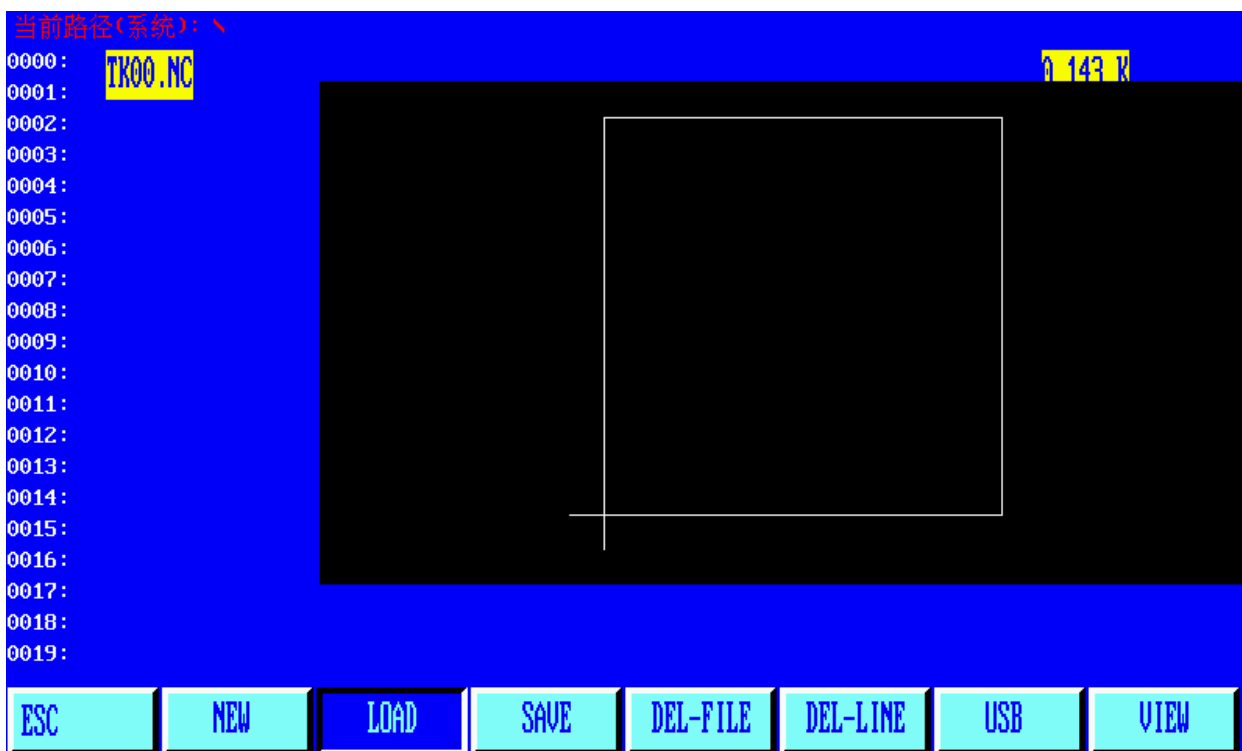
5.1. Description of EDIT Menu

5.1.1 【F1】 NEW

To establish a new program, i.e., clear the editing area of machining program and start to edit a new machining program.

5.1.2 【F2】 LOAD

To load a program. Specifically, it is used to load a program into the user program area. The system will tabularize the available program names and highlight the current program name. Move the cursor key to select different programs. If ENTER key is pressed, the highlighted program will be loaded into program editing area; if [ESC] key is pressed, the load function will be abandoned.



5.1.3 【F3】 SAVE

To save program. After a program is edited and ready to save, the system will prompt:
Enter program name: 1234.TXT

The system will display the current program name, which is modifiable. If ENTER key is pressed, the program in the editing area will be saved into the program area under the selected name; if [ESC] key is pressed, the save function will be abandoned.

Notes:

- 1) The program name and extension name should never exceed 12 characters.
- 2) For large-sized program above 200K, it is allowed to work along with USB disk instead of saving it.

5.1.4 【F4】 DELE

It is used to delete the program in the user program area.

5.1.5 【F5】 DELL

To delete a full program line in editing area and improve editing efficiency.

5.1.6 【F6】 USB-

To transmit programs. This system supports to transmit programs with USB disk. Press [F6] key to enter the lower level of menu, as shown below. :

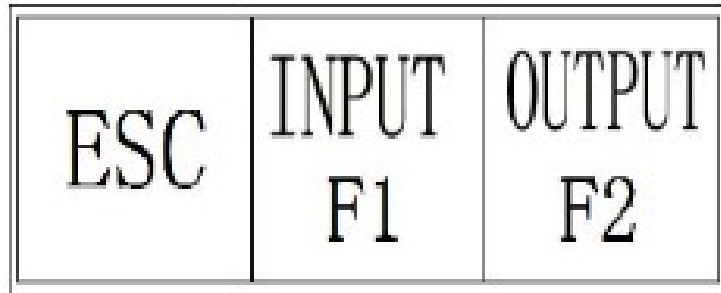


Fig. 5.2 Transmission menu of US USB disk

[F1] Input: to transmit program from USB disk to the machining program space.

[F2] output: to output the machining program to USB disk from the machining program space.

5.1.7 【F7】 VIEW

When you want to preview the currently selected program file processing by the graphics, can press F7 key.

6. Command System

6.1. Description of Programing Symbols

In CNC cutting operation, each step is taken according to a program. Each machining program is composed of several command segments, each command segment is composed of several function words and each function word is required to be a letter followed by parameter value.

Definitions of Function Words:

- N Serial number of command segment
- G Preparatory function
- M Auxiliary function
- T Tool function(refer to as flame width in this system)
- L Number of cycles, time delay
- X X-axis(diameter) absolute coordinate
- Y Y-axis absolute coordinate
- I The difference between the coordinate value of circle center and X-axis original value in arc cutting operation
- J The difference between the coordinate value of circle center and Y-axis original value in arc cutting operation
- R To designate arc radius
- H To designate arc chordal height
- A Auxiliary variable
- F To designate machining speed for G01, G02 and G03

Note 1: The following rules are valid in the definitions below:

X[U]n--- It is allowed to be either X or U and n stand for a value; 同理,

Y[V]n----It is allowed to be either Y or V, and n stand for a value;

PPn-----It is allowed to be a combination of any two axes or one axis at least.

Note 2: In command execution, the previous program has higher priority than the following one; for the same program, M, S and T command have higher priority than G command.

6.2. Coordinate System

This CNC system uses standard rectangular coordinate system, as shown in the figure below.

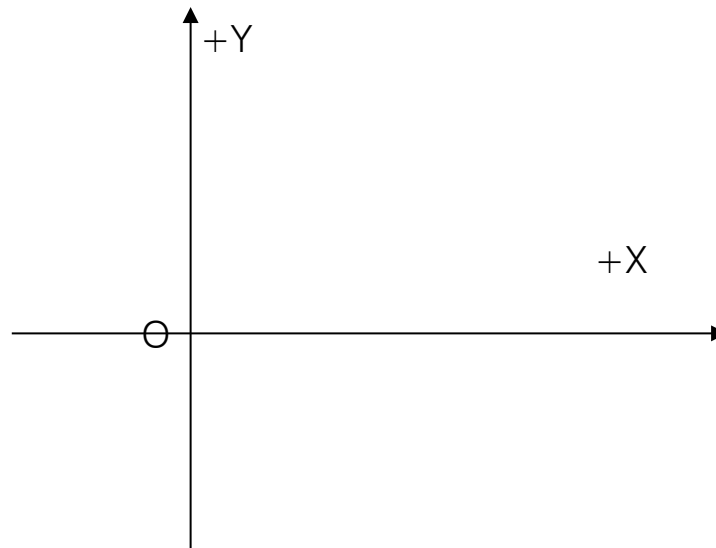


Fig. 6.1 Rectangular coordinate

6.3. G: Basic Preparatory Commands

◆ 1. G92: Reference Point Setting Command

When developing a program, it is required to place the coordinate value(absolute coordinate) of machining origin point(reference point) at the very beginning.

Format: **G92 Xn Yn**

If G92 is not followed with X/Y coordinate values, then the current X/Y coordinate will be taken as reference point. Generally, when the origin point of machine is used as **reference point** for positioning purpose, G92 will not be followed with X/Z values.

◆ 2. G90/G91: Absolute/Relative Coordinate Commands

When using G90, X/Y represent coordinate values and U/V represent the values relative to the current point; when using G91, X/Y and U/V all represent the values relative to the current point.

Format: **G90**

Format: **G91**

Exemple 1: **G92 X0 Y0**

G91 // Relative coordinate system

G00 X100 Y100 // Quickly reach point(100, 100), equivalent to

G00/U100/V100

G01 X500 Y100 // Machining to point(600,200) along a straight line,

equivalent **G01/U500/V100**

Exemple 2: **G92 X0 Y0**

Service Support Spirit

G90 // Absolute coordinate system, by default
 G00 X100 Y100 // Quickly reach point(100, 100)
 G01 X600 Y200 // Machining to point(600,200) along a straight line

◆ 3.G20/G21: English/Metric Commands

G20 stands for English system. Following G20, the values of X, Y, I, J, R, U, V, H and F are all in English unit;

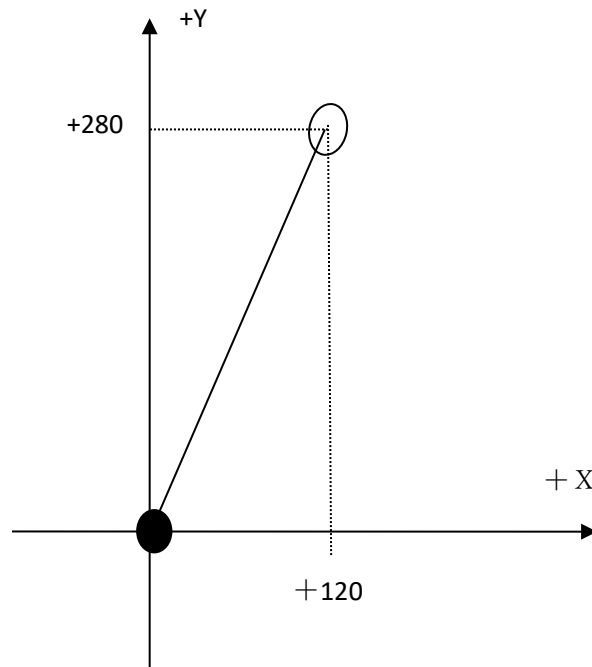
G21 stands for metric system. Following G21, the values of X, Y, I, J, R, U, V, H and F are all in metric unit;

Format: G20

Format: G21

E.g.: G92 X0 Y0
 G00 X120 Y280
 (or G00 U120 V280)
 M02

● Current Torch Position
 ○ Expected Torch Position



◆ 4. G00 Quick Move

This command can move the cutting torch quickly to a designated position. In the event of two axial movements, the cutting torch will move along a straight line from the origin point to the end point at the speed of the maximum speed limit multiplied by multiplying factor. G00 movement is subject to speed multiplying factor.

