HUST Mill CNC Controller

Manual

Model: HUST CNC H4CL-M

Version: Sep 2006

1 Main Features of HUST Mill CNC Controller

- ☐ Controlled Axis: X, Y, Z and Spindle Encoder Feedback
- ☐ Or control: X, Y, Z, A and no Spindle Encoder Feedback
- ☐ Program Designed by CAD/CAM on PC. Program input and DNC on-line execution from PC through RS232C interface.
- ☐ Memory Capacity for CNC main board - 512k.
- ☐ USB can be used to increase memory capacity.
- ☐ The LCD display is a 800x600, 16 color screen.
- ☐ Battery Backup for CNC program storage in case of power-off.
- ☐ Backlash error compensation for worn lead screw.
- ☐ Provide 40 sets of tool length offsets.
- ☐ Self-designed MACRO Program.
- ☐ Single block and continuous commands.
- ☐ Optional Skip functions.
- ☐ Optional Stop and Feed hold functions.
- ☐ Simultaneous use of absolute and incremental programmable coordinates.
- ☐ Self-diagnostic and error signaling function.
- ☐ Direct use of “ R”, “ I” and “ J” incremental values for radius in circular cutting.
- ☐ MPG hand-wheel test and collision free function for cutting products at the speed controller by MPG.
- ☐ Equipped with 24 standard programmable inputs and 16 outputs.

This operator’s manual includes basic operation, program editing, G/M code, parameter settings, connections and maintenance (plus warning descriptions) with examples and explanations for each command instruction.

If there are any problems with application, please fill out a problem sheet indicating the nature of the problem. Send it by either fax or mail. We will respond to you as soon as possible.

2 Operation

2.1 Basic Operation

Screen Description

*** Startup Screen**

After powering the controller, the following startup screen displays:

*** Standby Screen**

After 3 seconds, the standby screen displays (this screen will also display when the Reset key is pressed and no mode is selected), as the following figure shows:

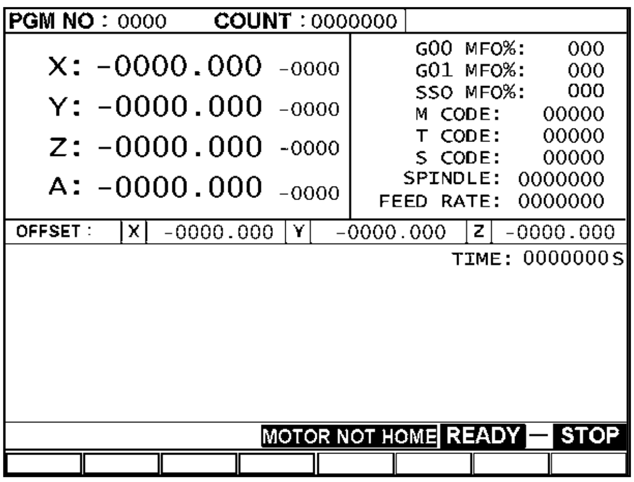


Fig 2-2

* MPG – TEST (MPG hand-wheel test)

Turn the Mode to the “MPG – TEST” to enter this mode.

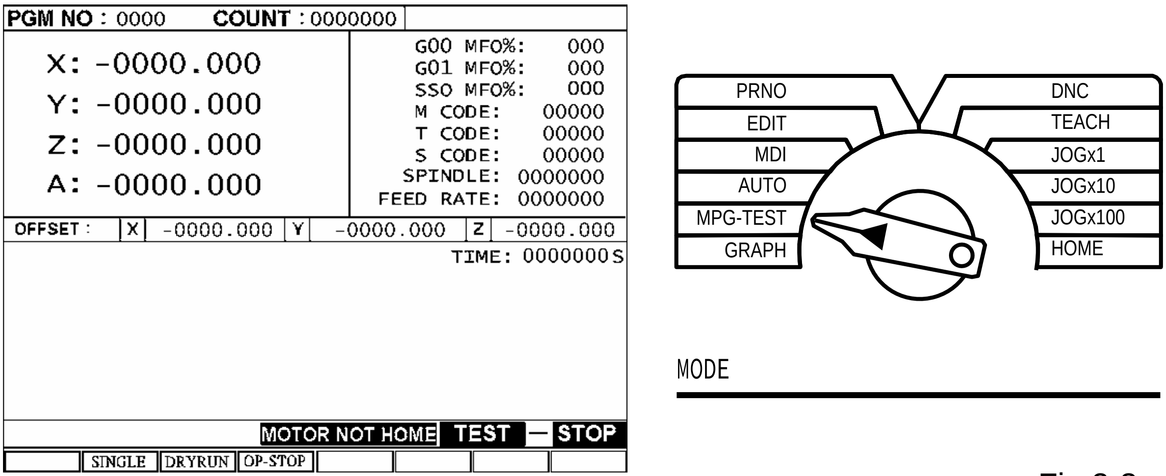


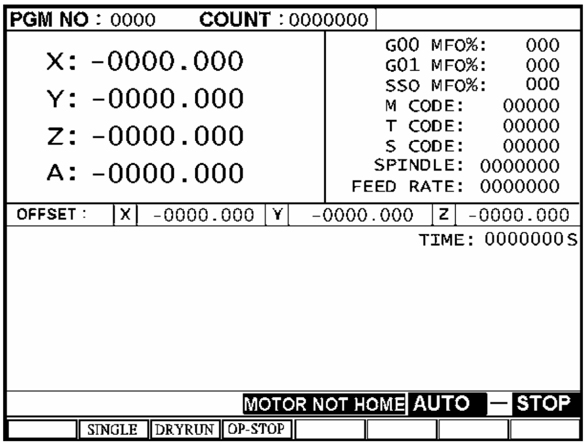
Fig 2-3

When the start key is pressed in the “MPG – TEST” mode, no axis will move before the hand-wheel is rotated. The axes will stop moving when the hand wheel stops rotating. This function is very useful for checking the changes of each block and ensuring the program correctness at the initial stage of program development.

Switching between the “MPG – TEST” mode and “Auto” mode is possible when the program is running. When a program section failure is suspected, the mode can be switched to MPG – TEST to check the changes in the program. It then switches back to Auto mode when the problem is removed.

* Auto Mode Screen

Press the “Auto/ MDI” key or turn the mode to Auto to enter Auto mode. The following screen displays:



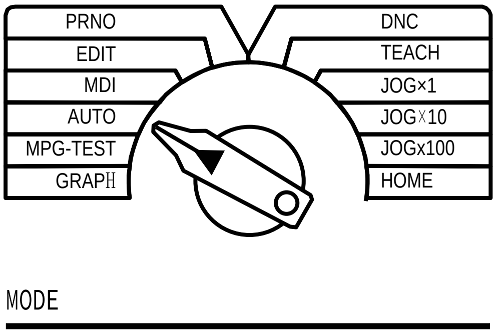


Fig 2-4

Press the PAGE “” key and the following screen displays:

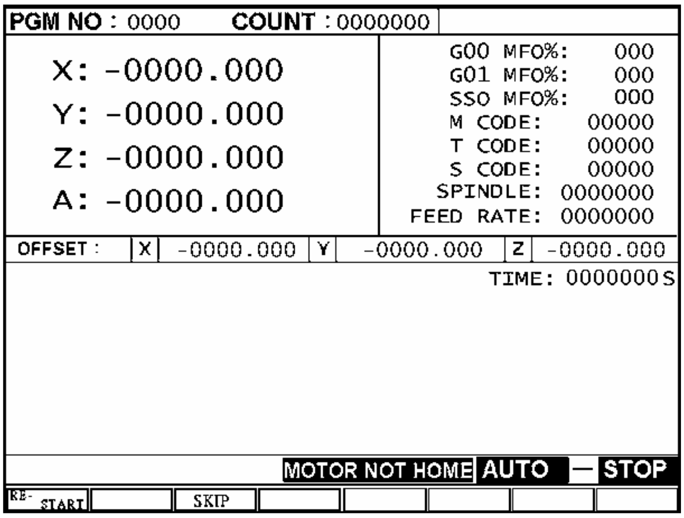


Fig 2 – 4 - 1

Functions of the “soft keys” in Auto mode:

1. **SINGLE:** This function can be selected at any time no matter whether the program is running or stops.
   Whenever the "Start" key is pressed with this function selected, only the next command line will be executed instead of the entire program.

2. **Dry Run:** This function can be selected or canceled only when the program stops running.
   When this function is selected, all specified feed-rates (F value) in the

program become invalid and the program runs at the highest speed set in MCM #148~151.

3. **OP_Stop:** This function can be selected at any time no matter whether the program is running or stops.
   When this function is selected, the M01 command in the program is used as stop command. It doesn't function if the Optional Stop is not selected.

4. **OP_Skip:** This function can be selected at any time whether the program is running or not.
   When this function is selected, the /1 command in the program will be skipped (not executed). This command line will be executed if the Optional Skip is not selected.

5. **Re_start:** This function needs to be selected before the program runs.
   When the "Re_start" function is selected, program operation proceeds with the execution from the position interrupted.

COUNT:

The 『COUNT』 will be increment by 1 when the program runs to M02, M03, or M99. Press the "0" key twice to reset.

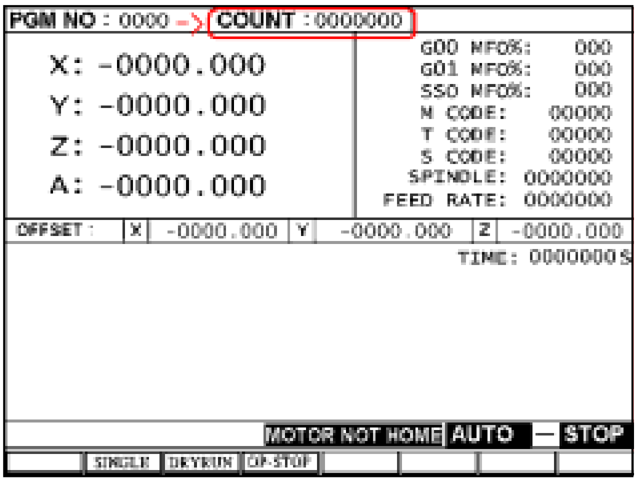


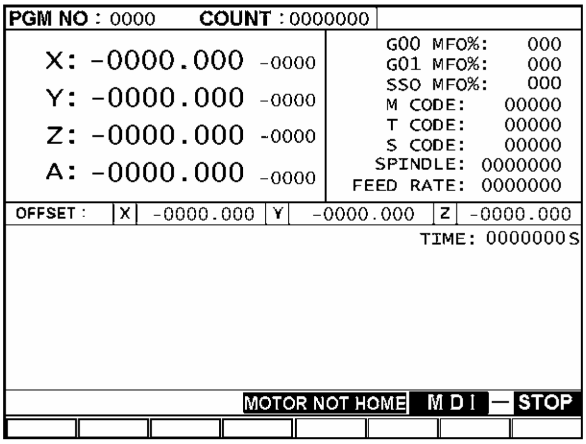
Fig 2-5

RUN Time:

The current RUN time (sec.) is displayed. Resetting is performed automatically when the controller is restarted after interruption of ending of the program.

* MDI Mode Screen

Press the “Auto/ MDI” key twice or turn the mode to MDI,to enter MDI mode. The following screen displays:



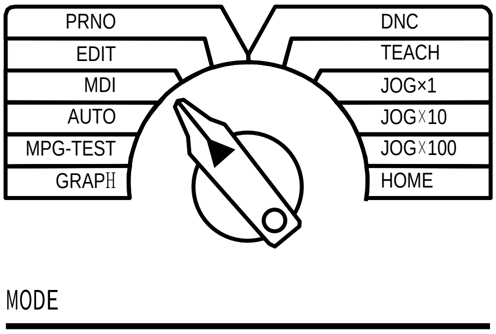
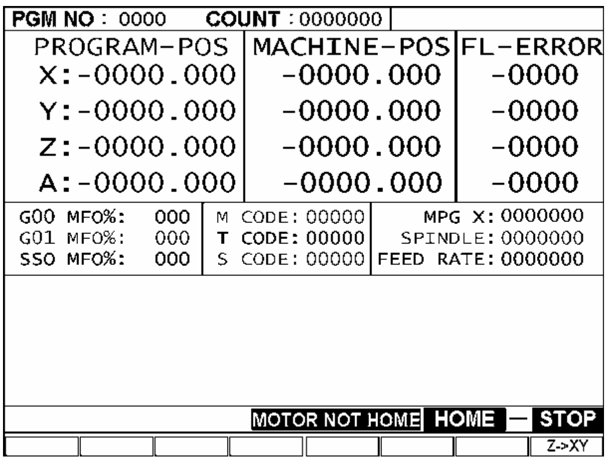


Fig 2-6

A single command line is executed during MDI mode.
You can enter the MDI mode when the program is running to check the machine coordinates and following error.

* HOME Mode Screen

Press the “JOG / Home” key twice or turn the mode to Home, to enter the HOME mode. The following screen displays:



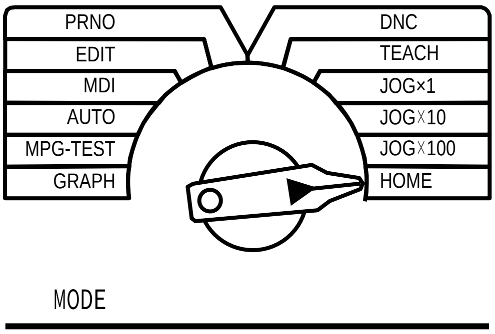


Fig 2-7

Methods for returning to the HOME:

Select an axis for returning to the HOME with the **axis** and press the “Start” key to perform the action of returning to the HOME

When Z-axis needs to return to the HOME before X and Y axes, press the “Z->XY” key at the bottom of the screen for **1 second** to execute the homing action of Z-axis. X and Y-axes return to the origin simultaneously after the homing action of Z-axis is completed.

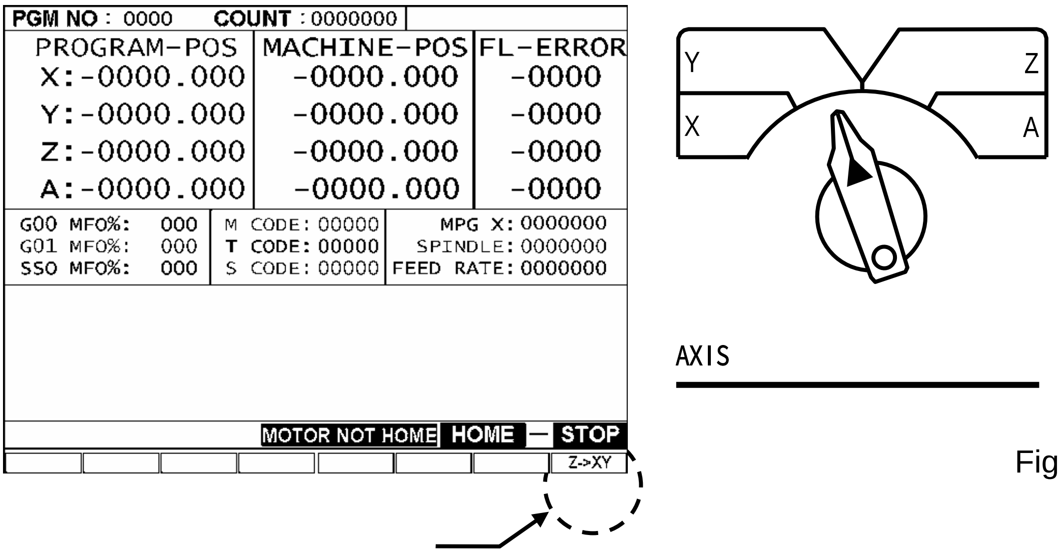


Fig 2-8

* Jog Mode Screen

Press the “JOG / Home” key once or turn the mode to JOG to enter Jog mode. The following screen displays:

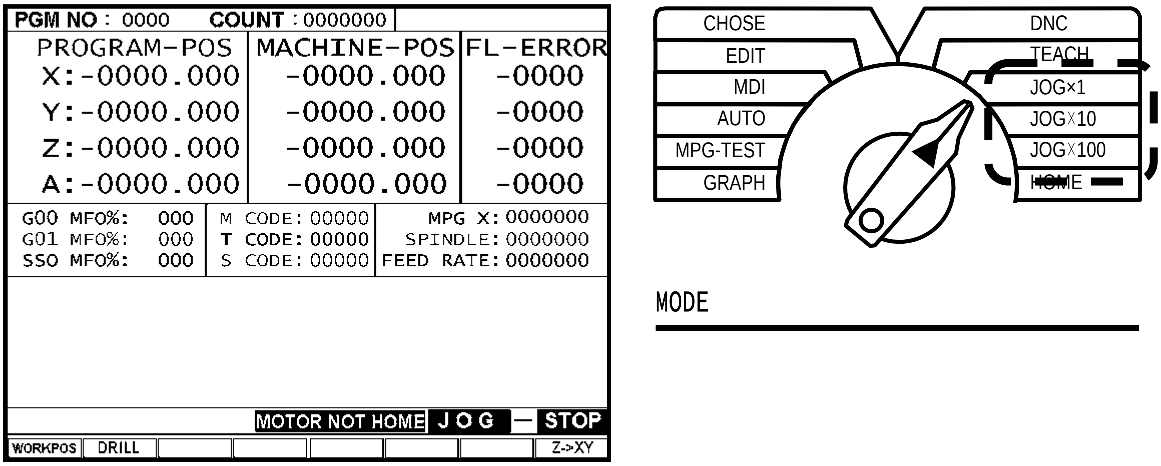


Fig 2-9

The jog mode provides the following functions:

1. Axial Positioning:

◆ **Continuous Movement:** Select an axis with the knob and press the JOG + or JOG- on the auxiliary panel.

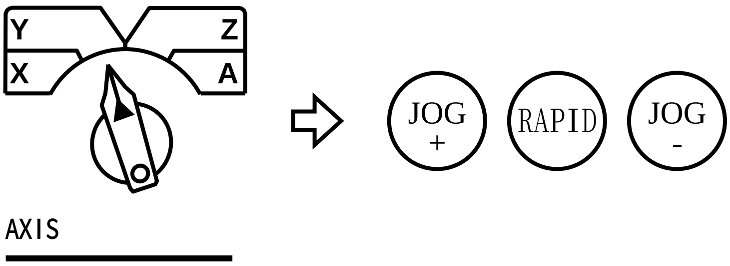


Fig 2-10

◆ **Hand-wheel:** Select an axis with the knob and turn the hand wheel (when an external hand-wheel function is selected, it will be controlled externally).

2. Manual Switch (Soft_Key):

   a. Spindle: CW, CCW, stop.
   b. Coolant: Press the key to turn on and press it again to turn off.
      When the LED indicator lights up, it indicates that an action is running.

c. Lubricant: Press the key to turn on and press it again to turn off. The LED illuminates to indicate that an action is running.

3. Press the Origin Setting and the following screen displays:

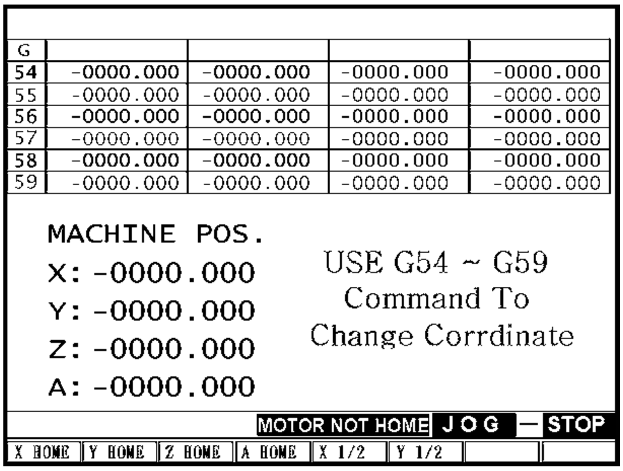


Fig 2-11

(1).Standard work origin setting:

a. Have the tool touch the end face of the work-piece manually or using the hand-wheel.
b. Press the soft key X Origin, Y Origin, and Z Origin for three seconds.
c. Write the machine coordinates of this point in work coordinates.

(2).Work-piece center point setting:

a. Have the tool touch the End Face A of the work-piece manually or using the hand-wheel as described in (1). Press the soft key X Origin, Y Origin, and Z Origin for **3 seconds** and write the machine coordinates of this point in work coordinates.

b. Then have the tool touch the End Face B of the work-piece manually or using the hand-wheel.

c. Press the soft key X 1/2 and Y 1/2. The controller sums up the machine coordinates of the end faces A and B and divide the total value by 2. Apply the quotient to the work coordinates and this position is the center point of the work-piece in the work coordinate system.

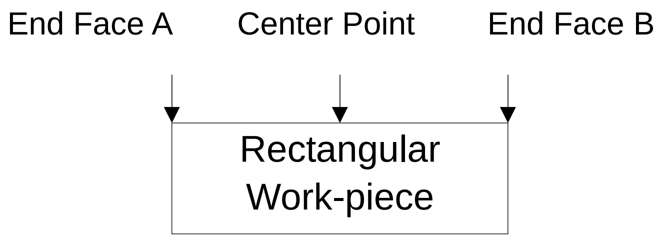


Fig 2 -11 - 1

4. Press the Manual Drilling and the following screen displays:

*** Simple manual drilling**

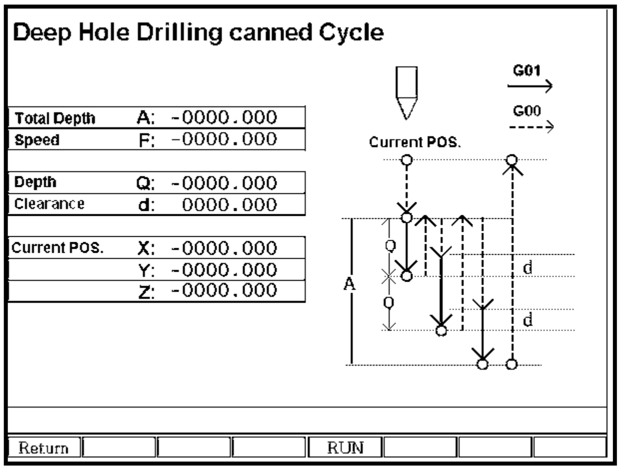


Fig 2-12

(1).Complete the fields and press the Execute key to start the processing procedure according to the value entered by the user.

Drilling Depth: The downward drilling depth along the current Z-axis.

Drilling Speed: The speed of drilling a hole (G01).

Each Feed Depth: The depth of each drilling at which the Z-axis returns to the start point.

Reserved Distance: Quickly feeding to the last drilling depth – coordinates of the reserved distance.

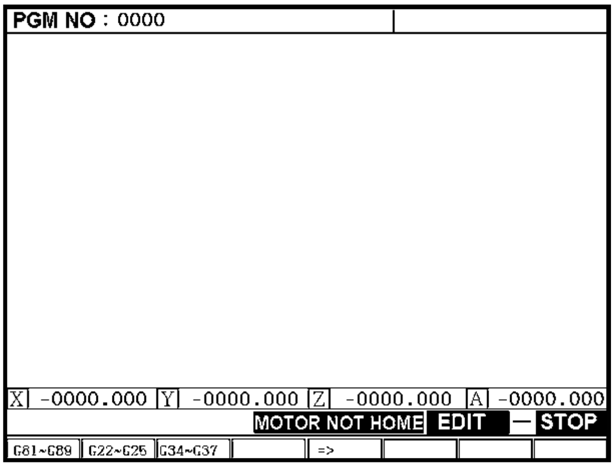
Current Coordinates: Display the current X, Y, and Z coordinates. (The user may complete this field, if required.)

<u>Press and hold the “Execute” key (not the Start key) at the bottom of the screen to execute the drilling action. (Start the spindle manually before drilling).</u>

(2).Press the soft key "Return" to return to the manual jog screen.

*** Edit Mode Screen**

Press the “Edit/PRNO” key once or turn the function mode to **<u>“Edit”</u>** to enter the Edit mode. The following screen displays:



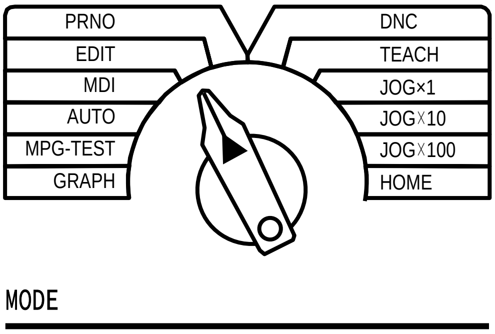
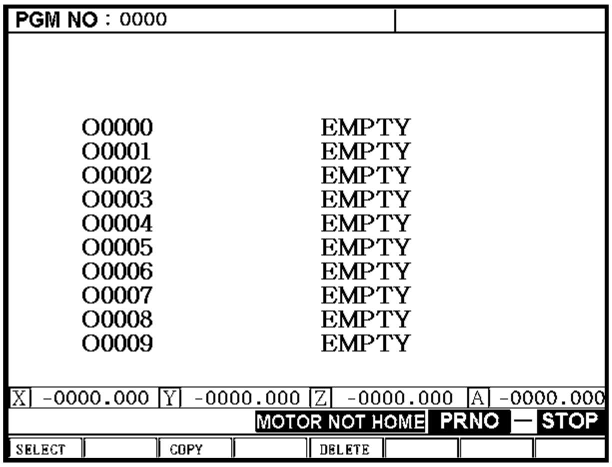


Fig 2-13

The program can be edited in this mode. Refer to Section 2.3.2 ~ 2.3.7 for more information about program editing.

* Program Selector Screen

Press the “Edit/PRNO” key twice or turn the function mode to **<u>“PRNO”</u>** to enter the PRNO mode. The following screen displays:



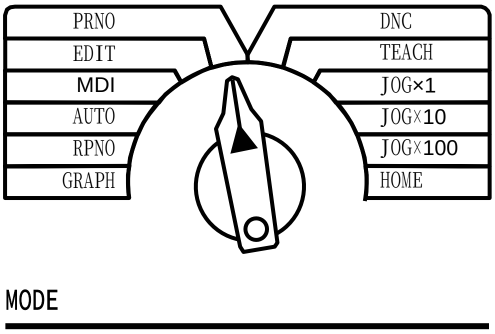


Fig 2-14

Program selection methods:

1 Select a program:

(1). Use the “Cursor ” key or “Page " key to move the arrow to the desired program number.

(2). Press the “Enter” or “Select” key.

2 Program comments:

(1). Use the “Cursor ” key or “Page " key to move the arrow to the program number for which program comments are entered.

(2). Enter letters or numbers.

(3). Press the "Enter" key.

3 Delete a program:

(1). Use the “Cursor ” key or “Page " key to move the arrow to the program number to be deleted.

(2). When you press the “Delete” key, a dialogue pops up to request your confirmation.

(3). Press the “Y” key to conform and delete the program.

4 Copy a program:

(1). Press the “Copy” key and the following screen displays:

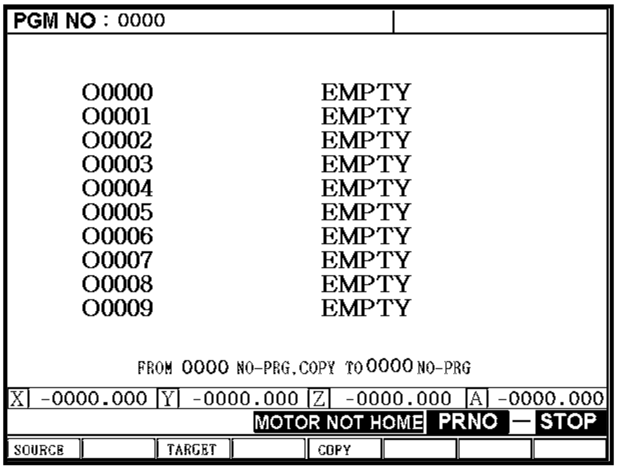


Fig 2-15

(2). Use the “Cursor” key or “Page" key to move the arrow to the program number of the **Source program**.

(3). Press the “Source” key

(4). Use the “Cursor” key or “Page" key to move the arrow to the program number of the **Target program**.

(5). Press the “Target” key.

(6). When program numbers of the source and target files are confirmed, press the “Copy” key to execute the copy action.

* I/O Mode Screen

Press the “I/O/Parameter” once to enter the I/O mode. The following screen displays:

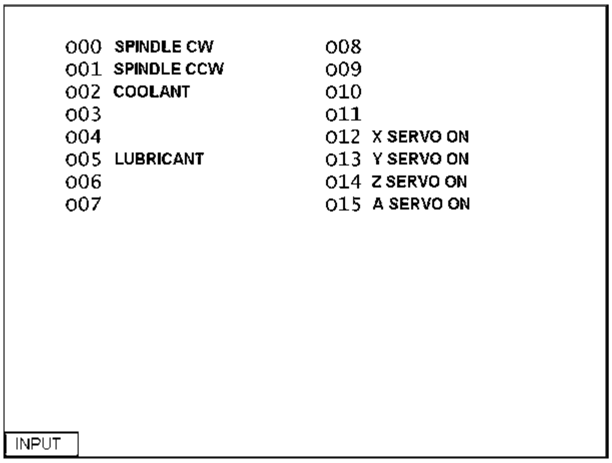


Fig 2-16

000-023 is the INPUT status. (Highlight shows input)

000-015 is the OUTPUT status. (Highlight shows output)

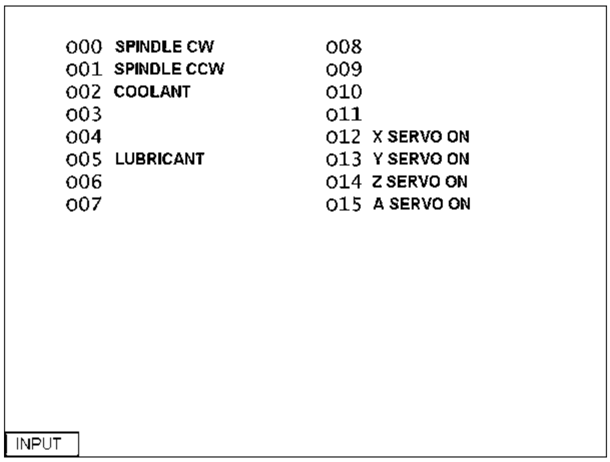
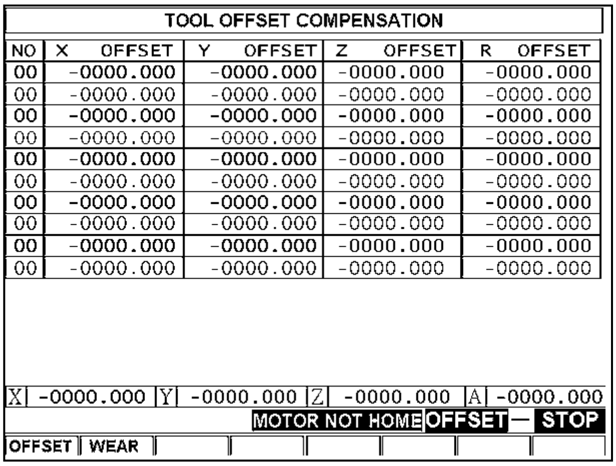


Fig 2-17

* Tool Compensation Screen

Click the “T.Radius/T.Offset” once to enter the tool length compensation mode. The following screen appears: ※ (Refer to Section 3.15.6 for more information)



TOOL OFFSET COMPENSATION

NO	X OFFSET	Y OFFSET	Z OFFSET	R OFFSET
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000

X -0000.000 Y -0000.000 Z -0000.000 A -0000.000

MOTOR NOT HOME OFFSET — STOP

OFFSET WEAR

Fig 2-18

Switching between the screens is possible using the soft key in this mode.

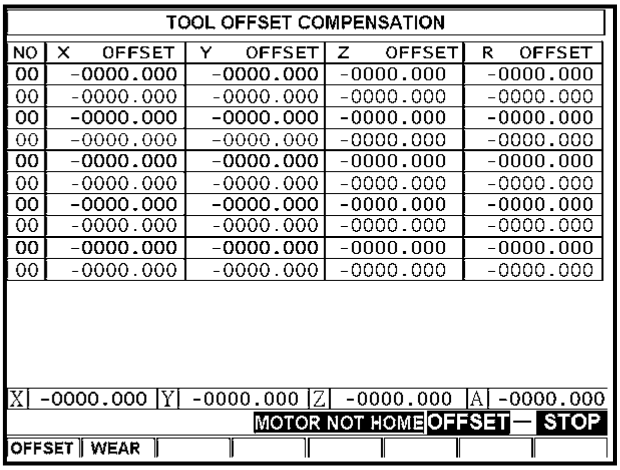
<u>Tool-wear compensation screen and tool length compensation screen</u>

1. Follow the steps below to configure the parameters for tool length compensation:

   a. Use the “Cursor” key to move the cursor to the parameter to be changed.

   b. Enter the desired wear value.

   c. Press the “Enter” key.



TOOL OFFSET COMPENSATION

NO	X OFFSET	Y OFFSET	Z OFFSET	R OFFSET
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000

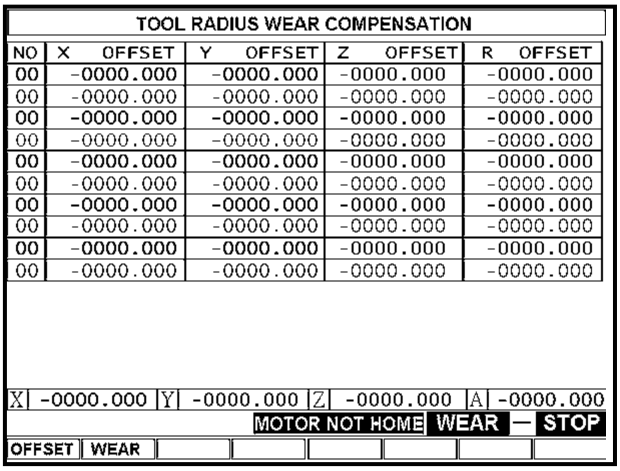
X -0000.000 Y -0000.000 Z -0000.000 A -0000.000

MOTOR NOT HOME OFFSET — STOP

OFFSET WEAR

Fig 2-19

2. Press the soft key “Wear” on the tool length compensation screen to enter the tool length compensation mode as shown in Fig 2-12 below: ※ (Refer to Section 3.15 for more information)



TOOL RADIUS WEAR COMPENSATION

NO	X OFFSET	Y OFFSET	Z OFFSET	R OFFSET
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000
00	-0000.000	-0000.000	-0000.000	-0000.000

X -0000.000 Y -0000.000 Z -0000.000 A -0000.000

MOTOR NOT HOME WEAR — STOP

OFFSET WEAR

Fig 2-20

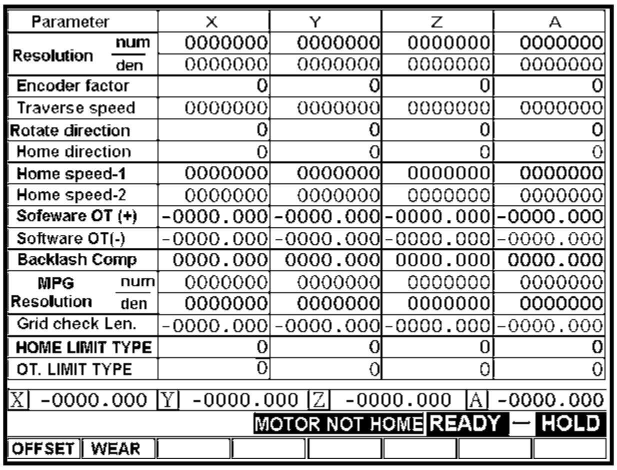
Follow the steps below to configure the parameters for tool wear compensation:

a. Use the "Cursor ⌄ ⌃" key to move the cursor to the parameter to be changed.

b. Enter the desired compensation value.

c. Press the "Enter" key.