Format: G00 X[U]n Y[V]n
 or G00 PPn

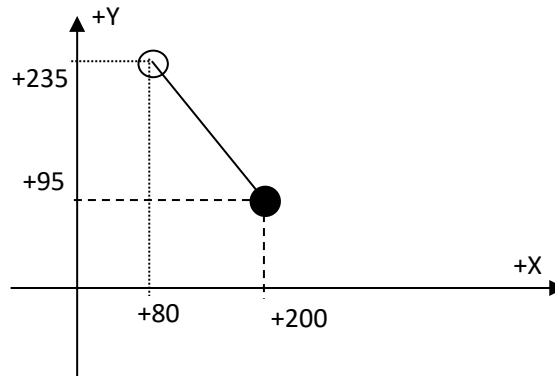
◆ 5. G01: Linear Cutting Command

Service Support Spirit

This command makes the torch cut to a designated position along a straight line. As machining movement command, it allows uniaxial or biaxial linear interpolation movement. The feed speed is designated by F command.

Format: G01 X[U]n Z[W]n [Fn]
or G01 PPn [Fn]

EG: G92 X0 Y0
G00 X200 Y95
G01 X80 Y235
(OR G01 U-120 V145)

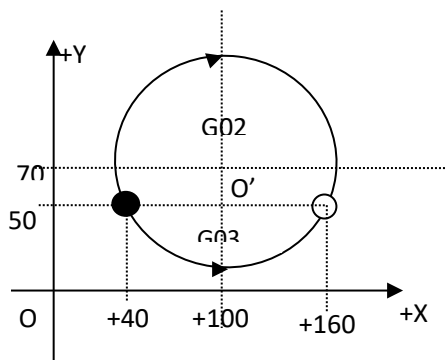


- Current Torch Position
- Expected Torch Position

◆ 6. G02/G03: Circular Cutting Commands

This command is used for circular interpolation, including clockwise circular cutting command G02 and counter clockwise circular cutting command G03. See the figure below for the setting of directions.

Format: G02[03] X[U]n Y[V]n In Jn [Fn] or G02[03] X[U]n Y[V]n Rn [Fn]
G02[03]PPn In Kn [Fn] or G02[03] PPn Rn [Fn]



EG(G02):

```
G92 X0 Y0
G00 X40 Y50
G02 X160 V0 I60 J20
G28
M02
```

- Current Torch Position
- Expected Torch Position

EG (G03) :

```
G92 X0 Y0
G00 X40 Y50
```

Notes:

I and J are the increased values of circle center relative to the origin point in X-axis and Y-axis respectively (distance between the circle center and the origin point).

R is the radius of circle (R is a positive value and can be used to represent radius where circular arc $\leq 180^\circ$).

Alternatively, if I and J are designated, R will be not necessary; if R is designated, I and J will be not necessary.

◆ 7. G04: Pause/Delay Command

This command is used for settling time delay. When this command is executed in a program, the time delay defined by command L in seconds will be given.

Format: G04 Ln

Example: G04 L2.4 (To delay 2.4 seconds)

In the execution process of G04 command, if [START] key is pressed, the time delay will be disabled and the subsequent program will be executed continually. If [Exit] key is pressed, the execution of the current program will be aborted.

◆ 8. G26/G27/G28: Return Reference Point Commands

These commands make the cutting torch return to the reference point automatically.

Format: G26

Return to the reference point along X axis.

Service Support Spirit

G27 Return to the reference point along Y axis.

G28 Return to the reference point simultaneously along both X and Y axes.

Example: G28(simultaneously return to reference point, equivalent to execute G00 command)

◆ 9. G22/G80: Repetitive Commands

These commands are used for execution of program loop. G22 is used to designate the start of loop and number of cycles L. G80 is the end mark of loop. These commands may be used for nested loop not more than 5 levels. G22 and the nearest G80 form a loop.

Format: G22 Ln_ (L to designate number of cycles)

Loop

G80 (End mark of loop)

Example:

```

N000 G92 X100 Y100
      N001 G00 X60 Y80
      N002 G22 L5           - Level 1 loop starts.
      N003 G00 V50 U-25
      N004 G22 L5           - Level 2 loop starts.
      N005 G01 U5 V-10
      N006 G80             - Level 2 loop ends.
      N007 G80             - Level 1 loop ends.
      N008 G28
      N009 M02

```

◆ 10. G40/G41/G42: Cutter Radius Compensation Commands

Format: G41 (or G42) Rn

Program segment to be compensated.

G40

Notes:

G41 is to compensate a half diameter of flame leftwards along the machining path.

G42 is to compensate a half diameter of flame rightwards along the machining path.

G40 represents end of offset.

Since cutter compensation is made automatically, there must be G00 Quick Move command before G41 or G42 commands so as to assure the position of cutting torch is corrected. Behind

Service Support Spirit

G40 the end mark of cutter compensation, a G00 command is necessary to adjust the position back.

6.4. M Auxiliary Commands

M00 **Program suspension** command. The program execution will be suspended at this command until [START] key is pressed.

M02 **Program end** command. The program will be pended at this command.

M30 The same as M02.

M10/M11 **Acetylene(GAS)** valve switch commands, M10 to open and M11 to close the valve.

M12/M13 **Cutting oxygen valve** switch commands, M12 to open and M13 to close the valve.

M14/M15 **Cutting torch lift** switch commands, M14 to turn on and M15 to turn off.

M16/M17 **Cutting torch lower** switch commands, M16 to turn on and M17 to turn off.

M24/M25 Reserved switch commands, M24 to turn on and M25 to turn off.

M20/M21 Ignition switch commands, M20 to turn on and M21 to turn off.

M07 **Fixed loop of piercing(M07 allows to relocate the cutting torch, but impossible to return)**

M08 **Fixed loop of cutting stop**

The operation sequence of M07 in flame cutting operation is as follows:

Open acetylene(gas) valve and ignite the torch if the valve is not open.

Lower the cutting torch(for time delay of torch lowering, see M71 command);

Open the oxygen preheating valve and count preheating time delay. If the preheating time is insufficient, it will be prolonged to 150 seconds automatically by pressing [PAUSE] key. If preheating is done in the midway, press [START] key to abort the time delay. The new preheating time will be automatically saved in the parameter of preheating time delay.

Lift the cutting torch(and count the time delay of lifting torch, M72);

Open the cutting oxygen valve(M12), count the time delay and lower the cutting torch(count the time delay of lowering torch , M73).

Switch on height controller(M38) and run the subsequent program.

The operation sequence of M07 in plasma cutting operation is as follows:

Lower the cutting torch(and count the time delay of lowering torch, see M71);

If hole positioning option is enabled(refer to SETUP), the cutting torch will move down until hit the lower limit switch. Then the cutting torch will move up until the hole positioning time delay is up.

Service Support Spirit

Turn on the arc ignition switch.

Test the signal of "Arc Voltage Successful", which will not be tested if the arc voltage test option is taken to 0(not test). When arc ignition is successful, the piercing time delay will start to count in seconds.

Switch on height controller (M38) and run the subsequent program.

M08: fixed loop of cutting stop

The operation sequence of M08 in flame cutting operation is as follows:

- 1.Close the cutting oxygen valve(M13).
- 2.Turn off the height controller(M39).
- 3.Lift the cutting torch(M70).

The operation sequence of M08 in plasma cutting operation is as follows:

Turn off the arc voltage switch.

Turn off the height controller(M39).

Lift the cutting torch(M70).

M50: Piercing action

Lift the cutting torch(M72). There is no this step in plasma cutting operation.

Open the cutting oxygen valve(M12) or turn on plasma arc ignition and check for the signal of "Arc Voltage Successful".

Lower the cutting torch(M73). There is no this step in plasma cutting operation. Switch on height controller(M38).

M52: Fixed loop of ignition

Operation sequence: open acetylene(gas) valve(M10); turn on the high voltage ignition(M20); count the ignition time delay and turn off the high voltage ignition(M21).

M70: Fixed loop of lifting torch

Used at the beginning of a program and at the end of cutting section, this command is to raise the cutting torch for possibility to move quickly to next cutting position. The operation sequence is as follows: turn on the torch lifting switch(M14); count the time delay of lifting torch(refer to 7.3 for flame parameter); turn off the torch lifting switch(M15).

Service Support Spirit

M71: Fixed loop of lowering torch

This command is used before piercing operation, opposite to M70 in function and little bit smaller in value. Because of gravity, the lowering action is faster than lifting action. Operation sequence: turn on the torch lowering switch(M16), count the time delay of lowering torch(refer to 7.3 for flame parameter) and turn off the torch lowering switch(M17).

M72: Loop of lifting the piercing torch

At the end of preheating, this command is used to lift the cutting torch somewhat to prevent the torch nozzle from blocked by splashing slag. The operation sequence is as follows: turn on the torch lifting switch(M14); count the time delay of lifting the piercing torch(refer to 7.3 for flame parameter); turn off the torch lifting switch(M15).

M73: Loop of lowering the piercing torch

At the end of preheating after M72 is executed and the cutting oxygen valve is opened, this command is used to move the torch to the cutting position, opposite to M72 in function and little bit smaller in value. Because of gravity, the lowering action is faster than lifting action. Operation sequence: turn on the torch lowering switch(M16), count the time delay of lowering the piercing torch(refer to 7.3 for flame parameter) and turn off the torch lowering switch(M17).

M75: Time delay of torch positioning

When positioning the plasma gun, lower the torch(M16). It will not stop until the torch hits the lower limit switch (refer to input port 8 XXW)(M17). Turn on the torch lifting switch (M14). After count the time delay of torch positioning (refer to 7.4 for plasma parameter), the lifting torch will stop(M15).

M80: Master switch

All the output ports will be turned off if the command of M80 is executed.

7. SETUP (Parameter Setting)

In the system main menu, press [F4] to enter parameter setting window as shown below:



Fig. 7.1 parameter interface

7.1. Parameter Description

Speed parameters

Starting speeds of every axis, adjustment time and maximum speed limit.