3. Press the "MCM" key to enter the parameter screen as follows:



Parameter		X	Y	Z	A
Resolution	num	0000000	0000000	0000000	0000000
	den	0000000	0000000	0000000	0000000
Encoder factor		0	0	0	0
Traverse speed		0000000	0000000	0000000	0000000
Rotate direction		0	0	0	0
Home direction		0	0	0	0
Home speed-1		0000000	0000000	0000000	0000000
Home speed-2		0000000	0000000	0000000	0000000
Sofeware OT (+)		-0000.000	-0000.000	-0000.000	-0000.000
Software OT(-)		-0000.000	-0000.000	-0000.000	-0000.000
Backlash Comp		0000.000	0000.000	0000.000	0000.000
MPG Resolution	num	0000000	0000000	0000000	0000000
	den	0000000	0000000	0000000	0000000
Grid check Len.		-0000.000	-0000.000	-0000.000	-0000.000
HOME LIMIT TYPE		0	0	0	0
OT. LIMIT TYPE		0	0	0	0

X -0000.000 Y -0000.000 Z -0000.000 A -0000.000

MOTOR NOT HOME READY — HOLD

OFFSET | WEAR

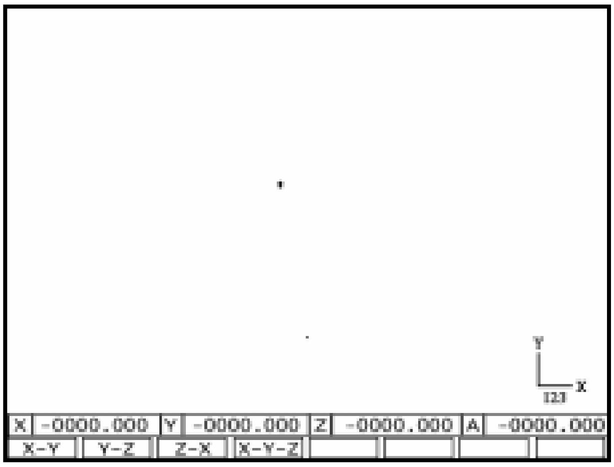
Fig 2-21

Enter **M9998** in the MDI mode and press the Start key. Then the parameter setting key displays on the tool length compensation screen.

* Note that changing the parameters without professional guidance may cause serious damage to the machine.

* Graph Mode Screen

Press the “Graph” key to enter the Graph mode. The following screen displays:



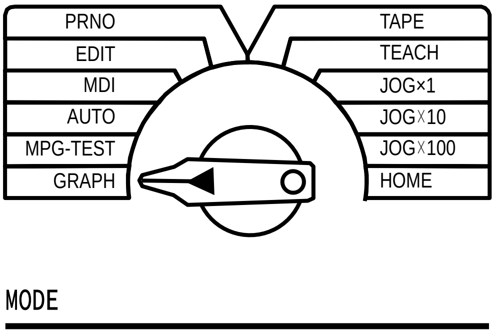


Fig 2-22

The “*” in the center of the screen indicates the zero position. It can be moved to other places with the **Cursor key**.

The current coordinate system displays at the bottom right corner of the screen. You can use the four keys, X-Y, Y-Z, X-Z, and X-Y-Z, at the bottom of the screen to select the coordinate system you need.

The number "123" at the bottom right corner of the screen represents the current horizontal ratio of the graph. The ratio can be changed using the Page Up and Page Down keys.

To clear the image, press the "Clear" key.

Note that the program automatically switches to Auto mode after entering graph mode.

* Hand-wheel Interruption

Description: This function is used to move all axes with the hand-wheel when the program suspends.

Operation: Press the “Dwell” key and the “Cursor Down” key. The “Hand-wheel

Interruption" displays on the screen. When you press the "Hand-wheel Interruption", it becomes highlighted and you can select an axis and operate the hand-wheel. When the positioning is complete, press the "Hand-wheel Interruption" again to cancel this function and the highlight displays accordingly. Press the "Start" key to proceed with the program.

2.2 Programming Overview

2.2.1 Part Programs

The movement of a numerical control machine is controlled by the program. Prior to part machining, the part shape and machining conditions must be converted to a program. This program is called a part program. A comprehensive machining plan is required for writing the part program. The following factors must be taken into account when developing the machining plan:

1. Determine the machining range requirements and select a suitable numerical control machine.
2. Determine the work-piece loading method and select appropriate tools and chucks.
3. Determine the machining sequence and tool path.
4. Determine the machining conditions, such as the spindle speed (S), feed rate (F), coolant, etc.

A part program is a group of sequential commands formulated according to the part diagram, machining plan, and command code of the numerical control unit. It is used to plan the tool path with the assistance of the auxiliary functions of the machine. The part program can be transmitted to the memory of the control unit via a PC or keyboard.

2.2.2 Programming Methods

A numerical control unit executes actions exactly in accordance with the commands of the part program. So, programming is very important to numerical control machining. A programmer must have the following capabilities:

1. Good capability of reading part diagrams.

2. Rich experience in machining processes.

3. Familiar with the functionality, operation procedure and programming language of the machine.

4. Basic capability in geometric, trigonometric, and algebraic operations.

5. Good capability of determination of machining conditions.

6. Good capability in setting chucks.

7. Good capability in determination of part material.

Two programming methods are available for the part program of the numerical control unit:

☐ Manual Programming
☐ Automatic Programming

Manual Programming

All processes from drawing of the part diagram, machining design, numerically controlled program algorithm, programming, to the transmission of the program to the controller are performed manually.

The coordinates and movement of the tool used in machining operations should be calculated first during the manual programming process. Calculation will be easier if the part shape is comprised of straight lines or 90-degree angles. For curve cutting, however, the calculation is more complicated and geometric and trigonometric operations are required for accurate curves. After acquiring the coordinate of the work-piece, create a complete numerically controlled part program in a specified format using the movement command, movement rate, and auxiliary functions. Check the program and make sure that there are no errors before transmitting it to the controller.

Automatic Programming

All processes, from the drawing of the part diagram to the transmission of the numerically controlled program to the controller are performed with a PC.

For complex part shapes, manually calculating coordinates is time-consuming and can easily have errors, resulting in nonconforming machined products. To make use of the high-speed operating capabilities of computers, the programmer designs a simple language to describe the machine actions and the shape, size, and cutting sequence of the part, reinforcing the communication and processing capability of the computer. The input data is translated into a CNC program using the computer, which will be in turn transmitted to the CNC controller via RS232C interface. It is called CAD/CAM system and used by many units using CNC machines to create a program especially for machining 3-D work-piece.

2.2.3 Program Composition

A complete program contains a group of blocks and a block has a serial number and several commands. Each command is composed of a command code (letter A~Z) and some numbers (＋.－.0~9). An example of a complete part program containing 10 blocks is shown in the table below. A complete program is assigned with a program number, such as O001, for identification.

A complete program:

```
N10 G00 X0.000 Y0.000 Z0.000
N20 M3 S1000
N30 G01 X10.000 Y10.000 Z10.000 F200
N40 X20.000
N50 Y20.000
N60 Z20.000
N80 G0 X0.000 Y0.000 Z0.000
N90 M5
N100 M2
```

Blocks are the basic units of a program. A block contains one or more commands. No space should be inserted between commands when transmitting a program. A block has the following basic format:

N___G___X(U)___Y(V)___Z(W)___F___S___D(H)___M___

N	:	The serial number of the block, which is not essential.
G	:	Function command.
X,Y.Z,A	:	Coordinate positioning command (absolute movement command).
U, V, W	:	Coordinate positioning command (incremental movement command).

F	:	Feed rate.
S	:	Spindle speed.
D, H	:	Tool number.
M	:	Auxiliary functions (machine control code).

Except for the block serial number (N), the command group of a block can be classified into four parts:

1. **Function Command:** The G-code, for example, is used to instruct the machine to perform actions, such as linear cutting or arc cutting.

2. **Positioning Command:** X, Y, Z, U, V, W commands, for example, instruct the tool of the machine to stop cutting at a specified position; i.e. destination or end point of the action.

3. **Feed Rate Command:** This command instructs the tool to cut (G-code) at a specified speed.

4. **Auxiliary Function:** The M, S, D, L commands, for example, determine the start, stop, spindle speed, tool selection, and execution times of the machine.

However, not every block contains these four commands. Some blocks have only one command. This will be further discussed in Chapter III.

Except for the block serial number of the block N___, all other components of the basic block format are commands. A command contains a command code (letter), a +/- sign, and some numbers.

<u>Basic Command Format</u> (e.g. the positioning command):

X-10.000

X : Command code

"-" : +/- sign (+ can be omitted)

10.000 : Destination of tool positioning action (the current position of the tool is the start point).

The command codes include the function command code, positioning (or coordinate) command code, feed-rate, command code, and auxiliary function command code. Each command code has its own definition and the machine behaves according to the command code given. The command codes of H4CL-M Series and their definitions are described below.

D	:	Tool radius compensation number.
F	:	Feed-per-rotation command.
G	:	Function code.
H	:	Tool length compensation number.
I	:	The X-axis component of the arc radius.
J	:	The Y-axis component of the arc radius.
K	:	The Z-axis component of the arc radius.
K, L	:	Repetition counters.
M	:	Machine control code.
N	:	Program serial number.
O	:	Program number.
P	:	Dwell time; call subprogram code; parameter in canned cycles.
Q	:	Parameter in canned cycles.
R	:	Arc radius or parameter in canned cycles.
S	:	Spindle speed.
U	:	Incremental positioning command on X-axis.
V	:	Incremental positioning command on Y-axis.
W	:	Incremental coordinates in Z-axis.
X	:	Absolute positioning command on X-axis.
Y	:	Absolute positioning command on Y-axis.
Z	:	Absolute positioning command on Z-axis.

Each block has a specified format and this format must be used during the programming. The system either does not accept an incorrect format. Major errors may occur if an incorrectly formatted command is accepted.

Each block has a serial number for identification. Though the serial number is not essential, it is recommended to use it for easy search. The serial number contains the letter “N” and some numbers. It is generated either automatically or by keying in from the keyboard when editing the program. (Refer to Chapter IV). The numbers should not be repeated and it is not necessary to be arranged in order. The program runs in order of blocks from top to bottom rather than their serial numbers. For example:

Ex: N10……(1) program execution order
N30…….(2)
N20…….(3)
N50…….(4)
N40…….(5)

2.2.4 Coordinate System

Fabrication of a work-piece is accomplished by the cutting motion of the machine-mounted tool. The tool can move in an arc or straight line. A coordinate system is used to describe the geometrical positions of the intersecting point and end point of an arc or line. The cutting action is done by the controlled change of these geometrical positions (positioning control)

2.2.4.1. Coordinate Axis

The H4CL-M Series uses the well-known 2-D/3-D Cartesian coordinate system. The 2-D coordinate system of the H4CL-M is X-Y, Y-Z, or Z-X, and X-Y is used as an example in this manual. As shown in the figure below, the intersecting point of X-Y is the zero point, i.e. X=0 and Y=0.

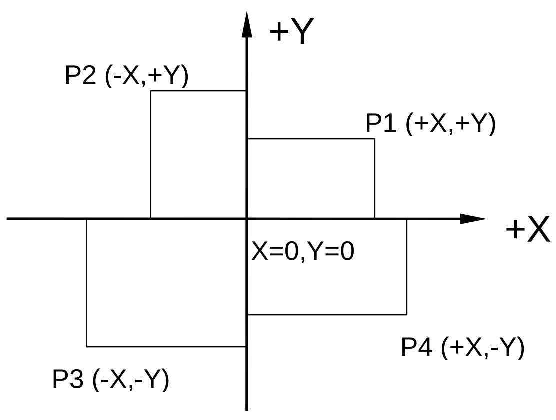


Fig 2-23

The 3-D coordinate system is a rectangular coordinate system (right hand rule) consisting of the X-axis, Y-axis, and Z-axis. A right angle is formed with the thumb, forefinger, and middle finger. The directions to which the thumb, forefinger, and middle finger point represent the X-axis, Y-axis, and Z-axis respectively. The intersection point of the axes is the zero point, indicating X=0, Y=0, Z=0. When you grasp an axis with your right hand, your thumb points to the positive direction of the axis and the rest four fingers point to the direction of its normal rotation.

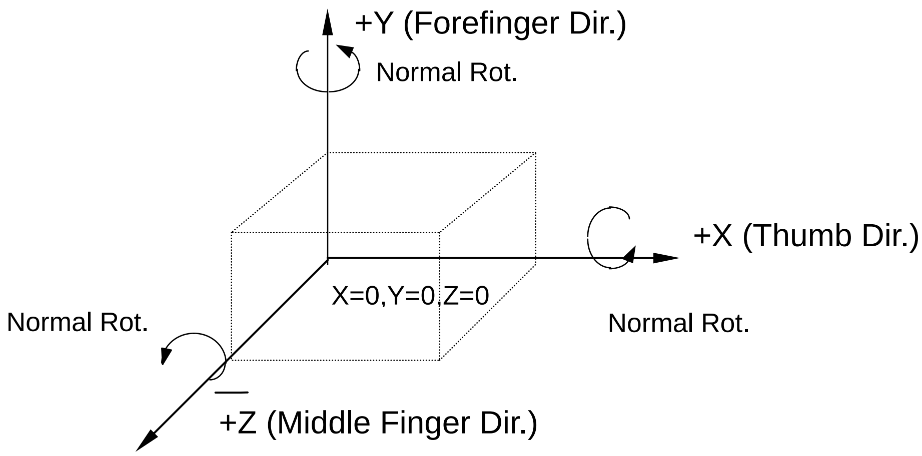


Fig 2-24 3-D Coordinate System (Horizontal Milling Machine)

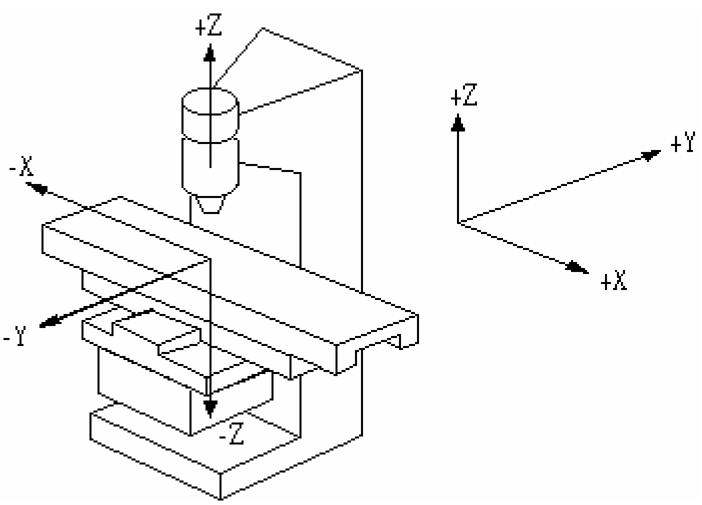


Fig 2-25 3-D Coordinate System (Vertical Milling Machine)

<u>The Z-axis in the 3-D coordinate system should be parallel to the spindle.</u> (<u>The spindle is the rotating axis and the tool is clamped.</u>) After the Z-axis is set, the X-axis and Y-axis can be determined using the rectangular coordinate system (the right hand rule) in Fig. 23.

2.2.4.2. Coordinate Positioning Control

The coordinates of the H4CL-M Series controller are either absolute or incremental, depending on the command code of the coordinate axis, i.e.:

X, Y, Z, A: Absolute coordinate commands.

U, V, W : Incremental (or decremental) coordinate commands.

Absolute Coordinate Commands

Tool-positioning coordinates are acquired with reference to the origin (work origin or program origin) of the work coordinate system. The coordinates are either positive (+) or negative (-), depending on its position relative to the origin.

Incremental Coordinate Commands

The previous coordinates of the tool are the reference point for calculating the coordinate value of the next position. The end point of the previous movement is the start point of the next movement. The incremental coordinates are either positive (+) or negative (-). The negative sign represents decrement. Facing toward the direction of the movement, if the tool is heading to the positive (+) direction, U, V, W represents increment. If it is heading to the negative (-) direction, U, V, W represents decrement.