System parameters

Gear ratio, mechanical origin, reference point, reverse backlash, line bias and soft positive/negative limit switch in every axis

Flame cutting parameters

Ignition delay, preheating delay, torch lifting/lowering delay, lifting/lowering of piercing torch, piercing delay, etc.

Plasma parameters

Service Support Spirit

Torch positioning delay, M commands for arc ignition, M commands for arc breaking, arc voltage test option, positioning test option and piercing delay.

Control parameters

Flame/plasma mode selection, machining speed limit, MOVE HOLE option, metric/English selection, etc.

Save

Save the modified parameters into parameters space.

Continuous pressing S key, optional external manual control key and manual enable/disable option.

Notes:

When selecting the above parameters, it is required to save the change separately by pressing [F8] key for validation.

If the password of "1928" is entered in the main window of SETUP, the saving menu of [F8] will turn to factory setting. In this mode, every modification to parameter will be saved into the factory setting parameters and the active user parameters. When initializing parameters, it is suggested to set the factory parameters as active parameters; otherwise, the modifications will be validated to the active user parameters.

7.2. SETUP(parameter setting)

7.2.1 Speed parameters

In the SETUP, press [F1] key to enter speed parameters setting window, as shown in Fig. 7.2

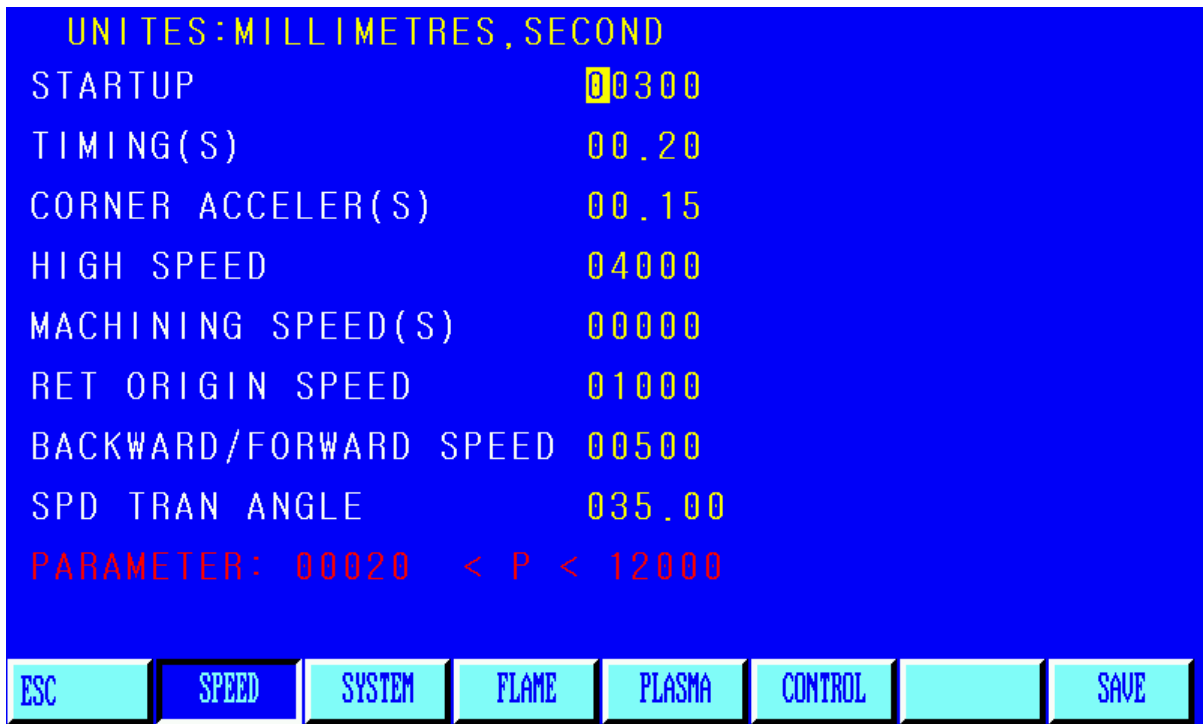


Fig. 7.2 Speed interface

Speed parameters include--

STARTUP---- System X, Y axes at the time of starting and stopping the speed (unit: mm or inches per/minute, See control parameters, the same below).

TIMING(S)---- System by starting speed adjustment to the highest speed (the speed) during the process of the time it takes. Unit: seconds.

CORNER ACCELER(S)---- In the process of add/deceleration, linear acceleration time is, usually slightly smaller than adjusting time (about nine over ten). Big machine this value accounted for the proportion of smaller.

HIGH SPEED---- The top speed for manual and execute commands G00 runtime (unit: MM or Inch/Min)..

MACHINING SPEED(S)----The top machining speed in flame/plasma machining operation, in mm/min. or inch/min.

RET ORIGIN SPEED ---- When the running direction changes between program segments and changing angle exceeds this angle, the system will decelerate to the starting speed around the corner. This value is normally set smaller when the system is comparatively heavier. In addition, such considerations as machining speed and machine vibration should be taken as well, smaller value for worse vibration.

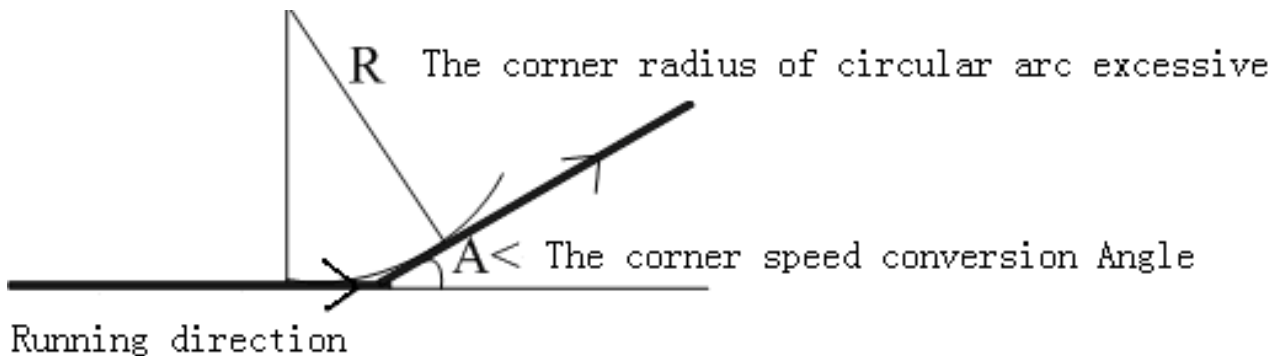
Service Support Spirit

BACKWARD/FORWARD SPEED — If this option is enabled (value:1), the system will not decelerate through a circular transition when the running direction change of system between program segments is within the conversion angle of corner speed, as shown in the figure below.

SPD TRAN ANGLE — When the change of the running direction between blocks exceeds this angle, when the system is at the corner, it will decelerate to the starting speed.

Generally, when the machine is heavier, select a smaller value. The adjustment can be determined according to the processing speed and the vibration of the machine tool.

If the vibration is large, select a small value;



CORNER RADIUS — See ARC RADIUS

ARC SPEED — Speed limit only for arc

ARC RADIUS — Arc radius setting, Used in conjunction with parameter arc speed limit, When the arc radius is less than this value, it runs at the arc speed limit; when it is set to 0, the arc speed limit function is invalid;

7.2.2 System parameters

In the SETUP submenu, press [F2] key to enter system parameters setting window, as shown in Fig.7.3.

Service Support Spirit

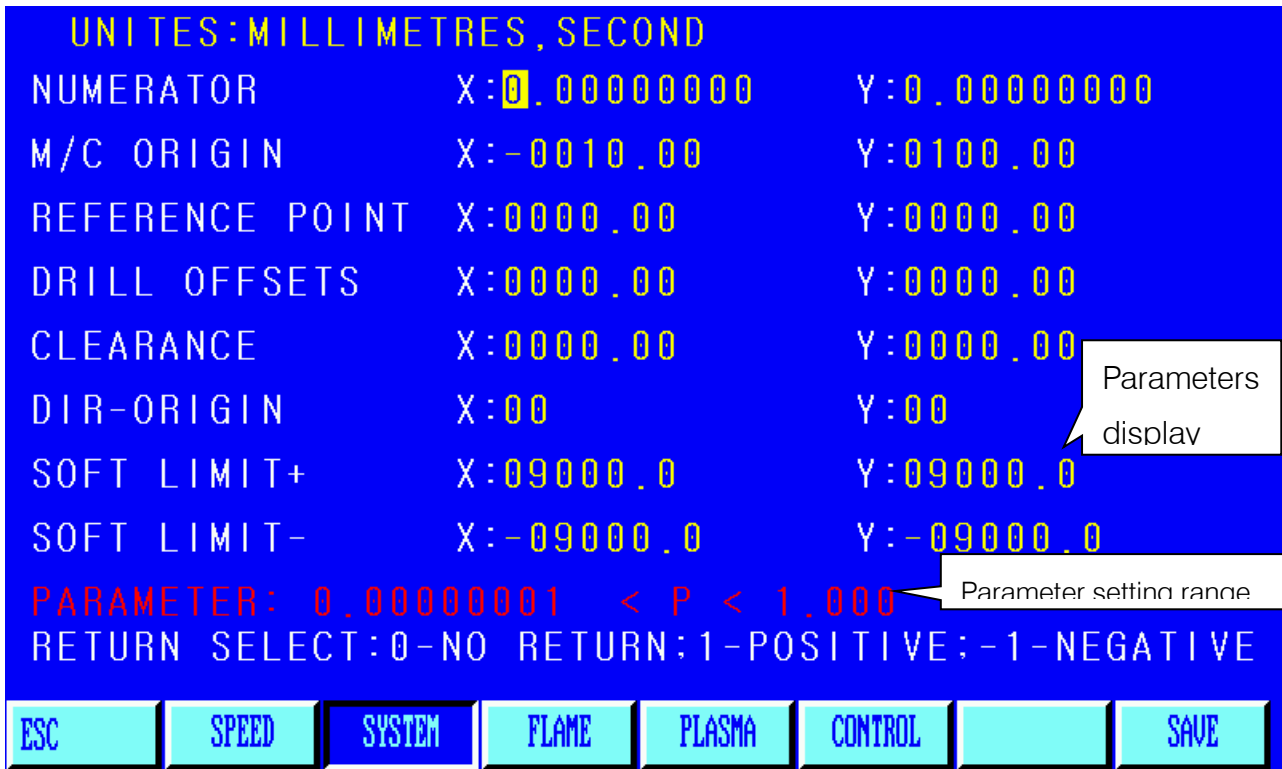


Fig. 7.3 System parameters setting

Numerator—The numerator/denominator ratio of electronic gear is pulse equivalency in micron.