X, Y, Z, and U, V, W are interchangeable in the program. The commands used for the absolute and incremental coordinates are described as follows:

Absolute Commands:

N10 G00 X0.000 Y0.000	...	Move to the work origin rapidly
N20 G90	...	Set the program to absolute coordinates
N30 G1 X12.000 Y12.000 F300.00	...	P0 to P1
N40 X26.000 Y16.000	...	P1 to P2
N50 X38.000 Y30.000	...	P2 to P3
N60 M2	...	Program ends

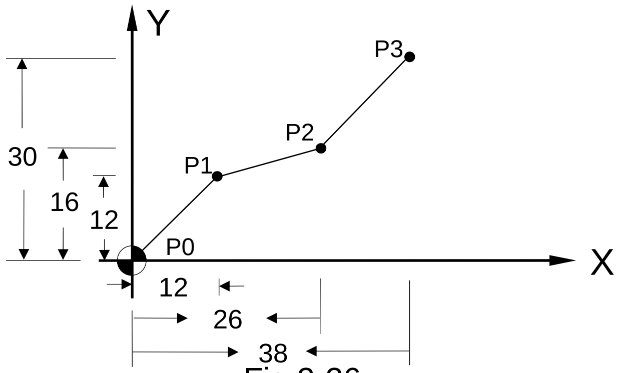


Fig 2-26

Incremental Commands:

N10 G00 X0.000 Y0.000	...	Move to the work origin rapidly
N20 G91	...	Set the program to absolute coordinates
N30 G1 X12.000 Y12.000 F300.00	...	P0 to P1
N40 X14.000 Y4.000	...	P1 to P2
N50 X12.000 Y14.000	...	P2 to P3
N60 M2	...	Program ends

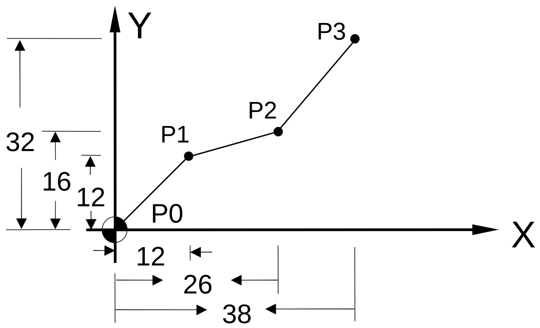


Fig 2-27

Note that the "U, V, W" commands will be invalid when the command G91 is used to set the program coordinates "X, Y, Z" to increment. The incremental command U, V, W is only valid when the G90 absolute coordinate command is used. The default of the H4CL-M series is in absolute coordinates. In this case, the above-mentioned program can be re-written as:

N10 G00 X0.000 Y0.000	...	Move to the work origin rapidly
N20 G1 U12.000 V12.000 F300.00	...	P0 to P1

N30 U14.000 V4.000	...	P1 to P2
N40 U12.000 V14.000	...	P2 to P3
N60 M2	...	Program ends

Coordinate Interchange:

N10 G00 X0.000 Y0.000	...	Move to the work origin rapidly
N30 G1 X12.000 V12.000 F300.00	...	P0 to P1
N40 X26.000 V4.000	...	P1 to P2
N50 X38.000 V14.000	...	P2 to P3
N60 M2	...	Program ends

or

N10 G00 X0.000 Y0.000	...	Move to the work origin rapidly
N30 G1 U12.000 Y12.000 F300.00	...	P0 to P1
N40 U14.000 Y16.000	...	P1 to P2
N50 U12.000 Y30.000	...	P2 to P3
N60 M2	...	Program ends

Simultaneous use of absolute and incremental coordinate systems in a part program is possible. For the absolute coordinate system, the input error of the previous position, if any, does not affect the coordinate of the next point. For the incremental coordinate system, however, all subsequent positioning is affected if the previous position is incorrect. Therefore, particular attention should be paid when using incremental coordinates.

There aren’t any rules about when to use the incremental or absolute coordinate system. It depends on the machining requirements. If each machining point is positioned relatively to the home position, it is recommended to use the absolute coordinate system.

For the command of the diagonal (simultaneous positioning on the X and Y-axis) or arc movement, the coordinate value of each axis acquired with the trigonometric operation will be rounded off. In this case, particular attention should be paid when the incremental coordinate system is used, as machining points may increase, and the more points it has, the more errors will occur. Basically, whether an absolute or incremental coordinate is used depends on the programming requirements and the specifications of the machining diagram.

2.2.4.3. Work Origin

The specifications of the machining diagram are converted to the coordinate system at the CNC programming stage. Before the conversion, a point on the work-piece is selected as the zero point of the coordinate system (i.e. work origin) and the coordinates of other points on the work-piece are calculated based on this work origin.

The programmer determines the position of the work origin. It can be any point on the chuck or the table of the milling machine. However, it is recommended to select an origin that makes reading the work-piece coordinates more easy.

The work origin is also called work zero point or program origin. In this manual, this zero point is always referred to as the work origin.

2.2.4.4. Machine Origin

There is a fixed point on the machine bed or bed rail. This point is used as a reference point for determination of the work coordinates (or work origin) and calibration of the tool length compensation. This reference point is called machine origin.

For the H4CL-M Series controller, the machine origin is the stop position of the X-axis, Y-axis, and Z-axis when the homing action on each axis is completed. In general, the machine origin is determined based on the position where the positioning measurement device and the touch plate of the limit switch are installed on the machine.

The homing action should be performed after powering on the machine. If the current position is lost due to power failure, the homing action should be performed again.

2.2.5 Numerical Control Range

The numerical and functional control range of the H4CL-M controller is described in the following two tables.

Min. setting unit	0.001 mm
Max. setting unit	8000.000 mm
Min. moving distance	0.001 mm
Max. moving distance	8000.000 mm
Min. Time	0.1 sec.
Max. Time	8000.000 sec.

The limits in the above table are applicable to 4/3-formats.

(The max. moving distance reaches up to 80000.00 mm when 5/2-format is used)

G code	G00~G99 (G01=G1)
M code	M00~M999 (M01=M1)
S code	S1~S9999 rpm
F code	0.01 ~ 80000.00 mm/min
X, Z, U, W, I, K, R	0.001~+/-8,000.000 mm
R (Radius)	0.001~+/- 4000.000 mm
G04 Time Setting	0~8000.000 seconds
Memory capacity	128 K
Lead screw compensation	0~255 pulses (related to tool resolution)
Max. Response Speed	500 KPPS

The numerical control range varies depending on the specifications of the numerical control unit. Refer to the operator’s manual of the machine for more information about the machine.

2.3 Program Editing

Program editing operation includes:

1. Program selection,
2. New program editing, and
3. Existing program modification.

2.3.1 Program Selection

The H4CL-M controller can store programs numbered O0~O999. You can select any one of the programs to edit or execute.

Program selection: Double-click the “EDIT/PRNO” key to enter the PRNO mode and open the file directory page on the LCD (Fig. 2-28). Move the cursor to the desired program and press the “Input” or “Select” key.

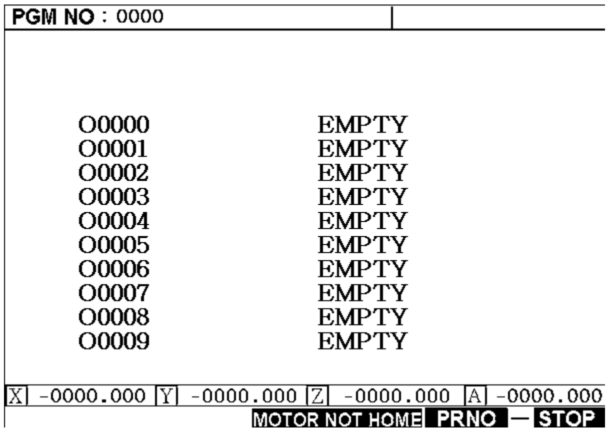


Fig 2-28

You can enter the program comments in this mode with a maximum of 12 characters.

Example: To add the comment “ TYPE-201” to O001:

1. Move the cursor to O001
2. Press T Y P E 2 — 0 1
3. Press Enter

3 Programming and Command Codes

3.1 Types of Command Codes

This chapter definitely describes the command codes of H4CL-M series and provides simple examples for each command to explain its applications.

The definition of G-codes in the H4CL-M series is similar to other controllers. They are classified into two groups: (Table 3-1).

3.1.1 One-shot G-code

A One-shot G-code (has no * mark in the table) is valid only in the specified program block.

Ex: N10 G0 X30.000 Y40.000 . . . G0 is Modal G-code

N20 G4 X2.000 . . . G4 is a one-shot G-code and only valid in this block.

N30 X20.000 Y50.000 . . . No G-code specified; G0 code of the N10 block is valid here.

3.1.2 Modal G-code

A Modal G-code (has a * mark in the table) is valid until it is replaced by another G-code of the same group.

<u>G00, G01, G02, G03</u> Same group

<u>G17, G18, G19 </u>Same group

<u>G40, G41, G42 </u>same group. <u>G43, G44, G49 </u>same group.

<u>G54, G59</u> Same group

<u>G80, G89</u> Same group <u>G90, G91</u> Same group

<u>G98, G99 </u>Same group

Ex: N10 G0 X30.000 Y5.000 . . . G0 is specified.

N20 X50.000 Z10.000 . . . No G-code specified, G0 remains valid.

N30 Y20.000 . . . No G-code specified, G0 remains valid.

N30 G1 X30.000 F200 . . . G1 replaces G0 and becomes valid, The G-codes of H4CL-M controller are listed in Table 3-1. Only one G-code of the same group can be set for one program block. If more than one G-code is set, only the last G-code is valid.

Table 3-1 H4CL-M Code Definitions

G-code List			
G- code	Function	G- code	Function
* 00	Fast positioning (fast feeding)	* 40 #	Tool radius compensation cancellation
* 01 #	Linear cutting (cutting feeding)	* 41	Tool radius compensation setting (left)
* 02	Arc cutting, CW	* 42	Tool radius compensation setting (right)
* 03	Arc cutting, CCW	* 43	Tool length compensation (+) direction
04	Dwell command (the interval is determined by X-axis)	* 44	Tool length compensation (-) direction
08	Clear the machine coordinate of each axis	* 49	Tool length compensation cancellation
10	MCM data input		
15	Servo spindle positioning	% 50 *	Proportion function cancel
* 17	Thread cutting, X-Y	% 51 *	Proportion function setting
* 18	Thread cutting, Z-X	* 54 #	First work coordinates
* 19	Thread cutting, Y-Z	* 55	Second work coordinates
		* 56	Third work coordinates
* 20	Measurement in INCH mode	* 57	Fourth work coordinates
* 21	Measurement in METRIC mode	* 58	Fifth work coordinates
		* 59	Sixth work coordinates
22 $	Linear grooving		
23 $	Arc grooving	* 68	X-axis mirror-effect cutting
24 $	Rectangular grooving	* 69	Y-axis mirror-effect cutting
25 $	Circular grooving		
		* 80 $	Drilling canned cycle cancellation
28	Tool moves to the 1st reference point	* 81 $	Drilling canned cycle setting

29	Return to the previous position from the ref. point	* 82 $	Drilling canned cycle (dwell at bottom)
30	Tool moves to the 2nd reference point (a total of 10 groups)	* 83 $	Deep hole drilling canned cycle
		* 84 $	Tap Cutting canned cycle
31	%Skip function	* 85 $	Boring canned cycle
		* 86 $	Boring canned cycle (spindle stop at hole bottom)
34 $	Circular drilling canned cycle	* 89 $	Boring canned cycle with dwell at hole bottom)
35 $	Angular linear drilling canned cycle	* 90	Absolute coordinate command
36 $	Arc drill canned cycle	* 91	Incremental coordinate command
37 $	Grid drilling canned cycle	* 98	Feed-rate specified by mm/min
		* 99 #	Feed-rate specified by mm/revolution

- * -- Modal G-codes
- # -- Default settings upon power-on of the controller
- $ -- Special functions of H4CL-M Series.
- % -- Optional functions

3.2 Fast Positioning, G00

Format:

G00 X(U)____Y(V)____Z(W)____

X, Y. Z : Positioned end point in absolute coordinate.

U,V. W : Positioned end point in incremental coordinates relative to the block starting point.

G00 (or G0) is used to instruct the tool to move to the specified end point of a program block at the maximum speed of MCM #79(X-axis) ,#80(Y-axis) ,#81(Z-axis). The start point is the position at which the tool is located before it moves. This command can control the movement of 1~3 axes simultaneously. The axis that is not set by the command does not execute any movement.

The path of the tool movement is straight. Where the distance of movement is different among axes, the controller selects the axis that has the longest movement distance for fast positioning. The feed-rate of other axes is determined based on their movement distance and the components of the axis with the longest movement distance. If the calculated speed of any axis exceeds the MCM setting value, the controller will re-calculate the feed-rate of other axes based on the feed-rate of the overrun axis.

Ex: Fig 3-1 S point moves to E point rapidly.

G90
G00 X100.000 Y50.000 Z20.000

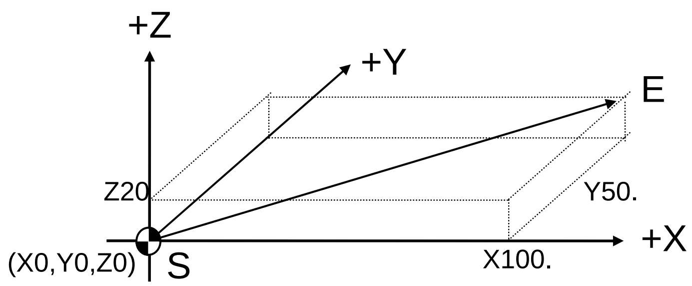


Fig 3-1 G00 Programming Example

The distances to move are 100.000 (X-axis), 50.000 (Y-axis), and 20.000 (Z-axis). Since the movement distance of each axis is different, the controller selects the axis with the longest movement distance for fast positioning at the speed set in MCM #79. Assume that the speed of MCM #79 is set to 10000 mm/minute, the movement speed of each axis in the above figure is :

X-axis – Movement distance 100.000mm. Since X-axis moves farthest, the controller specifies 10000 of MCM #148 as the feed-rate of the X-axis.

Y-axis – Movement distance 50.000mm. It is divided by the distance of the longest movement distance 100.000mm and multiplied by the highest feed-rate 10000 of MCM #149 to acquire 5000 (i.e. 50.000/100.000*10000=5000)

The actual feed-rate of Y-axis is 5000 mm/min

Z-axis – Movement distance 20.000mm. It is divided by the longest movement distance 100.000mm and multiplied by the highest feed-rate 10000 of MCM #150 to acquire 2000 (i.e. 20.000/100.000*10000 = 2000)

The actual feed-rate of Z-axis is 2000 mm/min.

3.3 Linear Cutting, G01

Format:

G01 X(U)____Y(V)____Z(W)____A____F____

X,Y,Z,A : End point in absolute coordinates

U,V,W : End point in incremental coordinates relative to the start point of the program block.

F : Cutting feed-rate (F-code can be used in combination with any G-code)

The F-code can be used in the G00 block without affecting the fast positioning movement.

Start point:The position at which a tool is located before moving.

G01 (or G1) is used for linear cutting work. It can control the X, Y, Z axes simultaneously. The cutting speed is determined by the F-code. The smallest setting value of the F-code is 0.02 mm/min or 0.2 in/min.

The starting point is the coordinate of the tool when the command is given. The feed-rate specified after an F-code (Modal code) remains valid until it is replaced by a new feed-rate.

After the feed-rate (F-code) is determined, the cutting feed-rate of X, Y, Z, and A axes is calculated as follows:

(U, V, and W are actual incremental values.)

$$\text{X feed-rate, } Fx = \frac{U}{\sqrt{U^2 + V^2 + W^2}} * F \quad (1)$$

$$\text{Y feed-rate, } Fy = \frac{V}{\sqrt{U^2 + V^2 + W^2}} * F \quad (2)$$

$$\text{Z feed-rate, } Fz = \frac{W}{\sqrt{U^2 + V^2 + W^2}} * F \quad (3)$$

Three G01 programming examples are described below. These programs are different in settings but execute the same linear cutting work.

4 G90 absolute program- fig3-2

N1 G90

N2 G01 X25.000 Y20.000 Z10.000 F100.00 ... P1

N3 X60.000 Y50.000 Z40.000 ... P2

5 G91 incremental program (Fig 3-2)

N1 G91

N2 G01 X25.000 Y20.000 Z10.000 F100.00 … P1

N3 X35.000 Y30.000 Z30.000 … P2

6 G90 increment program (Fig 3-2)

N1 G90

N2 G01 U25.000 V20.000 W10.000 F100.00 … P1

N3 V35.000 V30.000 W30.000 … P2

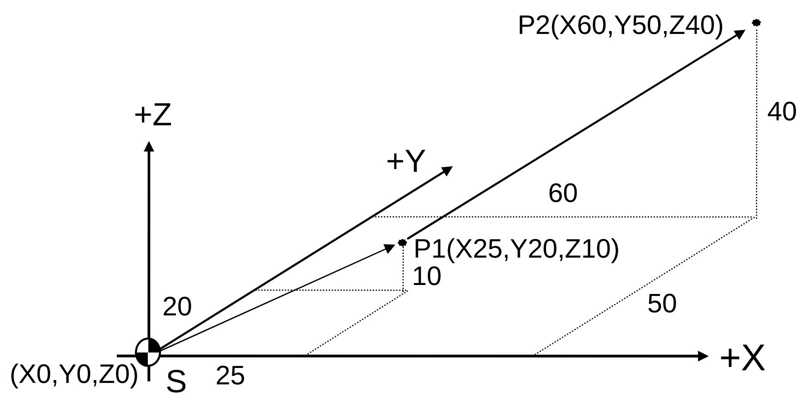


Fig 3-2 G01 Programming Example

3.4 CNC and Master/Slave Mode

When the part program is running, every block has a feed-rate (F), including the G0 block. When a feeding command is given in the CNC mode, the motor starts accelerating to the specified feed-rate. It maintains this speed and decelerates to zero when the tool approaches to the positioning point. When a feeding command is given to the next block, the motor repeats the acceleration and deceleration actions. The speed of the motor is reset to zero between blocks.