Numerator < 65535, denominator < 65535.

The computation method (Adjustment method) of electronic gear is as follows:

- (1) Firstly, assume an electronic gear ratio, e., g., 8:1.
- (2) Control the machine to move a reference distance(the longer, the more accurate). Measure the actual travel distance and take the measured value into the following formula:

【 The theoretical distance 】

【 The actual distance 】

$$\frac{\text{【 The theoretical distance 】}}{\text{【 The actual distance 】}} \times \text{System pulse value} = \text{new pulse value}$$

For example: the System Pulse value is 125, The input theoretical value is 1000; The actual distance is 1500

【 The theoretical distance:1000 】

【 The actual distance:1500 】

$$\frac{\text{【 The theoretical distance:1000 】}}{\text{【 The actual distance:1500 】}} \times \text{System pulse value: 125} = \text{New pulse:83.333}$$

M/C ORIGIN=Mechanical origin — A special point on the machine set with proximity switch. If mechanical origin is not to be used, set it to zero. Unit: mm(or inch)

Service Support Spirit

Reference point — It is defined as the machining origin of program and established automatically when a program is run(G92). Unit: mm(or inch).

DEILL OFFSETS---The offset distance of the punching or marking gun relative to the cutting gun.

CLEARANCE--- Due to reverse mechanical backlash, the system will compensate the backlash at direction change. The backlash value is obtained by actual measurement in mm or inch. **Generally, reverse backlash is not recommended.**

DIR-ORIGIN— Return to the machine zero running direction, 0-no reset 1-positive direction reset 2-negative direction reset..

Soft LIMIT+ /SOFT LIMIT- —The system will give an alarm when program coordinate exceeds the set soft positive/negative limit. If it will not be used, the parameter should be set higher than actual values in mm or inch.

7.2.3 Flame Cutting Parameters

In the SETUP submenu, press [F3] key to enter flame parameters setting window, as shown in Fig. 7.4.

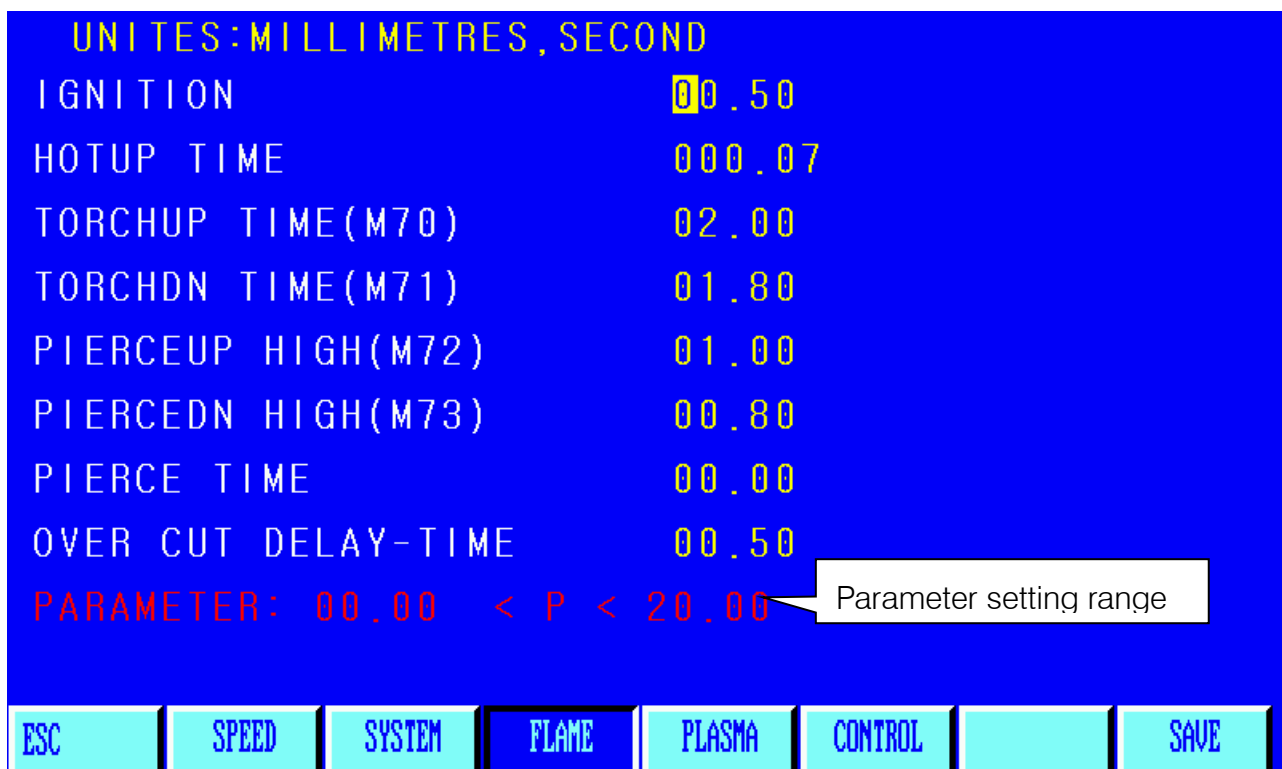


Fig. 7.4 Flame parameters setting

Service Support Spirit

IGNITION—It is the time delay when turn on the high voltage ignition switch if M20 is executed in flame cutting operation.

HOTUP TIME—It is the preheating time for piercing operation in second. If the preheating time is insufficient, it will be prolonged to 150 seconds automatically by pressing [PAUSE] key. If preheating is done in the midway, Press [START] key to abort the time delay. The new preheating time will be automatically saved in the parameter of preheating delay.

TORCHUP TIME(M70)—**Torch rising time** The time delay when M70 command is executed in second. Refer to 6.4 for M auxiliary commands.

TORCHDN TIME(M71)—**Torch down time**, The time delay when M71 command is executed, in second. Refer to 6.4 for M auxiliary commands.

PIERCEUP HIGH(M72)—**Torch rising time after pierce** The time delay when M72 command is executed, in second. Refer to 6.4 for M auxiliary commands.

PIERCEDN HIGH(M73)—**Torch falling time after pierce** The time delay when M73 command is executed, in second. Refer to 6.4 for M auxiliary commands.

PIERCE TIME—Lowering delay of cutting torch when M07 is executed in flame piercing operation after cutting oxygen valve is opened.

OVER CUT DELAY-TIME---- Automatically disconnect oxygen delay; After turning off the cutting oxygen, due to the residual pressure in the tube, after this delay, proceed to the next step to improve the cutting surface finish.

LINE DELAY-TIME--- After the program line is in place, after this delay time, run the next line Move to the next piercing point after completing a cut.

SELECT HI HOTUP M24 (0/1)--- When the thick plate is perforated, the high preheating aid is required. When this function is selected, M24 is turned on during preheating, and M24 is closed after preheating. Note: When this option is not selected, M24 (preheating) will not be turned off during the entire processing. 1 is to choose, 0 is not to choose;

GAS ON M ORDER--- Open the output port address used by the gas solenoid valve (see diagnosis function),

Service Support Spirit

LEAVE PREHEATS ON (0/1)--- After processing, the system does not close the output-select; 0 is closed, 1 is not closed;

7.2.4 Plasma Parameters Setting

In the SETUP submenu, press [F4] key to enter plasma parameters setting window, as shown in Fig. 7.5.

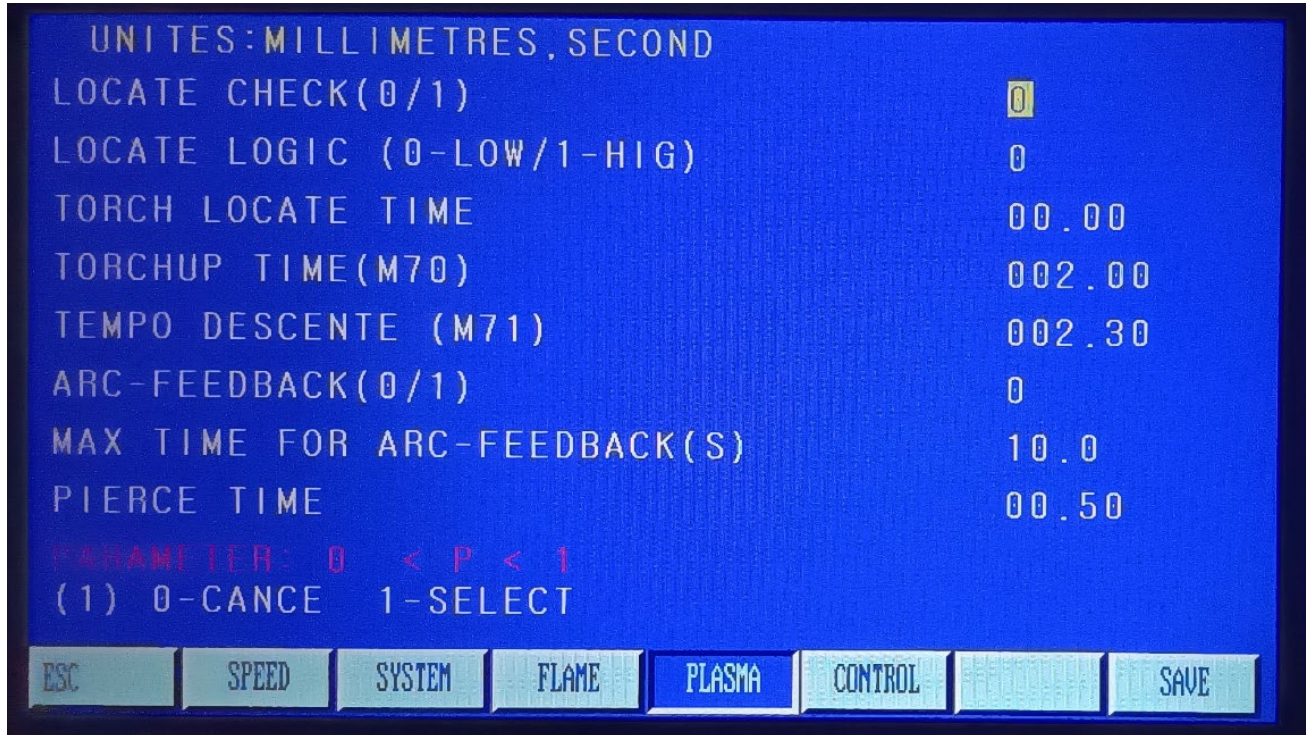


Fig.7.5 Plasma Parameters Setting

LOCATE CHECK (0/1)---When the initial positioning is required by the system, select this.