Master/Slave mode— In the master/slave mode, an axis is selected as the master axis and the rest axes are automatically set to slave axes. The motor speed of the master and all slave axes remains at the feed-rate and is not reset between blocks. In case that two adjoining blocks have different feed-rates, the motors of the master and all slave axes perform the acceleration and deceleration actions and the motor speed is adjusted to the feed-rate of the next block without being reset to zero. If the feed-rate of the master axis is zero, the controller will select the feed-fate of the slave axes.

MCM #93 is used to set CNC and master/slave modes: 0 = CNC mode, 1 = master/slave mode with the X-axis as the master, 2 = master/slave mode with the Y-axis as the master, 3 = master/slave mode with the Z-axis as the master, 4 = No-dwell Mode. The CNC and Master/Slave modes are exemplified below.

CNC Mode: MCM #93 is set to 0

In the CNC mode, the speed of the motor decelerates to zero at the end point of each block. The acceleration/deceleration of the motor is determined by MCM #124.

MCM parameter	MCM parameter	G00 Acc./Dec.	G01,G02 Acc./Dec.
Parameter#93=0	Parameter#222=0	Linear	Exponential Curve
Parameter#93=0	Parameter#222=1	S curve	S curve

Ex 1: Fig 3-3 shows the feed-rate adjustment between blocks when the G01 command is given in CNC mode. Acceleration/deceleration of the motor is executed in an exponential curve. The coordinates in this example are absolute coordinates.

N05 G00 X0. Y0. Z0

N10 G01 X100. F1000.

N20 G01 X200. Y100. Z50. F500

N30 G01 X300. F250.

N35 G01 X350. F100.

Explanation: N10 -- X-axis feed-rate F1000; Y-axis and Z-axis 0 feed-rate

N20 -- Same X and Y increment (100) with the same F500 feed-rate; Z increment = 50 with F250 feed-rate

N30 -- X feed-rate F250; Y and Z feed-rate 0

N35 -- X feed-rate F100; Y and Z feed-rate 0

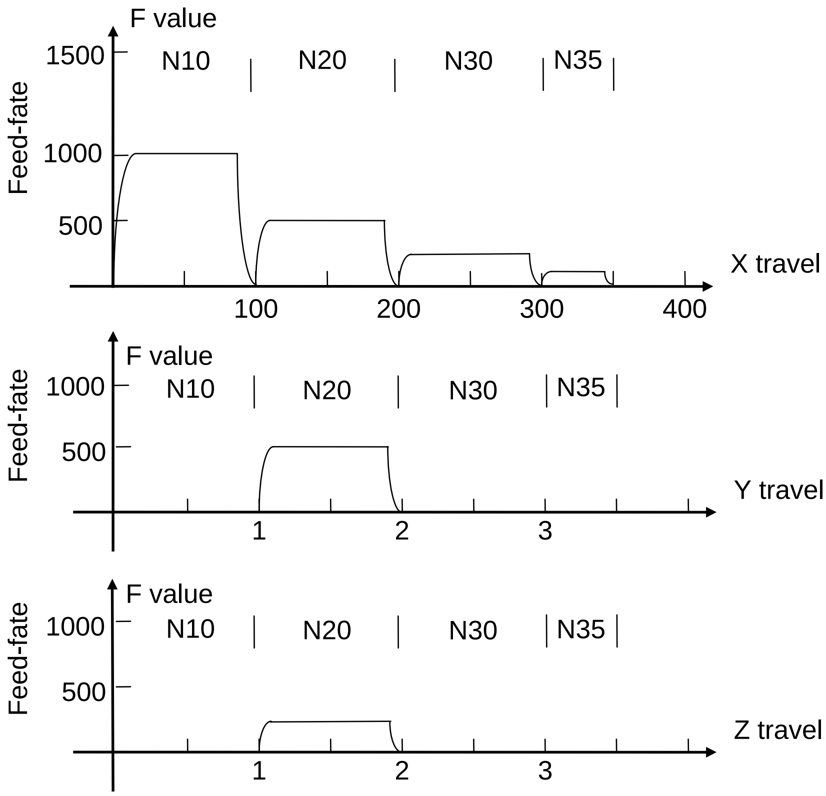


Fig. 3-3 CNC Mode with G01- exponential curve Acc. / Dec.

Ex.2 and Ex.3 show how to calculate X and Y feed-rate in CNC mode using formulae (1) and (2). In the examples, assuming that the highest feed-rate set in G00 MCM #148~151 of G00 is:

TRX=2000 mm/min (X-axis), TRY=1000 mm/min (Y-axis)

Ex. 2: G1 U100.0 V50.0 F1500

U, V composite vector = $(100^2 + 50^2)^{1/2}$ = 111.8, so

X feed-rate, Fx =(100/111.8) * 1500= 1341.6

Y feed-rate, Fy =(50/111.8) * 1500=670.8

Both axes are within the G00 parameter and, thus, valid for feeding.

Ex. 3: G1 X100.0 Y200.0 F2000

X, Y composite vector =$(100^2 + 200^2)^{1/2}$=223.6,so

X feed-rate, Fx =2000 *(100/223.6)= 894.4

Y feed-rate, Fy = 2000 *(200/223.6)=1788.9

Since Fy > TRY (1000), the feed-rate is limited to:

Fx = (894.4/1788.9) * 1000 = 500

Fy =(1788.9/1788.9) * 1000=1000

Master/Slave Mode: MCM #93 is not set to 0

If MCM #93 is set to 1 with the X-axis as the master and the other axes as slaves, the speed between blocks is not reset to 0 but adjusted to the feed-rate of the next block. The specified rate of a single block(F) is the feed-rate of the master axis. The controller adjusts the rate of the slave axes based on the rate of the master axis and MCM parameters. The example below demonstrates this relationship, assuming that the feed-rate does not exceed the G00 value. The acceleration/deceleration type of the motor is determined by MCM #222.

MCM parameter	MCM Parameter	G00 Acc./Dec. type	G01,G02 Acc./Dec. type
Parameter#93=1, 2,3,4	Parameter#222=0	Linear	Linear
Parameter#93=1, 2,3,4	Parameter#222=1	S curve	S curve

Ex 1: N10 G01 X100. F1000.

N20 X200. Y100. Z50. F500.

N30 X300. F250.

X-master axis, Y, Z-slave axes. The feed-rate of each block depends on the master axis, The feed-rate of each slave axis (Y, Z) adjusts according to the incremental ratio of X/Y, X/Z. The motor accelerates or decelerates linearly. The acceleration/deceleration status between blocks is shown in Fig 3-4.

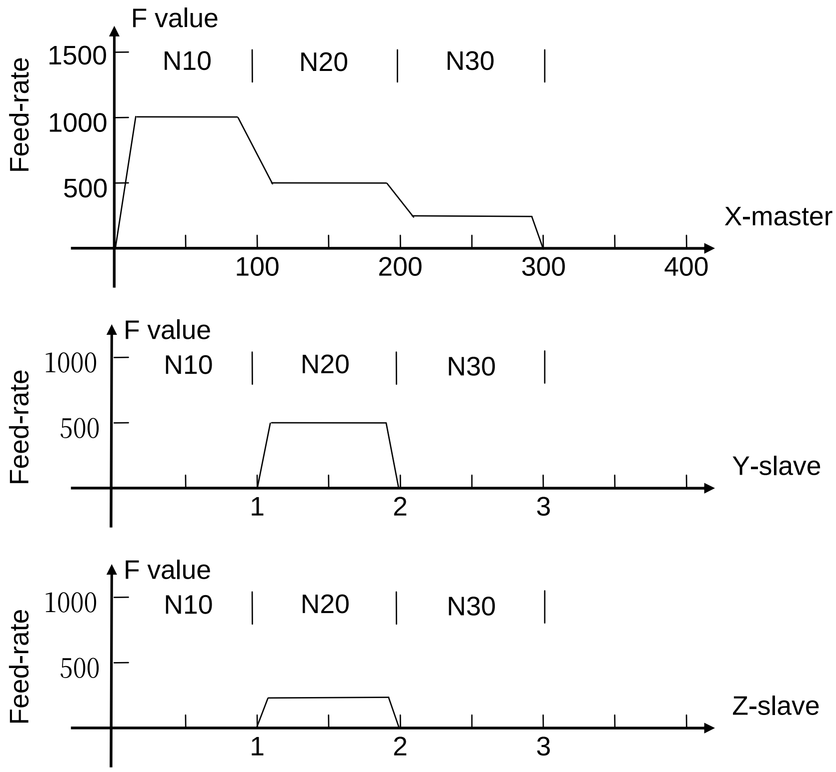


Fig 3-4 Master/ Slave Mode - linear Acc./Dec.

If the motor accelerates or decelerates in "S" curve, the acceleration/deceleration status between blocks is shown in Fig 3-4A:

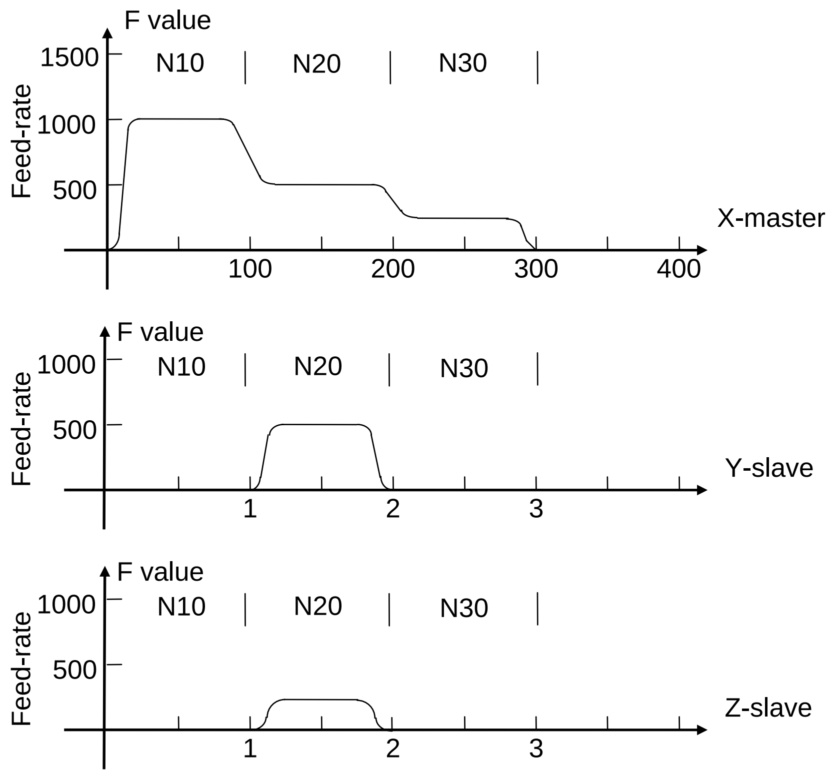


Fig 3-4A Master/ slave mode-S curve

Ex2: As shown in Fig 3-5, X is the master while Y and Z are the slaves. The feed-rate of the master axis (X) in each block doesn't change, but the feed-rate of the slave axes (Y, Z) changes along with the incremental slope ratio.

N10 G01 X100 Y50 Z0 F1000

N20 X200 Y75 Z50

N30 X300 Y175 Z100

In Example 2, the feed-rate of Y-slave changes along with the incremental slope rate of X, Y and is not reset to zero. Since both increments of the Z-axis are 50, the feed-rate remains the same. Note that there is a small interval between blocks during acceleration/deceleration. (Fig. 3-5)

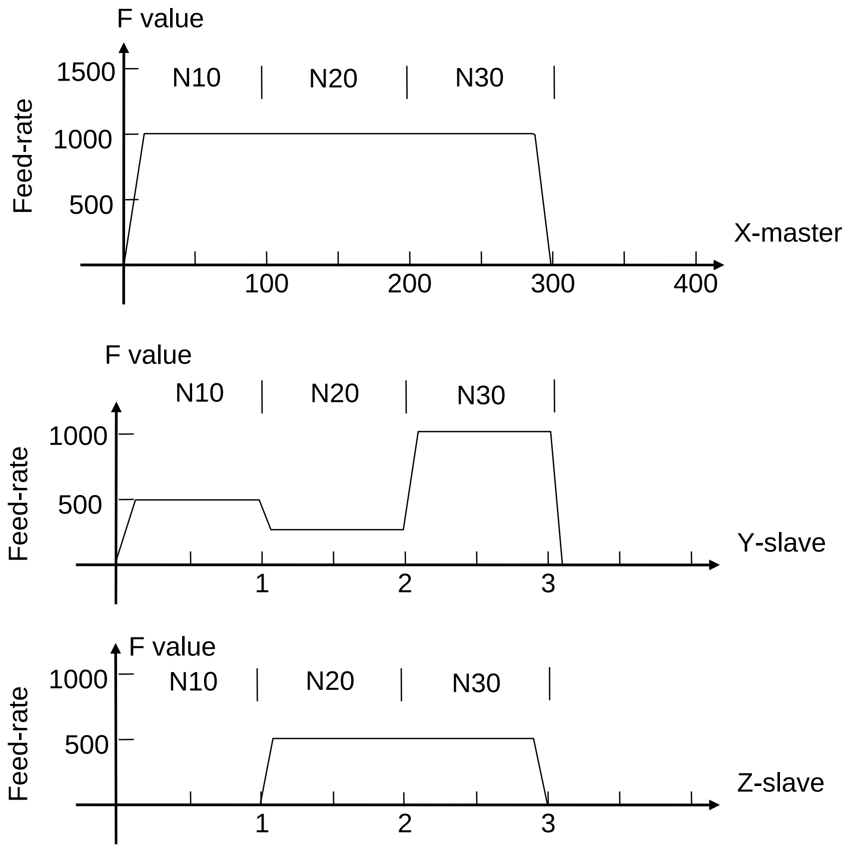


Fig. 3-5 Mater/Slave mode – Master Speed unchanged

The examples below show how to calculate the feed-rate of the master and slave axes (assuming MCM 148(TRX) = 2000, MCM 150(TRY) = 4000 mm/min). The relationship between the feed-rate and the max. feed-rate setting (MCM #148 – 151) is taken into consideration during the calculation.

Ex. 3: G0 V100.0 V50.0 (X-axis is the master; MCM #93 is set to 1)

Master feed-rate   Fx = 2000

Slave feed-rate Fy =(50/100) * 2000.00=1000

Fy < TRY (4000),

So, the feed-rate is determined by the TRX value of MCM#148 (X-axis).

Ex. 4: G0 U100.0 V300.0 (X-axis as Master, MCM#93=1)

Master feed-rate Fx = 2000

Slave feed-rate Fy =(300/100) * 2000=6000

Fy > TRY (4000), thus the speed is limited to:

Master feed-rate Fx =(4000/6000) * 2000= 1333.33

Slave feed-rate Fy = 4000

The feed-rate is determined by the TRY value of MCM#149 (Y-axis).

3.5 Arc Cutting, G02, G03

Format:

G17 <--- power-on default

G02 (or G03) X(U)____ Y(V)____ I____ J____ F____ (1)

(R can replace I,J)

G18

G02 (or G03) X(U)____ Z(W)____ I____ K____ F____ (2)

(R can replace I,K)

G19

G02 (or G03) Y(V)____ Z(W)____ J____ K____ F____ (3)

(R can replace J,K)

The three command groups control arc cutting on the X-Y (1), X-Z (2), Y-Z (3) plane. The dimension is controlled by G17, G18, and G19, and G17 is the default power-on dimension. When executing arc cutting in the X-Y dimension, G17 can be omitted. The function of G17, G18, and G19 will be described in the next sections. The format of these command groups is a special thread cutting (refer to the next sections) format. Arc cutting is then executed when the linear axis does not move during the thread cutting. Definitions of other commands are described below:

X(U), Y(V), Z(W).

The end point coordinates of arc cutting. The start point is the coordinates of the tool when G02 or G03 execute.

I, J, K and R

I, J and K are the increment or decrement from the start point of the arc to the center of the circle. If the coordinates from the start point to the center of the circle are incremental, the value is positive. Otherwise, it is negative. The definition of this increment/decrement is the same as the incremental commands U, V, and W. All these commands can be replaced by the R command.

F: The feed-rate for arc cutting is determined by F-value. The minimum value is 1mm/min

The path and the direction of the tool are determined by G02, G03 and G17~19(Fig 3-6).

G02: Clockwise (cw)

G03: Counterclockwise (ccw)

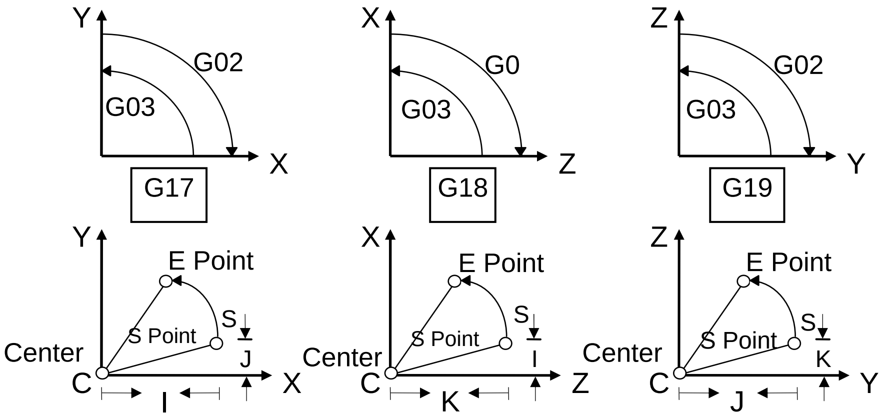


Fig 3-6 Arc cutting

If the angle of an arc is between –1°~1° or 179°~181°, I, J, K cannot be replaced by R. Fig 3-7 is the example showing the replacement of the I, J, K value with the radius R-value.