LOCATE LOGIC (0-LOW/1-HIG)---When the positioning switch is normally open (high effective), select 0, and normally closed (low effective) select 1.

TORCH LOCATE TIME---When the plasma torch is positioned, the torch will down and when it hits the lower limit, the torch will stop. Then, the torch rises, through the torch positioning delay, the torch stops rising (see M75 instruction), unit: second;

TORCHUP TIME(M70)---**Torch rising time.** Delay time when executing M70 instruction (see 6.4M auxiliary instruction), unit: second. Note: For convenience, in plasma and flame mode, the torch up/down delay is separate;

TEMPO DESCENTE(M71)---**Torch falling time.** Delay time when executing M71 instruction (see 6.4M auxiliary instruction), same as above, unit: second;

ARC-FEEDBACK(0/1)---When plasma operation, whether arc voltage feedback.

Setting 1: Need arc voltage feedback function; Need to detect arc voltage feedback when starting an arc, The arc voltage feedback needs to be monitored during operation. When the

Service Support Spirit

arc voltage feedback is cut off, the system will stop processing and prompt. Generally, arc voltage feedback function is required for thick plate processing.

Setting 0: Do not use arc voltage feedback function, The whole cutting process has no arc voltage feedback function, Generally, the arc voltage detection function is not required for thin plate processing.

MAX TIME FOR ARC-FEEDBACK(S)---Select the arc voltage feedback function. During piercing, if the piercing time exceeds this "maximum delay", and the arc starting success signal has not been received, the system will report an error, and the current work will be suspended for processing. After troubleshooting, press the start button again to restart piercing.

PIERCE TIME---When the arc is started successfully, the system cuts and runs after the pierce time;

FIRST PIERCE TIME---For the first piercing, it takes a long time because the plasma is cooler;

OFF DELAY---Arc breaking delay, the delay time to turn off the arc signal after cutting;

DIS.COR. OFF AHCS--- When the program segment (corner) is converted, the speed may change (the arc voltage will also change, causing the cutting gun to hit the steel plate), Therefore, when the cutting gun is at this distance from the end point, the system will automatically close the height adjustment control, unit/mm;

SPEED OFF AHCS---When the processing speed is lower than this value, turn off the automatic height adjustment, set to 0, the function is invalid;

ANTICIP OFF ARC(MM)--- The processing is usually a closed curve; when the processing is completed, the start point and the end point overlap, which will often cause over burning and affect the finish. After choosing this distance: When the distance from the cutting torch to the end point is this distance, the arc voltage and height adjustment are automatically closed;

AHCS AUTO-SIG DELAY(S)--- Because the arc voltage is not very stable at the start of cutting, after this delay, the automatic height adjustment is turned on after the start of cutting.

ARC ON M ORDER--- Set arc start output port

ARC OFF M ORDER--- Set arc off output port

7.2.5 Control Parameters Setting

In the SETUP submenu, press [F5] key to enter control parameters setting window, as shown in Fig. 7.6

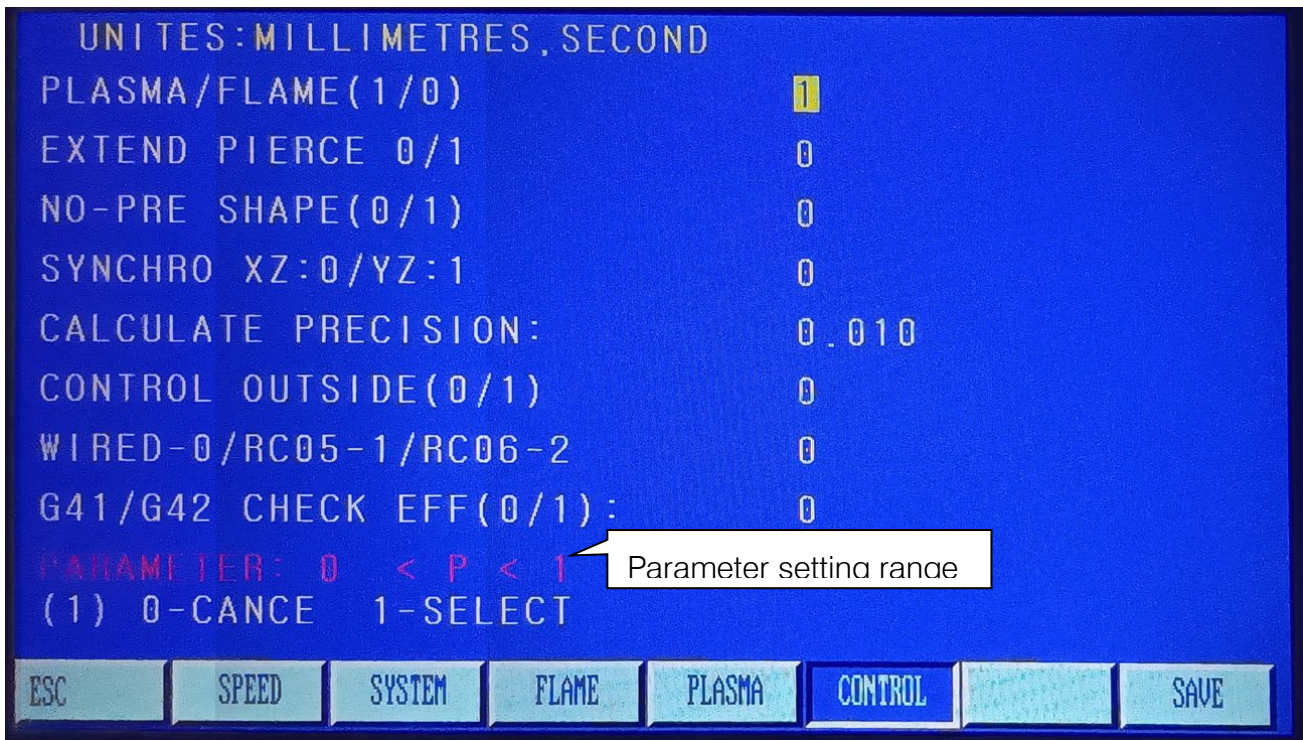


Fig. 7.6 Control parameters menu

PLASMA/FLAME(1/0)— Set to 0 for flame processing and 1 for plasma processing;

EXTEND PIERCE 0/1—0 means not to select the edge piercing function, 1 means to select the edge piercing function;

NO-PRE SHAPE (0/1)— There is no need to display graphics during processing, so set this to 1.

SYNCHRO XZ: 0/YZ: 1—Because the system can control 3 motion axes, For machines requiring bilateral drive, whether the third axis is shared with the X axis or the Y axis is set this parameter. When 0 is selected, Z-axis and X-axis are the same axis; When 1, Z-axis and Y-axis are the same axis.

CALCULATE PRECISION—Calculation control accuracy.

CONTROL OUTSIDE (0/1)— If you use an external hand control box, select 1, otherwise select 0;

WIRED-0/RC05-1/RC06-2—For the external hand control box, select 0 for wired remote control and 1 for wireless RC05 remote control and 2 for wireless RC06 remote control;

G41/G42 CHECK EFF(0/1)— There is a check on the correctness of the compensation (G41/G42) in the system. Set this parameter to 0, when there is a problem with the kerf compensation value, an error line will be prompted, but no alarm, the program can continue to run.

Service Support Spirit

INITIAL CLEAR COOR (0/1)— Whether to automatically clear the coordinates before processing; Select 1, before processing, automatically clear the coordinates to 0;

SELECT CYLINDER UP/DOWN (0/1)— Pneumatic lifting torch, Used in thin plate processing; 0 is not to select pneumatic, 1 is to select pneumatic;

SELECT AUTO-REFERENCE (0/1)— Automatically return parameters after processing-select. Select 1 to automatically return to the reference point after processing.

LIMIT POSI EFFECT (0/1)— External electronic limit is valid. Set to 0 = external electronic limit is valid, Set to 1 = external electronic limit is invalid.

SOFTWARE LIMIT EFFECT (0/1)— **Software limit is valid.** Set to 0 = CNC-software limit is invalid, Set to 1 = CNC-software limit is valid.

CHECK COLLIDE (0/1)—Collision torch detection is effective. When the machine has an anti-collision device, this parameter needs to be set; if THC can control the anti-collision signal, this parameter does not need to be set; Collision cutting torch detection valid=1, In case of collision, the system will immediately pause (and report collision warning);

COLLIDE-LOCATE SAME PORT (0/1)— Whether the torch collision detection and positioning detection are the same input port-select. 0 is the same, 1 is different;

COLLIDE PAUSE (0)/TORCHUP (1)— System pause/raise after torch hit. Set= 0, processing will be suspended after the torch is hit, Set=1, Automatically raise the torch after hitting the torch, and the processing continues.

PAUSE UP TORCH (0/1)— Raise the torch after pause (0/1) select; Set to 1, after the pause, there will be a raising torch action.

CUTTING LINE (BREAK.MM)— Cutting length (set breaking point. mm) —— When the bridge breaking function is selected during processing, each time this length is cut, the system will automatically set a breaking point;

BRIDGE CUTOFF LINE (MM)— Length of broken bridge (mm) —— After setting the broken bridge point, After running this distance; Restart cutting;

Service Support Spirit

DISP CO NAME (0/1)— Start screen, display company's text information, Set to 1 to display text information, 0 to display picture information

SELE PLOT KERF LINE (0/1)— Whether to display the coordinate track after kerf compensation. 0 is not displayed, 1 is displayed;

Note: Please do not modify the parameters when you do not understand the specific application of the parameters!

8. LIBMINIT (Library of Patterns)

8.1. LIBMINIT Setting

In addition to customized pattern module, the feature allows user to input the intended dimensions and get the expected work piece.

When entering parameter, the control system will conduct general check to the physical dimensions and prompt warning message if any error is detected.

Notes:

1. It is impossible for the control system to find all the error parameters; therefore, please enter correct dimensions for parameters as much as possible.
2. When the parameters are entered, the control system will draw the pattern based on the entered parameters, which is very helpful to graphic check.

In the system main menu, press [F6] to enter the LIBMINIT(Library of Patterns).

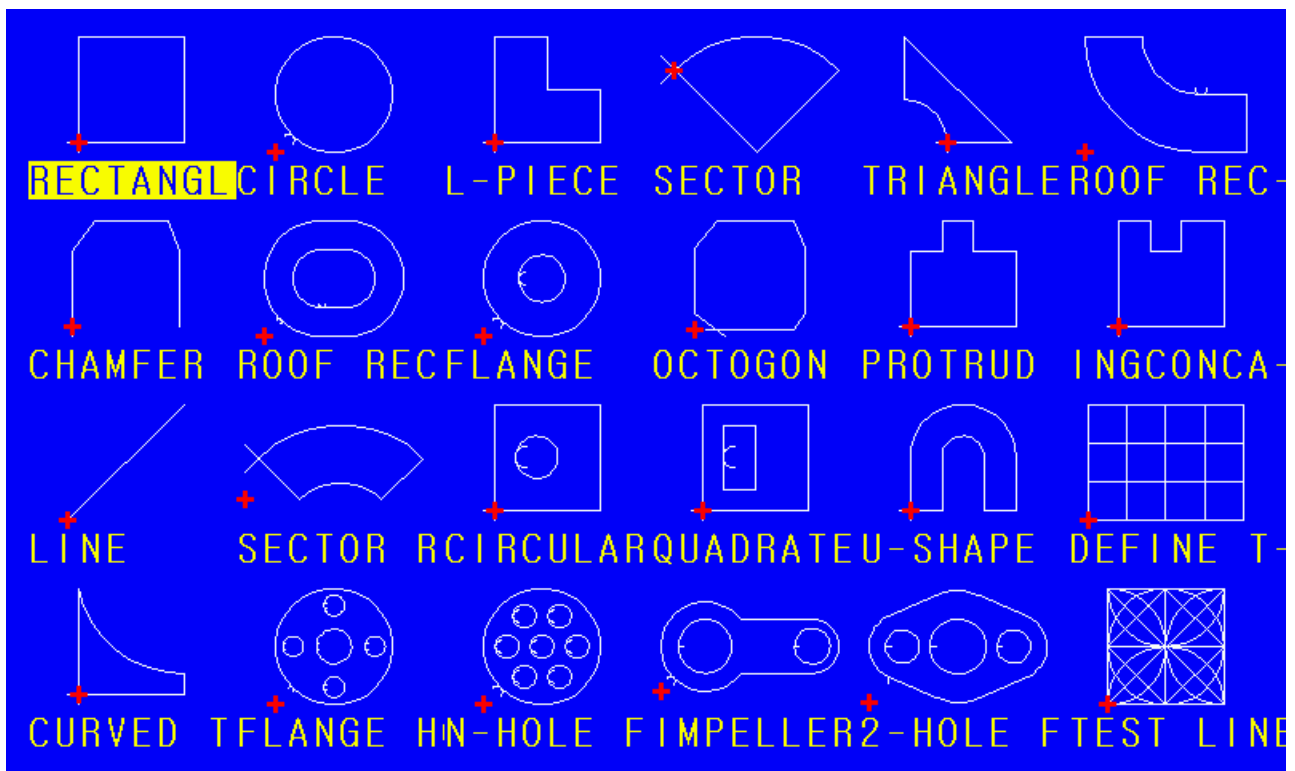


Fig. 8.1 LIBMINIT Feature(Library of Patterns)

Service Support Spirit

8.2. Selection of Pattern Elements

So far, this system provides 24 pattern elements, which is expandable at any time as per client requirement. It is allowed to select the interested pattern by pressing [↑], [↓], [←] and/or [→] key to move the cursor to right position and press [ENTER] to acknowledge the selection.

Note: The 17th pattern element is customized element.

8.3. Arrangement and Layout of Pattern Element

After pattern element is selected as per the above-mentioned procedure, the system will prompt at the upper right corner to input parameters for the pattern, as show in Fig. 8.2

【F1】 **iPART Workpiece**: To machine into workpiece that is solid internally.

【F2】 **ePART Hole**: To machine into hole that is solid externally.

【F3】 **ROT Rotate**: The system will prompt to input rotation angle and press [ENTER] or [F6] to acknowledge. The pattern rotated such an angle will appear. Counterclockwise direction is taken as positive angle.

【F4】 **NEST** : The system will prompt user to provide:

ROWS----Row number of machined parts array

COLUMNS ----Column number of machined parts array

H-BETWEE ----Distance between rows.

W-BETWEE----Horizontal distance between machined parts

OFFSET----The deviation of raw misplacement, as shown in Fig. 8.3.

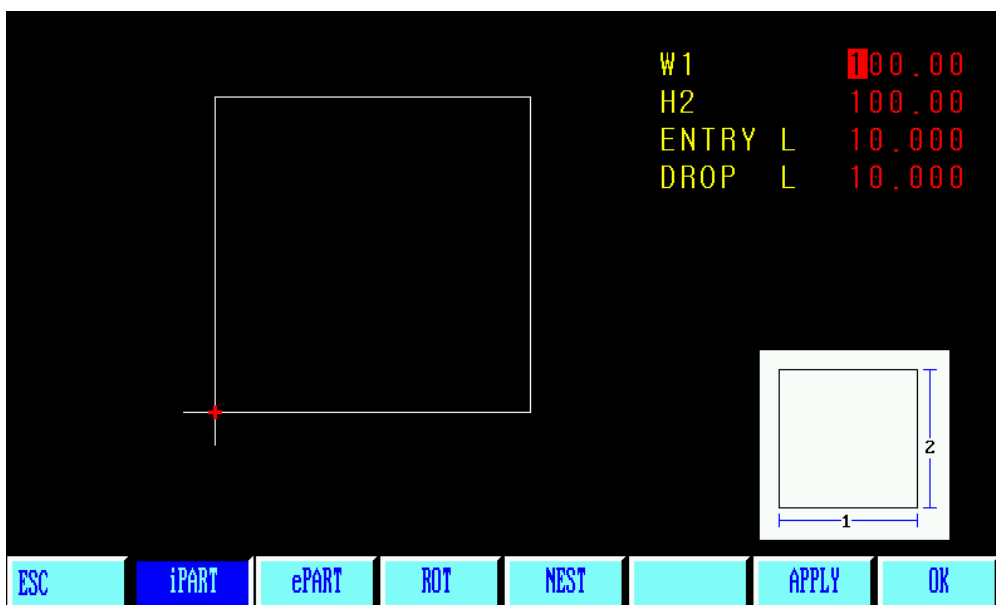


Fig. 8.2 Pattern setting menu

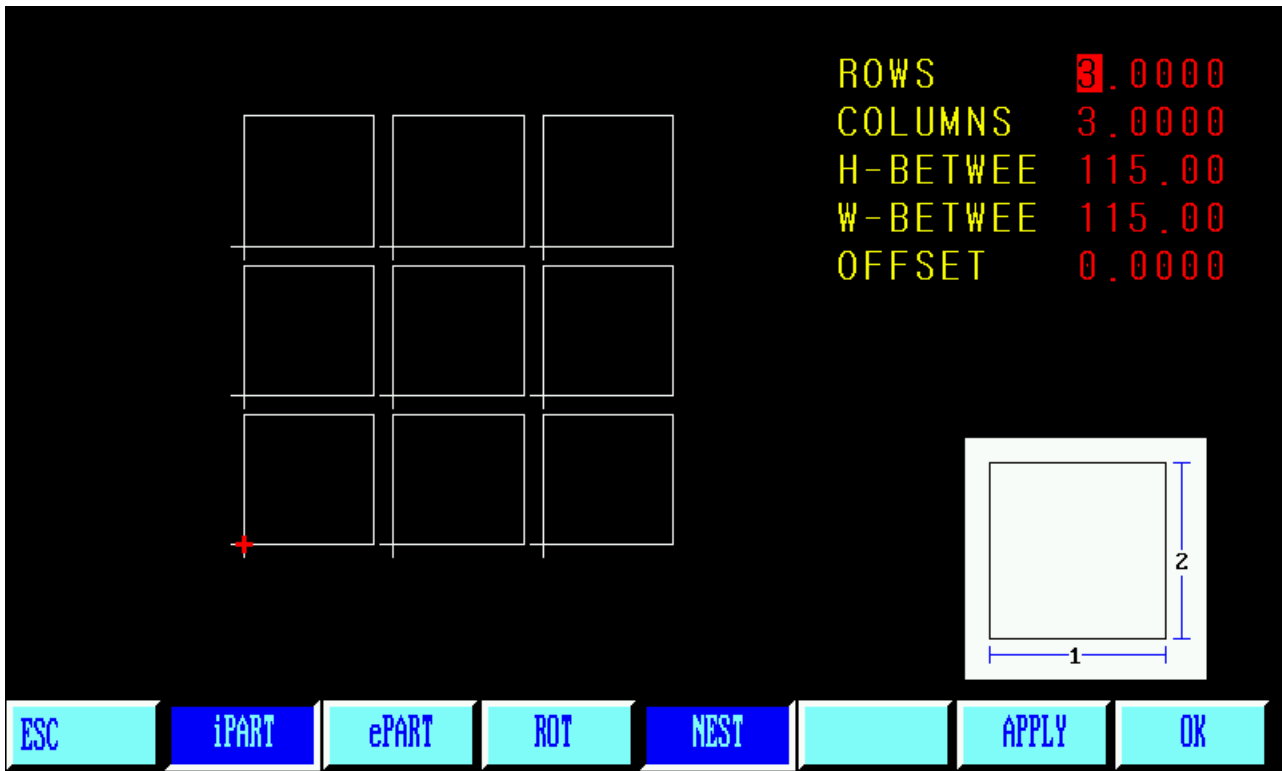


Fig. 8.3 Schematic diagram of part layout

【F6】 **APPLY**: After parameters are set, Press this button to generate a new graph, in this interface;

【F7】 **OK**: After parameters are set, press this key to generate machining program.

9. Diagnose

In the system main menu, press [F5] to enter DIAGNIZE main window, as shown in the figure below:

```

OUT:
  M10 M12 M14 M16 M38 M20 M22 M24
  0    0    0    0    0    0    0    0

INPUT:
  LWP LWN SX0 SY0 DLZ STO PAS LDW
  1    1    1    1    1    1    1    1
  SXP SXN SYP SYN PRE SPP SPN CHE
  1    1    1    1    1    0    0    0
    
```

26-06-10 17:29:16

In Flame Operation

```

OUT:
  M38 M12 M14 M16 M10 M20 M22 M24
  0    0    0    0    0    0    0    0

INPUT:
  LWP LWN SX0 SY0 DLZ STO PAS LDW
  1    1    1    1    1    1    1    1
  SXP SXN SYP SYN PRE SPP SPN CHE
  1    1    1    1    1    0    0    0
    
```

26-06-10 17:29:45

In Plasma Operation

Service Support Spirit

9.1. Check Input/output Ports

The system diagnosis will show the hardware resources available. In DIAGNOSE window, it is allowed to check the state of input/output port.