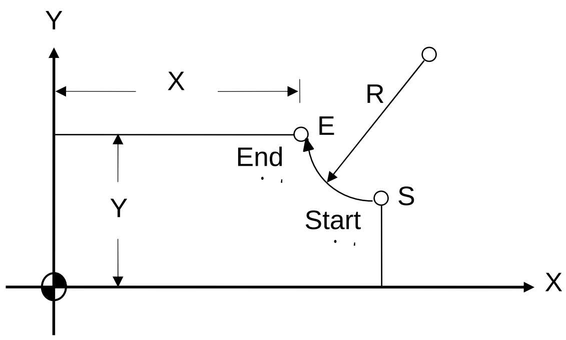


Fig 3-7 Arc Cutting Indicated by the Radius R Value

As shown in Fig. 3-8, R-value is either positive (+) or negative (-) during the arc cutting. R-value ranges from –4000. mm to +4000.mm.

1. R values must be <u>positive</u> when an arc <u>less than</u> 180° is cut

2. R values must be <u>negative</u> when an arc <u>greater than</u> 180° is cut

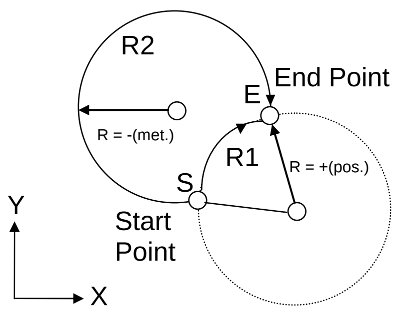


Fig 3-8 Arc cutting (+/- R)

<u>Programming Example</u>: The following four commands are different in settings but execute the same arc cutting work.

Start point X=50.000, Y=15.000

End point X=30.000, Y=25.000

Radius R=25.000, or I=0.000, J=25.000

1. G02 X30.000 Y25.000 J25.0000 F200.
2. G02 U-20.000 V10.000 J25.000 F200.
3. G02 X30.000 Y25.000 R25.000 F200.
4. G02 U-20.000 V10.000 R25.000 F200.

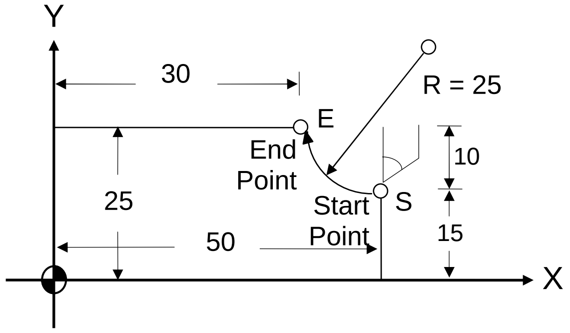


Fig 3-9 G02 Programming Example

When cutting a full circle, only the I, J, K values, rather than the R-value, can be used.

Ex.: G90

G00 X40.000 Y0.000

G03 X40.000 Y0.000 I50. F100.

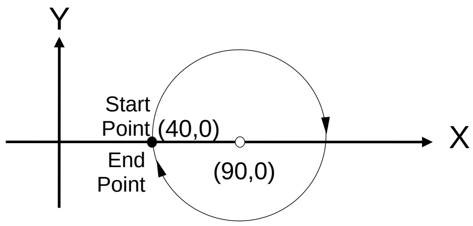


Fig 3-10 Cutting a Full Circle

Please note the followings when executing an arc cutting:

1. The F value (i.e. the feed-rate) of G02, G03 is the tangential cutting speed. This speed is subject to the radius of the arc and the F value of the program because H4CL-M system uses a fixed 1μm chord height error. (Chord Height Error is the maximum distance between the arc and chord)

2. When the calculated tangential cutting speed of the arc is greater than the F value of the program, the F-value is used as the tangential cutting speed. Otherwise, the calculated value prevails.

3. The maximum tangential cutting speed is estimated with the following formula:

   $Fc = 85 * \sqrt{R * 1000}$ mm/min

   Where R= Arc radius in mm.

3.6 Servo spindle positioning command, G15

Format:

G15 R___

R : Servo spindle position

Description

1. This G-code is only applicable to the servo spindle.
2. Ranging 0.000°~359.999°

3.7 Thread Cutting, G17, G18, G19

This command is set as an independent block before the arc cutting command. It executes an arc cutting on a plane specified by G17, G18, G19 and perform a linear cutting on a third axis along the path same as the path of a constant-diameter spring. The dimension of the arc cutting is determined by G17, G18, G19 and the size of the arc are determined by G02, G03 plus the end coordinates of the linear cutting. The tool radius compensation function is only available for the specified cutting plans. Details are described below.

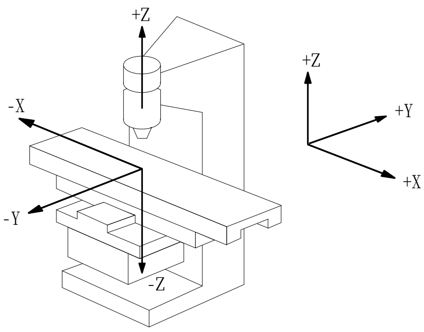


Fig 3-11 X, Y, Z-axes 3D Diagram

G17, X-Y Arc Cutting Plane

As shown in Fig 3-12, if you look down at the machine from the above (along the Z-axis toward the negative direction), you have the X-Y arc cutting plane with Z-axis as the linear axis. Clockwise is G02 and counter-clockwise is G03.

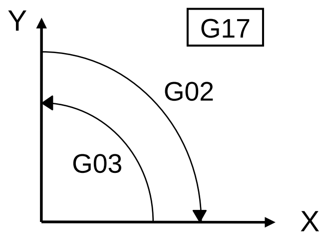


Fig 3-12

Format: (Refer to Fig 3-6 and Fig 3-12)

N1 G17

N2 G02 (or G03) X___Y___ I____ J____ Z____ F____

(R can replace I, J)

G18, Z-X Arc Cutting Plane

If you look at the machine from the back (along the Y-axis toward the negative direction), you have a Z-X arc cutting plane with Y-axis as the linear axis. Clockwise is G02 and counter-clockwise is G03.

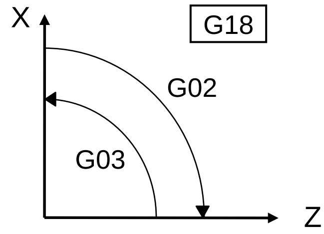


Fig 3-13

Format: (Refer to Fig 3-6 and Fig 3-13)

N1 G18

N2 G02 (or G03)Z___X___ K____ I____Y____F____

(R can replace K,I)

G19, Y-Z Arc Cutting Plane

If you look at the machine from the right side (along the X-axis toward the negative direction), you have Y-Z arc cutting plane with X-axis as the linear axis. Clockwise is G02 and counter-clockwise is G03.

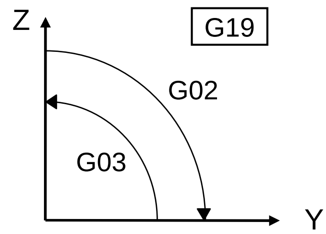


Fig 3-14

Format: (Refer to Fig 3-6 and Fig 3-14)

N1 G19

N2 G02 (or G03)Y___Z___ J____ K____X____F____

(R can replace J,K)

Ex: X-Y arc cutting plane with Z-axis as the linear axis

N1 G17

N2 G03 X80.000 Y30.000 R30.000 Z40.000 F100

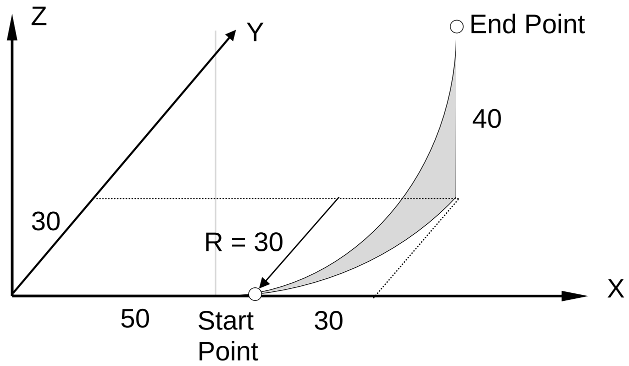


Fig 3-15

3.8 Dwell Command, G04

Format:

G04 X____

X: Dwell time in seconds (the X here indicates time rather than coordinates).

To meet machining requirements, the axial movement may need to hold for a while when the execution of a program block is completed before the command for the next block is executed. This command can be used for this purpose.

The minimum dwell time is 0.01 second. It can be set up to 8000.0 seconds.

Ex.: N1 G1 X10.000 Y10.000 F100.

N2 G4 X2.000. . . Hold 2 seconds,

N3 G0 X0.000 Y0.000

3.9 Clear Machine Coordinates, G08

Format:

	G08	...	Clear machine coordinates for all axes, X, Y, Z
or	G08 X____Y____	...	Clear the machine coordinates of X and Y axes
or	G08 Z___	...	Clear the machine coordinates of Z-axis.
or	G08 X___ Y___ A__	...	Clear the machine coordinates of X, Y, A axes.

or any combination of X, Y, Z, A.

X, Y, Z values in the format are meaningless. They only indicate the machine coordinates of the axis to be cleared. If G08 is set to an independent block, the X, Y, and Z machine coordinates are cleared. If X (or Y or Z) command is given, only the machine coordinates of that axis will be cleared regardless of its value.

3.10 Data Settings, G10

Table 3-2 HUST H4CL-M G10 Command Code List

G10 Command Code List	
G10 X＊＊ Y＊＊ Z＊＊ A＊＊	Set the work origin on the G54~G59 work coordinates system
G10 X＊＊ Y＊＊ Z＊＊ P＊＊	Set the tool length compensation
G10 P200 L＊＊＊	Set the counter limit, L=MCM #170
G10 P201	Clear the counter limit, MCM #169=0
G10 P510 L4800	Set the baud rate of RS232 interface on the controller to 4800
G10 P510 L9600	Set the baud rate of RS232 interface on the controller to 9600
G10 P510 L19200	Set the baud rate of RS232 interface on the controller to 19200
G10 P600 L01	Burn the downloaded part program into FLASHROM
G10 P600 L02	Burn the downloaded MCM parameters into FLASHROM
G10 P600 L03	Burn the downloaded ladder program into FLASHROM
G10 P600 L05	Burn the downloaded system data into FLASHROM
G10 P800 L＊＊＊	Set G01 Accel./Decel. time, MCM#166
G10 P1000	Load MCM parameters from FLASHROM
G10 P2000	Clear the current program of the controller

G10 P2001	Clear all programs in the memory of the controller
G10 P2002	Clear all variables #1 ~ #9999 to zero
G10 P2100	Load the part program from FLASHROM to memory.

3.10.1 Set the Work Origin Using G10 (Recommended) ,G10

Set the work origin on the G54~G59 work coordinate system using G10 command. The user may use the MDI key on the HUST H4 CNC controller or execute the function through build-in PLC by customization.

Format:

G10 X____Y____Z____A____. Select an axis or all three axes.

Steps for setting the work origin (G54~G59) using G10:

1. Return to Home manually.
2. Enter JOG mode.
3. Move the tool to the desired position where the work origin is to be set.
4. Enter the MDI mode, input G54, and press CYCST.

5A. If the coordinates of the tool in Step 3 is the desired position for the work origin, do the following:

Press G10 Input,

X0. Input,

Y0. Input,

Z0. Input,

Press the CYCST key to finish the setting.

5B. If the coordinates of the tool in Step 3 is at some distance (say X=20, Y=100, Z=15) away from the desired work origin, do the following:

Press G10 Input,

X20. Input,

Y100. Input.

Z15.   Input.

Press the CYCST key to finish the setting.

The following precautions should be observed when using G10 to set the work origin.

1. Do not add P__ to the G10 block, otherwise, it becomes a tool length (movement) compensation command.

2. The same procedure is applicable to the G55~G59 coordinate system, except that G54 is replaced by G55~G59 in Step 4. If no coordinates from G54 to G59 are specified in step 4, the work origin data will be entered into the currently valid work coordinate system.

3. The G10 command can also be applied in the program.

4. When G54~G59 is selected by G10, the machine position data of the origin will be entered into MCM #1~#36.

3.10.2 Set the Tool Length Compensation Using G10

Format:

1. G10 X____ Y____ Z____ P1__

2. G10 U____ V____ W____ P1__

3. G10 I____ J____ K____ P1__

P1_ :1~40 represents the tool group number in MCM #9000~#9399.

X, Y,Z :Setting the tool length compensation data to the corresponding X, Y, Z of MCM #9000~#9199.

U, V, W :Setting the tool wear compensation data to the corresponding U, V, W of the MCM #9200~#9399.

I, J, K :<u>Adding</u> the tool wear compensation data to the corresponding I, J, K of the MCM #9200~#9399.

P –tool group number	X -axis Tool length compensation	Y -axis Tool length compensation	Z -axis Tool length compensation	R -Tool radius compensation
1	VAR9000	VAR9001	VAR9002	VAR9003
2	VAR9005	VAR9006	VAR9007	VAR9008
3	VAR9010	VAR9011	VAR9012	VAR9013

4	VAR9015	VAR9016	VAR9017	VAR9018
.	.	.	.	.
.	.	.	.	.
39	VAR9190	VAR9191	VAR9192	VAR9193
40	VAR9195	VAR9196	VAR9197	VAR9198

P –tool group number	X -axis Tool wear compensation	Y -axis Tool wear compensation	Z -axis Tool wear compensation	R -Tool radius wear compensation
1	VAR9200	VAR9201	VAR9202	VAR9203
2	VAR9205	VAR9206	VAR9207	VAR9208
3	VAR9210	VAR9211	VAR9212	VAR9213
4	VAR9215	VAR9216	VAR9217	VAR9218
.	.	.	.	.
.	.	.	.	.
39	VAR9390	VAR9391	VAR9392	VAR9393
40	VAR9395	VAR9396	VAR9397	VAR9398

Ex 1: Execute command G10 X0.02 Y0.03 P101→set the length compensation value for the first tool group.

>> MCM#9000 = 0.02, A MCM#9001 = 0.03

Ex 2: Assume: The original MCM#9205~9207 settings are X=0.02, AY=0.03, AZ=1.25

Execute G10 U0.01 V0.02 W1.72 P102 → Set the tool wear compensation value for the second tool group.

>> MCM#9205 = 0.01, MCM#9206 =0.02, MCM#9207 =1.72

Ex 3: Assume: The original MCM#9205~9207 settings are X=0.02, AY=0.03, AZ=1.25

Execute G10 I0.01 J0.02 K1.72 P102 → Add the tool wear compensation value to the second tool group.

>> MCM#9205 = 0.02+0.01 = 0.03

MCM#9206 = 0.03+0.05 = 0.05

MCM#9207 = 1.25+1.72 = +2.97

3.10.3 Set and Clear the Counter Limit Using G10

Counter Limit Data Setting

Format:

G10 P200 L___ --P200 is a fixed value while L__ is used to set up the counter limit data of MCM #170.

Ex: G10 P200 L100 -- MCM#170=100

Set up the maximum counter limit for MCM #170. When the program runs, the actual cycles are recorded in MCM #169. P200 is a fixed value indicating that G10 is used to set up the counter limit here. The maximum value of L is 9,999,999. If L=0, the counter of MCM #169 is unlimited.

Whenever a cycle is complete (encountering M02, M30 or M99), the number in MCM #169 increases by 1. When the number increases up to the set value of MCM #170, the cycle stops.

Counter Limit Data Clearing

Format:

G10 P201—P201 is a fixed value to clear the counter limit of MCM #169 to zero.

3.10.4 Set G01 Acceleration/Deceleration Time Using G10

The acceleration/deceleration time is stored in MCM #166. This setting can be adjusted using one of the following 3 methods.

1. Change the setting directly in the MCM EDIT mode.
2. Execute G10 P800 L___ in the MDI mode
3. Change the setting by executing the work program in AUTO mode.

Note: The "RESET" key must be pressed before the new setting is valid.

Format:

G10 P800 L____ -- Set the G01 acceleration/deceleration time (msec) of MCM #166

Ex1: Change the G01 acceleration/deceleration time in the MDI mode.

Step 1: Double-click AUTO to enter the MDI mode

Step 2: Execute the command G10 P800 L100

G - 1 – 0 INPUT

P - 8 - 0 – 0 INPUT

L - 1 - 0 – 0 INPUT

Step 3: Click CYCST

Step 4: Press the "RESET" key to make the new setting valid.

Result>> MCM #166=100

Ex2: Adjust G01 acceleration/deceleration time based on the travel distance in the AUTO mode,

A/D time = 100 milliseconds if 200.000 ≧ #1

A/D time = 50 milliseconds if 100.000 < #1 ≦ 200.000

A/D time = 30 milliseconds if #1 ≦ 100.00

Step 1: Edit the work program 0001

O001

N001 G65 L85 P005 A#1 B100

N002 G10 P800 L30

N003 M02

N005 G65 L85 P008 A#1 B200

N006 G10 P800 L50

N007 M02

N008 G10 P800 L100

N010 M02

Step 2: Enter AUTO mode and execute O001

Result>>Since #1 = 120, MCM #166=50 (m/sec)

3.11 Imperial/Metric Measuring Modes, G20, G21

Format: G20 -- System measurements use Imperial units.