9.2. Output Check

Move cursor to any position of the 8 optical isolation output ports and change the level state of output with "0" and "1". 1 stands for enable while 0 stands for disable. Refer to "Definitions of Input/output Port" for the definitions of output ports.

9.3. Input Check

Let the system to show the state of active optical isolation input ports. 0 stands for low potential (grounding) while 1 stands for high potential (24V or overhead). See "Definitions of Input/output Ports" for the symbol definitions of input ports.

Input diagnosis:

LWP---External limit (normally closed switch), choosing bump shot detection, the foot a gun testing point.

LWN---External negative limit (normally closed switch).

SX0 / DUP---Has two functions, reset direction is valid is X0 mechanical original, it is an external manual, cutting up switch (no lock, normally closed connection).

SY0 / DDN---Has two functions, reset direction is valid is Y0 mechanical origin, it is an external manual, cutting down the switch, no lock, normally closed connection).

DLZ---Plasma arc mouth (arc starting signal success) measurement, low effective.

STO---External abrupt stop switch (normally closed connection). **PAS** - external pause switch (normally closed connection).

LDW---Before the initial positioning of plasma perforation

SXP SXNSYP SYN PRE---Is an external wireless remote control coding input.

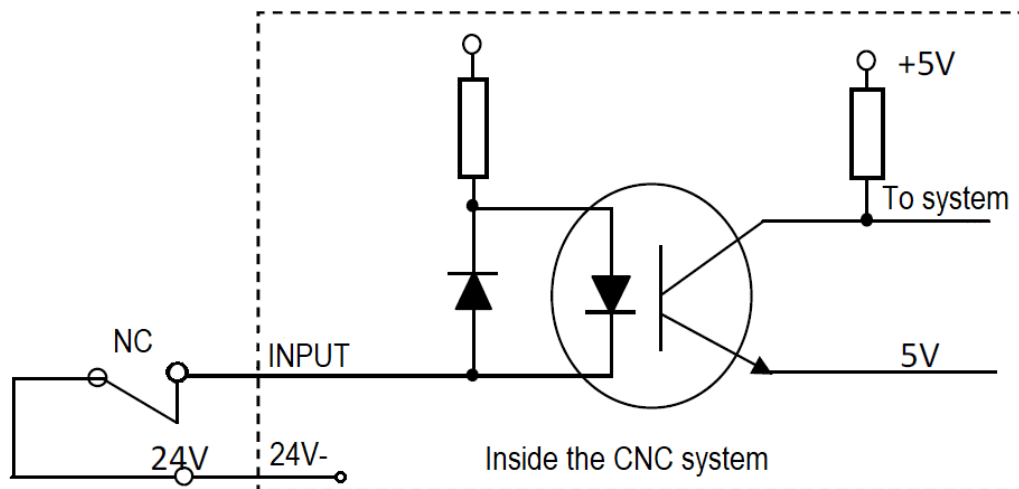
SPP SPN CHE---Useless interface.

10. System input/output interface connected

The system interface: including input/output (DB25 core), the motor (15 core), a wireless remote control (15 core), meet the company production of block.

10.1. The system input principle

Generally limit/start/stop using mechanical switch, to prevent interference to enter, usually use the normally closed contact of a mechanical switch, according to the chart mode connection.



Note:

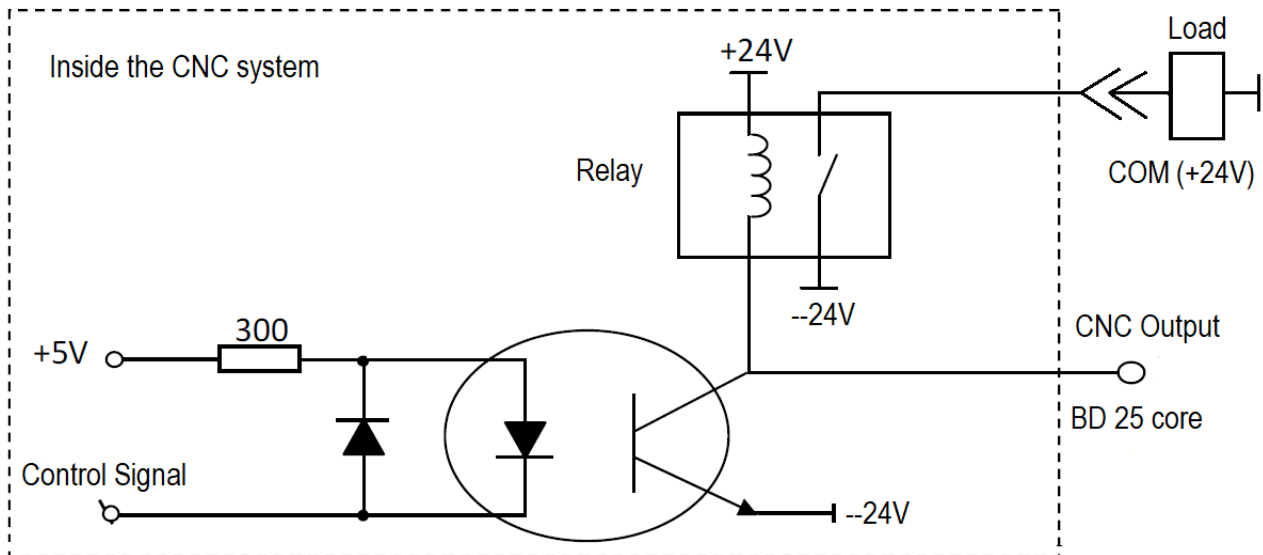
- ◆ System requires urgent stop, pause, the logic is consistent, namely by normally open or connected normally closed points (common). System boot automatically detects the external trigger a state, as a control.
- ◆ System boot automatically detects the external trigger a state, as a control.
- ◆ So, if you do not answer the external start switch, the corresponding bit should be received 24 V (similar to answer normally closed contact) or what all don't pick up (similar to connect normally open contacts).

Service Support Spirit

10.2. The principle of the system output:

Control signal = 0 switch/relay connected (24 V loop formation, low effective signal)

Control signal = 1 switch/relay disconnect (24 V did not form a loop, cancel)



10.3. Input/output connect definition

Detailed explanation:

Signal Definition	BD 25core	Detailed explanation	
>W+	1	Input	X/Y + limit, two axes is limit concatenated, high effective. If you don't use, please send short signal received 24 V, if do not use the limit, it is a bump shot detection (normally closed).
W-<	14		X/Y - limit, limit concatenated two axes, high effective. If you don't use, please send short signal received 24 V.
SX0/DUP	2		SX0 mechanical origin, NPN proximity switch, normally is open. Reset the direction is invalid when the mouth for external cutting up key, high effective, if you don't use, please send short signal received 24 V.

Service Support Spirit

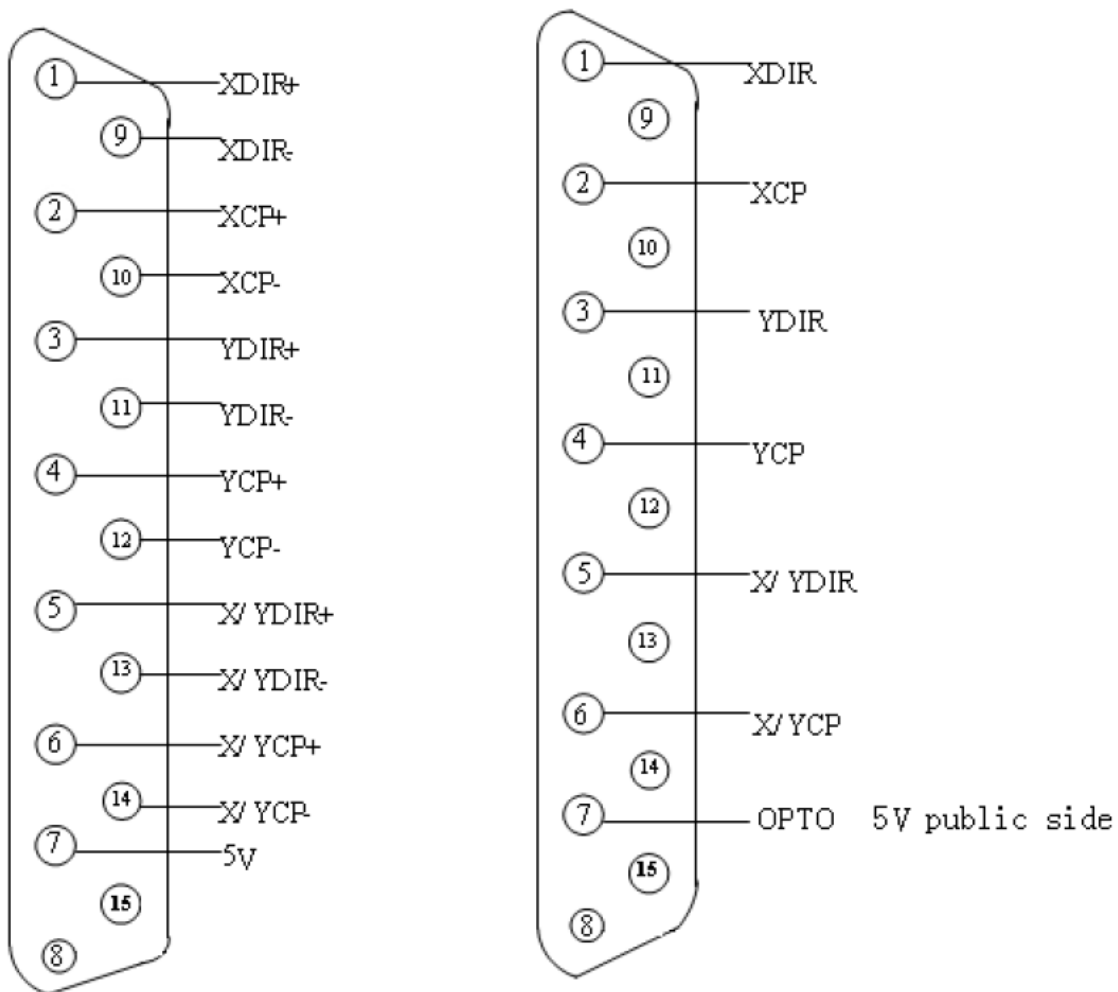
SY0/DDN	15		SY0 mechanical origin, NPN proximity switch, Normally is open. Reset direction is invalid when the mouth as external cutting down key, high effective, If you don't use, please send short signal received 24 V.
DLZ	3	Input	Detect arc voltage, low effective and high arc pressure when not connected to.
STO	16		External scram button and high effective, If you don't use, please send short signal received 24 V.
PAU	4		External pause button, high effective, If you don't use, please send short signal received 24 V.
XXW	17		A plasma cutting torch orientation, the lower limit.
	5		Raised corner signal (higher automatic/manual switch) relay normally open contact.
	18	Output	Output signal relay ignition normally open contact.
	6		COM (OH1 / OH2 OH3 / OH4 relay) to isolate the output public side of adjustable.
	19		Block cutting torch (OH4) normally open reduction is set.
	7		Block cutting torch (OH3) I normally open tuning.
	20		Block cutting torch (OH4) normally closed reduction is set.
	8		Block cutting torch (OH3) rose normally closed.
M10/M11 or M38/M39	21		Flame: M38 higher control M38, M39 close Plasma: M10 standby M10, M11 close.
M20/M21	9		M20 (open), M21 (closed), fire mode, used in the ignition switch. Plasma model, when the ignition delay time of 0, is always open (switch) as a function of flame/plasma.
M22/M23	22		Spare.
M24/M25	10		Spare.