G21 -- System measurements use Metric units.

3.12 Return to the First Reference Point, G28

Format:

G28

or G28 X____Y____Z____

The first reference point coordinates are set based on the X, Y, Z, A settings in MCM #94~97. The X, Y, Z, A values in this format are meaningless. They only indicate which axis is to return to the reference point. Therefore, regardless of whether G28 is an independent block or contains X, Y, Z, A commands simultaneously, the tools return to the reference point based on the X, Y, Z, A settings in MCM #94~97.

One to three axis coordinate commands can be specified after G28, and the tool returns to the reference point of the corresponding axis set in the MCM #94-97 accordingly no matter what the value of the command is. The axis, which is not specified via the command, does not execute any motion. The examples of axial coordinate commands are shown as follows. Users can set the axes when required.

G28	Three axes return simultaneously.
G28 X____	One axis returns.
G28 X____Y_____	Two axes return simultaneously.

Note that prior to executing the G28 command, the tool compensation command must be canceled.

Ex: G40 Tool compensation is canceled (it can not co-exist with G28 in the same block)

G28 X10. Tool returns to the 1st reference point on the X-axis, while the Y and Z axes do not move.

3.13 Return to Previous Position from Reference Point, G29

Format:

G29 X____Y____Z____

The X, Y, Z, A values in this format are meaningless. They only indicate the set of axes to return to the previous position from the reference point. When the tool returns to the position before G28 is executed, use the G29 command. This command cannot be used separately. It must be executed following the G28 or G30 command.

Ex: N1 X60. Y0. Z30. . . . Tool moves to the position X60, Y0, Z30.

N2 G28 . . . Tool returns from X60, Y0, Z30 to the 1st reference point.

N3 G29 . . . Tool returns from the reference point to X60, Y0, Z30

Like the G28 command, one to three axis coordinate commands can be specified after G29, and the tool returns to the position before G28 is executed regardless of the command value. The axis, which is not specified via the command, will not execute any motion. Examples of axial coordinate commands are shown as follows. Users can set the axes when required.

G29 . . . Three axes return simultaneously.

G29 X____ . . . One axis returns.

G29 X____Y____ . . . Two axes return.

3.14 Return to the Second (2nd) Reference Point, G30

Format:

G30 X____Y____Z____

Execution of this command is same as G28, but the reference point is set in MCM #100~103.

3.15 Skip Function, G31

Format:

G31 X(U)____Y(V)____Z(W)____

X, Y, Z: : Predicted end point in absolute coordinates.

U,V,W: : Predicted end point in incremental coordinates relative to the starting point.

To ensure valid skip function G31, it must be used in combination with an I/O signal. G31 functions same as G01 until the skip function is established' i.e. G31 executes linear cutting in the X, Y, Z coordinates. Once an I/O signal is detected during cutting, the G31 skip function establishes and the block G31 skips from the current operation to the next block.

When G31 is performing linear cutting, the feed-rate is determined by the currently effective F-value (G00 or G01). G31 is a one-shot G-code and only valid in the specified block.

Ex:

N40 G40

N50 G31 U100.000 F100.

N60 G01 V25.000

N70 X90.0 Y30.

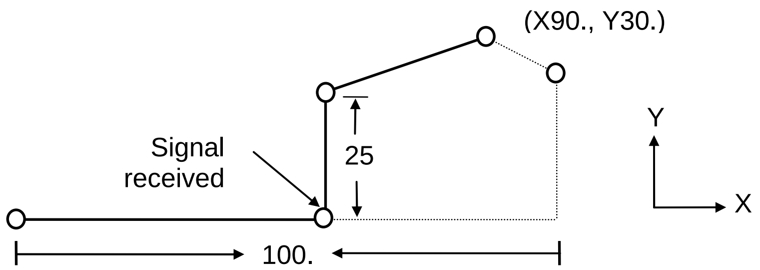


Fig 3-16 G31 skip function

In Fig 3-16, the dotted line represents the original path without the Skip function and the solid line is the actual tool path when the Skip function signal is received.

Note that G31 cannot be used in the tool radius compensation state. G40 must be executed to cancel the tool radius compensation before G31 can be used. The Skip function is invalid during program dry run, feed-rate adjustment or auto acceleration/deceleration.

3.16 Tool Compensation

The tool compensations of HUST H4CL-M CNC have three types The data of tool compensation are store in the tool length compensation and tool radius wear compensation, and can store 40 tool compensation data. These data can be called by G41, G42, G43, G44 commands. Use G40, G49 to cancel the compensation if required.

1. Tool radius wear compensation
   To compensate the error in x or y-axis resulting from tool radius wear after use
   This compensation is usually used in combination with the tool radius compensation. The compensation data are stored in the length wear radius compensation.

2. Tool length radius compensation
   To compensate the error in the tool axis (Z-axis) resulting from differences in tool lengths.
   The compensation data are stored in the tool length radius compensation.

3.16.1 Tool radius and radius wear compensation, G40, G41, G42

Format:

G41 D___ X___ Y___	Tool radius compensation - Left
G42 D___ X___ Y___	Tool radius compensation - Right
G40	Tool radius compensation - cancel

D : Tool number of tool radius and radius wear compensation, no.1~40

X, Y : Insert the coordinates of tool radius compensation.

Description

Where the tool-tip is used to cut along the profile of the work-piece during the execution of the part program, an over-cutting of the radius will occur on each processing path. With the tool radius compensation function, a tool radius value can be offset based on the actual travel of the tool and the specified path of the command to ensure that the processing result conform to the specifications of the drawing. Therefore, correct product size can be ensured by writing the work program based on the specifications of the drawing and the compensation function of the system. The tool radius does not need to be taken into account for the program.

The tool size of a milling machine varies significantly from 1mm to 50mm, and the tool radius compensation G41, G42 can be used to ensure that the tool cuts along the profile of the design plan.

Whether G41 or G42 is used depends on the relative position between the tool direction and the tool-tip. To the direction of the arrow in Fig. 3-17, G42 is used when the central point of the tool radius is located at the right side of the tool path (radius offset to the right). G41 is used when the central point radius is located at the left side of the tool path (radius offset to the left). G41 and G42 are Model G-codes and can only be cancelled using G40.

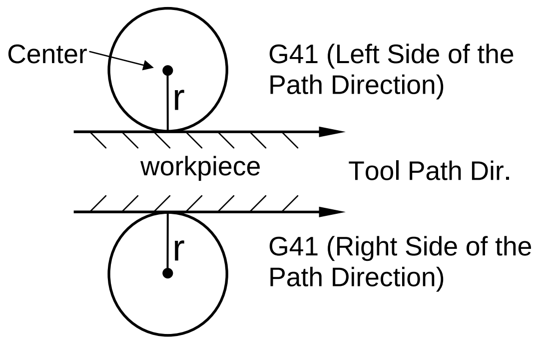


Fig 3-17 G41 and G42 Applications

Execution of Tool Radius Wear Compensation

Tool radius wear compensation is executed in the same way as the tool radius compensation is. When the G41/G42 command is calling the tool number for radius compensation using the N-code, the HUST controller simultaneously selects the tool radius length and radius wear compensation values for the called toll number and compensation the program.

Ex: The D3 tool compensation value is

Radius compensation=2.000 mm, radius wear compensation=-0.010mm

Tool radius compensation=2.000-0.010=1.990mm

Please note that the radius wear compensation value is input with a (-) sign. The radius compensation and radius wear compensation are only valid on the X, Y plane, not on the Z-axis.

3.16.2 The Initial Setting of the Tool Radius Compensation

When G41, G42 are executed, the tool will make linear motion to the X, Y coordinates specified in the G41, G42 blocks at G01 speed (or the X, Y, Z JOG speed of MCM#112, 113, 114). When reaching the specified X, Y coordinates, the tool-tip will shift at a distance equivalent to the tool radius. The start setting of the G41, G42 commands are only available in the G00 or G01 linear cutting model. The system will send an error message if it is executed in the G02, or G03 arc cutting mode. A simple description of the tool start setting function is given below:

1. Tool radius compensation is executed when the tool travels from A to B. Insertion of the radius compensation is complete at B.

   N1 G01 F200

   N2 G41(G42) D______ X______ Y______

N3 X______

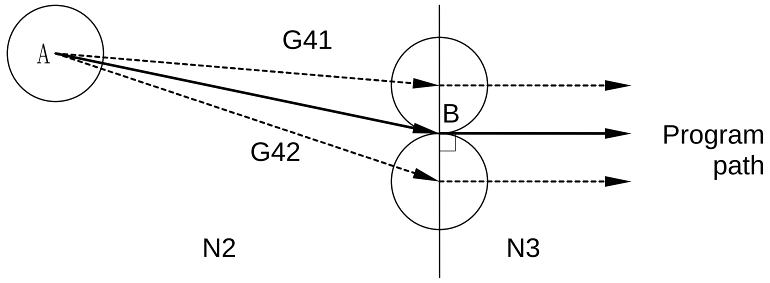


Fig 3-18 Radius compensation-1

2. Radius compensation is complete at the start point (B) of the arc cutting.

N1 G01 F200.00

N2 G41(G42) N______ X______ Y______

N3 G02 X______ Y______ J______

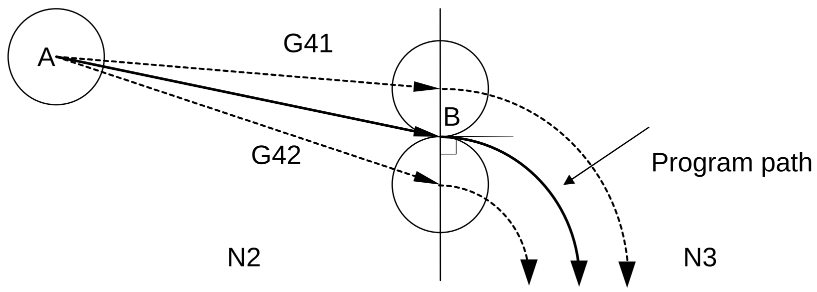


Fig 3-19 Radius compensation-2

3.16.3 Relationship between Radius Compensation and Tool Path

G41, G42 are Modal G-code command, so when the insertion of the G41, G42 in the tool path is complete and before they are cancelled by G40, the tool-tip do a vector offset to the amount of the tool radius value “r” along the program path. Calculation of the path is executed automatically for the tool-tip path. When the direction of the program path changes, the path of the tool-tip also changes and special attention must be paid to the corner of the changed path. Different corners are described as follows:

1 Inside corner ($\theta \leqq 180^\circ$)

A small part of the work-piece can’t be cut from the inside corner (Fig 3-20, P Point).

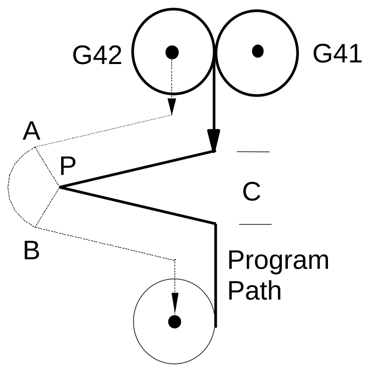


Fig 3-20

2 Outside corner (θ>180°)

The tool-tip moves in an arc motion (Fig. 3-21, P point) along the inside corner to create a new path.

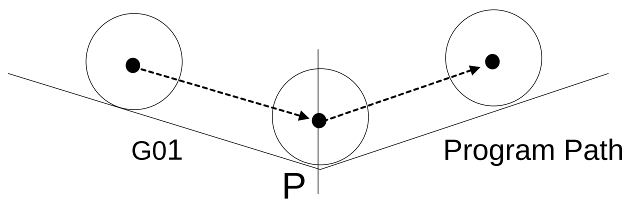


Fig 3-21

If the angle is convex as shown in Fig 3-22, the corner cut is correct while the correctness of cutting on the inside corner depends on the distance from the opening C. If the distance is less than the tool diameter, no cutting is possible. If it is greater than the tool diameter, the tool cuts toward the inside corner, part of the work-piece cannot be cut in the sharp inside corner.

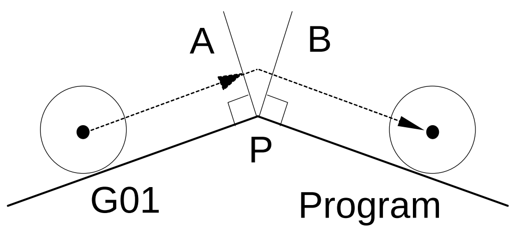


Fig 3-22

3. Compensation Direction Change

HUST H4CL-M <u>does not accept</u> the direct change of compensation direction from G41 to G42 or from G42 to G41. Where changing of the direction is required, the compensation must be cancelled using G40 before the direction can be changed.

4. Tool Radius Change

Like the direction change, HUST H4CL-M does not accept the direct change from one tool number to another for radius compensation. Where changing of the tool number is required, the compensation must be cancelled using G40 before tool number can be changed.

3.16.4 Tool Radius Compensation – Cancellation

Once G41 or G42 is executed successfully, G40 command must be used to cancel the tool radius compensation. The movement during the cancellation of the radius compensation can only be executed in the G00 or G01 mode. G40 is not directly available for G02, G03 blocks and the cancellation can be executed only after the arc cutting is executed successfully. Below are some examples of the cancellation of tool radius compensation

1. N20 G41(G42) D___ ....

   . . .

   . . .

   N31 G01 X_____ F______

   N32 G40 X_____ Y______

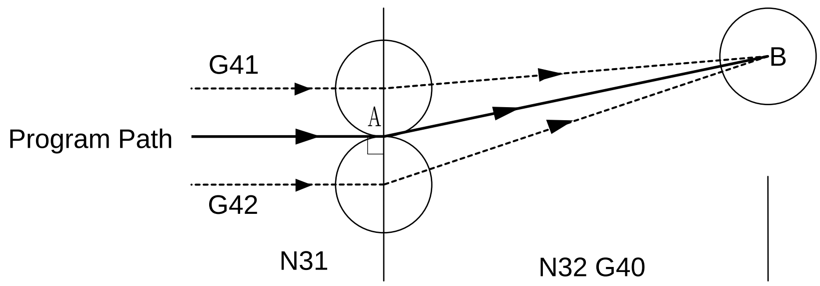


Fig 3-23

2. N10 G41(G42) D___ ....

   . . .

   . . .

   N15 G02 X____ Y____ I____ J____ F____

   N20 G01

   N25 G40 X____ Y____

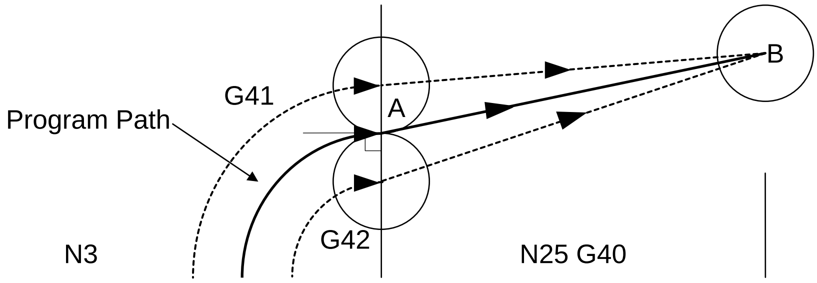


Fig 3-24

3.16.5 Notes on Tool Radius Compensation

1. When cutting around an inside corner, the arc radius of the inside corner must be equal to or greater than the tool radius (r). Otherwise an alarm will generate an alarm signal. The arc cutting around an outside corner is not subject to this regulation.

2. G41, G42 commands are not applicable to canned cycles (G80~G89). They must be cancelled using G40 before a canned cycle can be executed.

3. Where an arc cutting command exists during the tool radius compensation (G41,42), the writing method of the radius value “R” is applicable.

4. Where multiple axes are controlled simultaneously, the tool radius compensation of the HUST H4CL-M is only valid on the X, Y plane not on the Z- axis.

5. The tool radius compensation function is not available for MDI operation.

6. When cutting a stepwise work-piece with a step value smaller than the tool radius, over-cutting many occur as shown in Figure 3-26.