Service Support Spirit

	23	Output	Output Corner (OH1) normally closed.
	11		Output Cutting oxygen (OH2) normally closed.
24V	24		Output +24V/1A.
24V	12	Output	Output +24V/1A power supply.
24V-	25		Output 24V power supply ground.
24V-	13		Output 24V power supply ground.

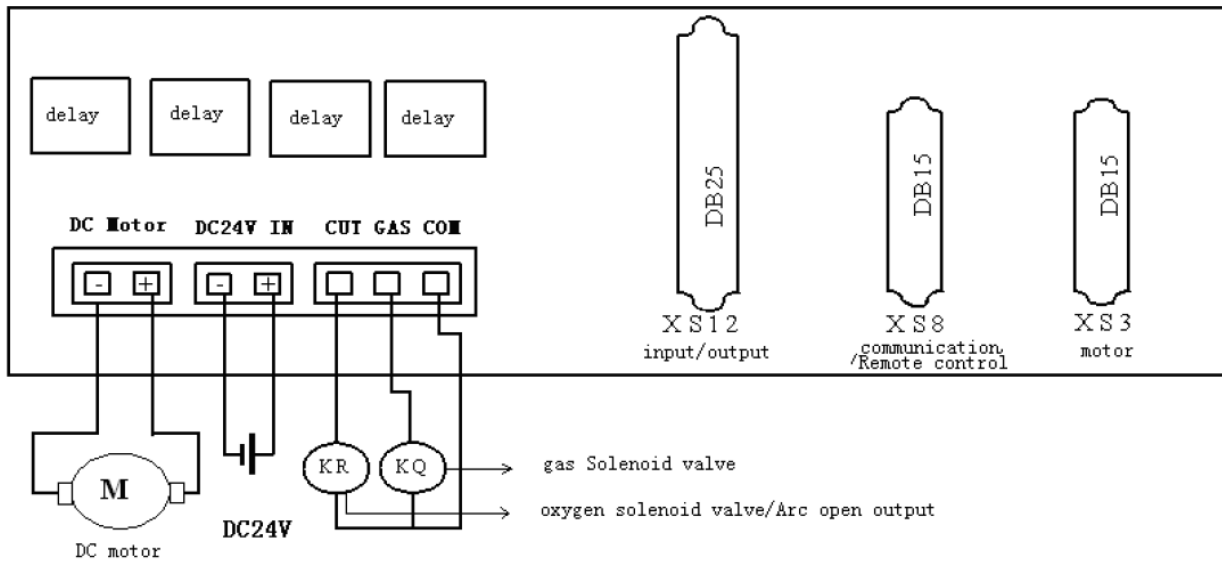
10.4. 15 core motor port definitions

1) Differential signal (2) a total of anode connection



Service Support Spirit

10.5. When using flame cutting typical wiring (DB15)



Appendix I: Instructions for Software Upgrading Operation

Functions:

The system software is allowed to upgrade through an USB disk.

The specific operation procedure is as follows:

1.File upgrading

Copy the upgrading file titled STARTCNC.EXE to an USB disk.

2.Operation procedure

Push and hold the button between  the red button 0 and the USB port.

Power the system on and release the button when upgrading window appears on the display screen. Plug the USB disk into the port and press Key F1 under the display screen(the key corresponds to "UPGRADE" in the menu displayed). The system will automatically start upgrading operation.

3.Upgrading end

If program upgrading is successful, the system will prompt "Upgraded Successfully" and sound one beep. If program upgrading fails, the system will prompt "Upgrade Failure" and sound continuous beeps. ◦

4.New program startup

Power the system off and unplug the USB disk. Reboot the system to start up the upgraded program. The version number of the upgraded program will appear on the screen after booted up.

Notes:

Please make the following check if upgrading operation fails.

- 1) The USB disk must be **FAT or FAT32 format**. FAT format is preferable.
- 2) The upgrading file must be titled as **STARTCNC.EXE**.
- 3) If no prompt appears in the upgrading process and the display returns to the main upgrading window or upgrading operation fails due to any other interruption, verify first the name of upgrading file and the format of USB disk. If these two factors are assured, try to upgrade the program again following the specific operation procedure.

Appendix II: Troubleshooting

Fault Symptoms	Fault Check	Fault Analysis	Recommended Solutions
Suddenly breakdown in normal operation	Always stop at the same position in every operation cycle	a. Machining program at fault b. CNC software at fault	Dispatch the software back to the manufacturer for professional troubleshooting.
	The system still runs but the cutting torch stops abnormally along all axes or one axis.	a The travel motor is locked. b The travel motor fails.	Check if the machine is frozen and level enough.
	The system still runs but the cutting torch does not move along one axis any more and gives no response to any keystroke, especially to the pause key.	The machine operation is interrupted.	a. Remove the interference and make sure the grounding wire of plasma power supply is connected to the machine. b. Check if the motor(previous model) has isolation board or not. c. If it is flame mode, check if the solenoid valve for cutting oxygen is given arc extinction.
Motor fails to run	Check the system wiring.	Loose connection or disconnection	Secure the connection.
	Check the driver.	Check for any alarm message or if the setting is correct or not.	Change the setting of driver.
	Check system setting.	Check if the start speed is zero or too high, the speed regulation time is too short and the electronic gear or pulse equivalency is appropriate to be zero.	Suggest to record the original parameters before making any modification.

Service Support Spirit

	Check for axle problem and if the same model of motor is available.	Replace parts one by one to isolate the real cause to the problem.	Replace the broken part.
Slow system response	Check the power supply.	Check if the output voltage of power supply is within the allowable range.	Adjust the output voltage or replace the power supply for switch.
	Check the system parameters for any error.	Suggest to record the original parameters before making any modification.	
Travel limit alarm at power-on	Check if any external limit switch is connected.	Check if limit switches are functional and if loose connection or disconnection exists.	Secure the connection.
	If no external limit switch is connected, check the system if the current coordinate values are too high and out of the travel limit set by software. Check if the travel limit set by software is too small.	Modify it to normal range.	
	In DIAGNOSE window, trigger the external limit switch.	Check for any change with the corresponding input port.	
No response to keystroke or wrong key assignments	Check the cable for keyboard.	Check if the keyboard is loosely connected or disconnected.	Secure the connection.
The system is locked up in manual/automatic mode.	Incorrect parameter setting.	Record the electronic gear in system parameters and then press G, G and 3 in the system main window successively. Save the parameters in parameter setting mode. Restart the system and fill the recorded value of electronic gear into the corresponding position. Save the parameter and test the system performance.	

Service Support Spirit

No input/output	Check the cables for input and output.	Check if they are well connected.	Secure the connection.
	Check if 24V power supply is normal or not.	Check the voltage of 25-pin port between Pin 24 and Pin 25.	Check the switching power supply.
	Check the cables inside the system for any loose connection.		Secure the connection.
Blank screen	Adjust "+" key and "-" key at the upper right corner of keyboard for proper contrast and check for any screen display.	If yes, adjust the contrast to normal display. If not, check the cables for LCD for any loose connection.	Make readjustment. Secure the connection.
	USB disk is not compatible with the system.		Try the USB disk of other model or brand.
USB transmission failure	Incorrect format of USB disk		Reformat the USB disk to FAT format.
	Check the lamp indicator of USB port for any light.	Check the cables inside the system for any loose connection; check the USB port for any foreign material and all the pins are in normal condition.	Secure the connection and remove foreign material from the port.
	The system returns to the main window after machining program is started.	Replace the machining program and test running the system; initialize the system if any parametric problem.	If the problem is gone, it means that machining program has bug. Check the program.
The system can only machine	Initialize the system if any parametric problem		Initialize the system parameters.

Service Support Spirit

straight line and fail to machine a circle.	Replace the machining program and test run again.	If no more problem, check that machining program.	Check the program or return it to the manufacturer for diagnosis.
Screen blinks when a solenoid valve is opened or plasma arc ignition is going on.	<p>a. The power supply unit for plasma, CNC system and the enclosure of machine must share a ground wire that is 2mm thick at least in diameter and well connected to the ground.</p> <p>b. Shielded cables must be used for the system input/output and the exposed part at the cable end must not be longer than 30mm.</p> <p>c. The unused input end must be connected to 24V ground wire.</p> <p>d. A 0.01uF capacitor should be used between the 24V ground wire and enclosure(shielded cable).</p> <p>e. Use an isolation transformer(380V-220V) and as interference suppressor if it is possible.</p> <p>f. Separate the operational power supply for high-power welding machine/cutting machine from that for CNC system.</p>		
Failure to save the program or clobber	Failure to save the program.	After replacing batteries, press G, G and 3 in the system main window successively to format the program area.	
No system output	Check if 24V power supply is normal or not.	Measure the voltage between Pin 24 and Pin 25.	Check and replace the switching power supply unit.
	Check if the four relays on the rear panel are reliably connected, if the relay operation is normal and if the contact is functional.		Replug or replace the relays.