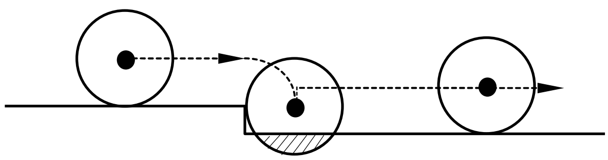


Fig 3-26 Over-cutting (Shaded area)

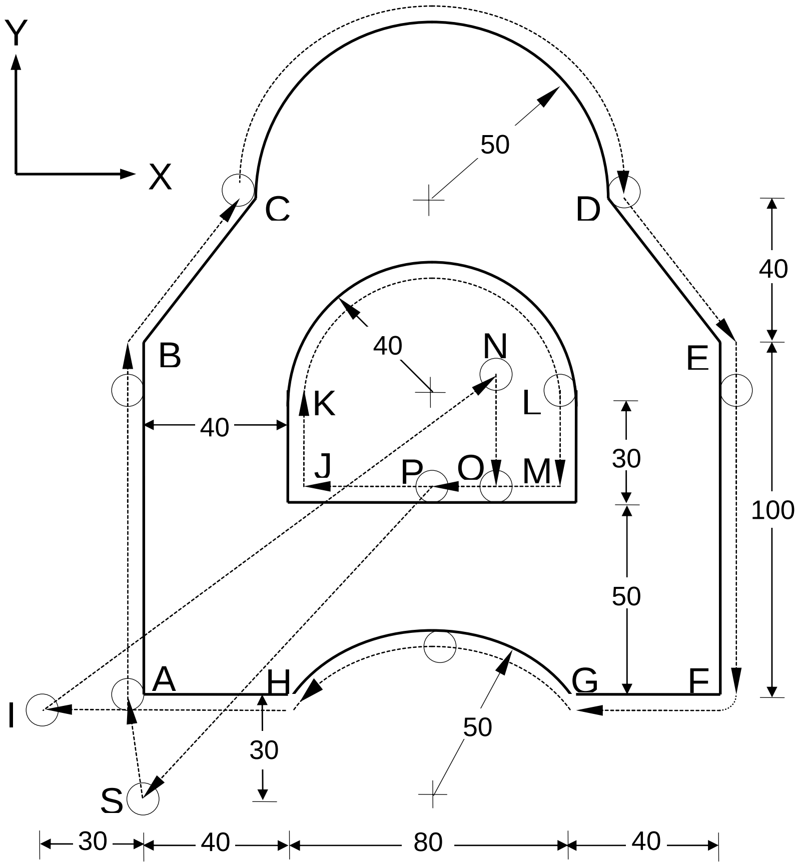


Fig 3-27 Programming examples

<u>Programming Examples of Tool Radius Compensation</u>:

N1 G91	...	Incremental coordinates setting
N2 G01 Z-2.500 F150.	...	Z-axis cutting by 2.5mm
N3 G17 F300.	...	X-Y cutting plane setting
N4 G41 D10 Y30.000	...	Point A, initial setting of tool radius compensation
N5 Y100.000	...	Linear cutting from A~B
N6 X30.000 Y40.000	...	Linear cutting from B~C
N7 G02 X100.000 I50.000	...	Half-circle cutting from C~D
N8 G01 X30.000 Y-40.000	...	Linear cutting from D~E
N9 Y-100.000	...	Linear cutting from E~F
N10 X-40.000	...	Linear cutting from F~G

N11 G03 X-80.000 R50.000	...	Arc cutting from G~H
N12 G01 X-70.000	...	Linear cutting from H~I
N14 Z2.500	...	Z-axis rising by 2.5mm
N15 G40	...	Compensation cancelled, ready for direction change
N16 M01	...	Program suspension.
N17 G0 X130. Y90. F200.	...	Positioning to N
N18 G01 Z-2.500 F150.	...	Z-axis cutting by 2.5mm
N19 G42 Y-40.000 F300.	...	Linear cutting from N~O Compensation direction change
N20 X-60.000	...	Linear cutting from O~J
N21 Y30.000	...	Linear cutting from J~K
N22 G02 X80.000 I40.000	...	Arc cutting from K~L
N23 G01 Y-30.000	...	Linear cutting from L~M
N24 X-60.000	...	Linear cutting from M~P
N25 Z2.500	...	Z-axis rising by 2.5mm
N26 G40 X-60.00 Y-80.00	...	Tool compensation cancelled; tool returning to S point.
N27 M02	...	Program end.

3.16.6 Tool length compensation, G43,G44,G49

Tool length compensation is available for the position of Z-axis to correct the error of the tool length. The length compensation data up to 40 sets are stored in the tool length compensation. Refer to 3.3.9 for entering the length compensation data using G10.

Format:

	G43(G44) Z_____ H_____	Length compensation setting
or	G43(G44) H_____	Length compensation setting
	G49	Length compensation cancellation

Z : Initial compensation coordinates

H : Tool number for which the length compensation is executed.

Explanation: Different tools are used for processing of work-pieces on a milling machine or machining center. Since the length is different among tools, the distance from the tool-tip to the work-piece varies to a significant extent. When the tool is changed during the execution of the program, the difference in the length of the tools before and after the change will cause an error in Z-axis. The purpose of the tool length compensation (G43/G44) is to correct the error of the tool length along the Z-axis.

Length Compensation Setting:

Method 1: Manually move a tool downward from the machine origin of the Z-axis until it touches the surface of the work-piece. Measure the distance of the movement and enter the tool length compensation value for each tool number. Set the tool number required for compensation within the H value of the command format.

Method 2: Choose a tool via the operation interface of the controller and calibrate its length in the G54 work coordinate system. This tool will be used as a reference for determination of the length difference and compensation value of other tools.

When G43 is executed, the controller selects the specified compensation value and adds it directly to the Z-axis.

When G44 is executed, the controller selects the specified compensation value and adds it to the Z-axis for compensation after changing direction.

Compensation direction is defined based on the direction of the Z-axis. A positive compensation means that the tool moves positively along the Z-axis after compensation. A negative compensation means that the tool moves negatively along the Z-axis after compensation. The relationship between the compensation direction and the positive/negative value of the length compensation under the G43/G44 command is described as follows:

	MCM, positive value	MCM, negative value
G43	Positive compensation	Negative compensation
G44	Negative compensation	Positive compensation

Ex 1: N1 G00 Z0.000

N2 G0 X1.000 Y2.000

N3 G43 Z-20.000 H10 (Length compensation-3.000)

N4 G01 Z-30.000 F200

N5 G49 Z0.000

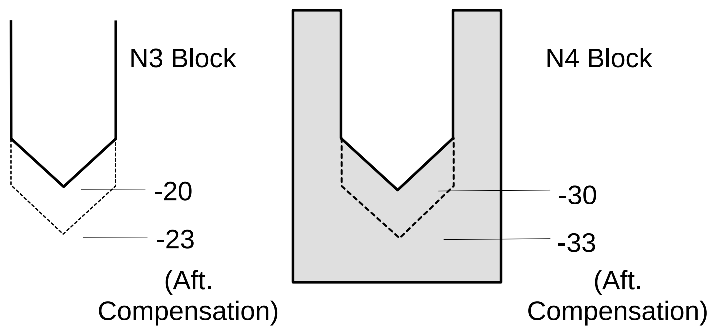


Fig 3-28

Ex 2: N1 G00 X-2.000 Y-2.000

N2 G44 Z-30.000 H1 (Length compensation 4,000)

N3 G01 Z-40.000

N4 G49 Z0.000

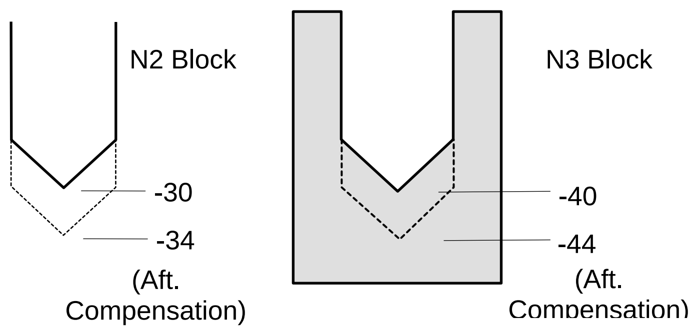


Fig 3-29

Ex 3: N0 G91

N1 G00 X120.000 Y80.000

N2 G43 Z-32.000 H01

N3 G01 Z-21.000 F100.

N4 G04 X2.000

N5 G00 Z21.000

N6 X30.000 Y-50.000

N7 G01 Z-41.000

N8 G00 Z41.000

N9 X50.000 Y30.000

N10 G01 Z-25.000

N11 G04 X2.000

N12 G00 Z57.000

N13 G49 X-200.000 Y-60.000

N14 M02

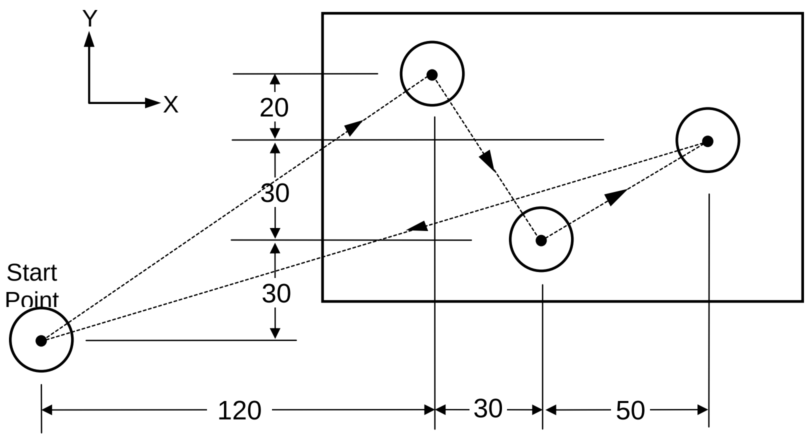


Fig 3-30

3.17 Work-Piece Size (Path) Enlarging and Shrinking, G50, G51

Format:

G51 I____J____K____ (Function Settings)

G50 (G51 cancelled)

I, J, K do not represent the coordinate but the proportions ranging from 0.01 to 100 for enlarging or shrinking the axial size. The ratio between the maximum and minimum value must be less than 100.

The G51 command is used to enlarge or shrink the profile and size on the path, it can operate 1, 2 or 3 axes at the same time. No specified axial direction is required, and the size is unchanged.

When using the G51 command, it's recommended to have the tool return to the start point after the cutting work is completed. Other actions should not be executed before using G50 to cancel the G51 command. The coordinates may change when G51 is used and the tool may hit the travel limit and cause an accident, particularly during the enlarging process. This command is a Modal G-code and should be cancelled using G50 after use.

Ex 1: N10 G1 X0.0 Y10.0 Z10.0

N2 G51 I0.500 . . . G51 is set and shrinks the X axial size by 50%

N3 X40.000 . . . Actual cutting X=20.000 mm with G51 executed

N4 G50

Ex 2: N10 G1 X0.0 Y10.0 Z10.0

N20 G51 I2.0 J2.0 . . . G51 is set and enlarges the X, Y axial size by 200%

N30 X40.0 Y40.0 . . . Actual cutting X=80.0 mm, Y=80.0 mm with G51 executed

N4 G50

3.18 Work Coordinate System Setting, G54~G59

There are two coordinate systems for CNC machine tools. This section describes how to use these coordinate systems.

1. Machine Coordinate System (Home)

2. Work Coordinate System

Work Coordinate System (G54~G59) --Set by To in MCM parameters (Recommended)

3.18.1 Machine Coordinate System (Home)

The origin of the machine coordinate system is fixed in the machine. When you press HOME from the control panel, the tool or machine table returns to the home limit switch, and detects the encoder GRID signal. When it locates the GRID, the tool stops. This location is the HOME position or Machine origin. The Machine origin is the calculation basis of all work and reference point coordinates. Its position is normally determined by the position of the travel-measuring rule on the machine table and the position of the over-travel limit switches (OTLS). Before any cutting, be sure to execute HOME to determine the position of the machine origin.

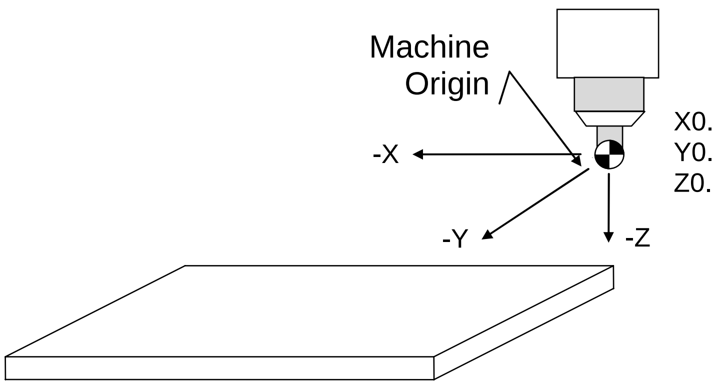


Fig 3-31 Machine Origin

Another origin may be required for cutting convenience. This origin is slightly shifted from the machine origin and, thus, is called HOME SHIFT. The shift amount is configured in MCM #183~186. When you execute HOME, the tool returns to the HOME position but the machine coordinate shows the shift value of MCM #183-186. If the shift value of MCM #183~186 is set to zero (0), the HOME SHIFT is the HOME position.

The methods to return to the HOME position are:

1. Manually return to the HOME position.

2. Use G28 or G30 to home the tools when the reference coordinates in MCM is set to zero for the X, Y, and Z axes.

3.18.2 Work Coordinate System Settings, G54~G59

H4CL-M series provides 6 sets of work origins. The coordinate system comprising these work origins is called Work Coordinate System. The 6 sets of work origins are located relative to the position of the machine origin. Their coordinates are called machine coordinates and stored in MCM #1~36. Coordinate data can be entered via:

1. G10 command in the MDI mode

2. Direct modification in MCM mode

3. Manual jog mode

The application of these work origins in the program is executed by the G54~G59 command codes. Depending on processing requirements and programming, the user can select up to six sets of work origins to work with. The most advantage of this work coordinate system is to simplify the coordinate operation of the part program. See the following examples:

G54 represents the work coordinate system using MCM #1~6.

G55 represents the work coordinate system using MCM #7~12.

G56 represents the work coordinate system using MCM #13~18.

G57 represents the work coordinate system using MCM #19~24.

G58 represents the work coordinate system using MCM #25~30.

G59 represents the work coordinate system using MCM #31~36.

Fig 3-32 shows the association of the G54~G59 work coordinate system with the 6 settings in the first item of MCM parameters. These coordinate parameters are determined depending on the machine origin. The G54-G59 work origin parameter settings are described as follows. (The XY drawing is used for illustration.)

G54 work coordinate system using MCM #1~6 with a setting of X-70,000, Y-10,000

G55 work coordinate system using MCM #7~12 with a setting of X-80,000, Y-30,000

G56 work coordinate system using MCM #13~18 with a setting of X-80,000, Y-50,000

G57 work coordinate system using MCM #19~24 with a setting of X-70,000, Y-50,000

G58 work coordinate system using MCM #25~30 with a setting of X-40,000, Y-60,000

G59 work coordinate system using MCM #31~36 with a setting of X-20,000, Y-40,000

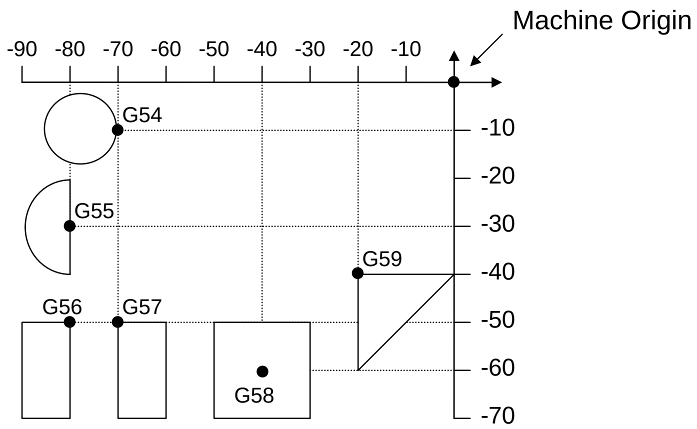


Fig 3-32 G54~G59 Work Coordinate System

Note that the program coordinates are changed when the work coordinate system is selected. The changed coordinates are determined based on the selected work coordinate system. When the action of cutting a circle or semi-circle is added to the above program, the application of G54 and G55 can be illustrated as follows. (Fig. 3-33)

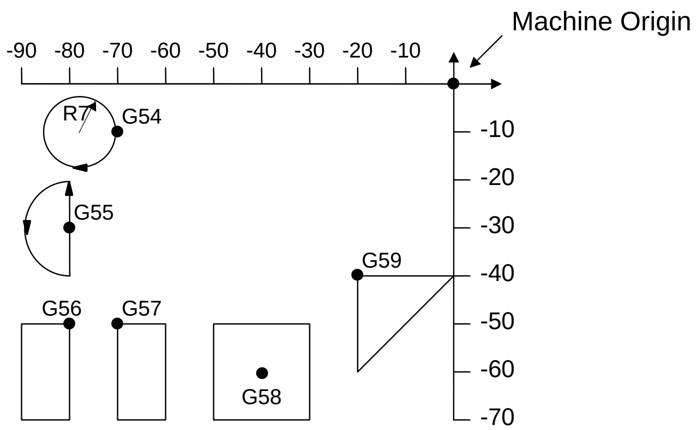


Fig. 3-33 G54~G59 Applications

Ex: Application of G54 and G55

N1 G0	...	Feed-rate set to fast move mode
N2 G54 X0. Y0.	...	Set to program coordinates X0, Y0 (machine coordinates X-70.,Y-10.)
N3 G2 I-7.0 F200.0	...	Cut a circle in CW with R=7.0
N4 G0	...	Feed-rate set to fast move mode
N5 G55 X0. Y0.	...	Move to coordinates X0, Y0 of the

		second work-piece (Machine coordinates X-80.,Y-30.)
N6 G1 V10.0 F300.	...	Y-axis feeding (incremental command) travels to +10.0
N7 G3 V-20.0 R10.0 F300.	...	Cut a semi-circle in CCW with R=10.0
N8 G1 V10.0 F300.	...	Y-axis feeding (incremental command) travels to +10.0
N9 G28	...	If the first reference point = 0, the program backs to the machine origin.
N10 M2	...	Program end

1. Power-on default is the G54 work coordinate system.
2. The work coordinate system is selected by executing G54~G59. If the X, Y, Z coordinates after executing the command are zero, the tool moves to the origin of the work coordinate system. If the X, Y, Z coordinates after executing the command are not zero, the tool moves the position corresponding to the X, Y, Z coordinates of the work coordinate system.
3. After executing G54~59, the machine coordinates of the work origin changes along with the setting of new coordinates.

3.19 Mirror-Effect Cutting, G68, G69

Format:

G68 -- X-axis mirror-effect cutting, with Y-axis as the mirror

G69 -- Y-axis mirror-effect cutting, with X-axis as the mirror

Mirror-effect cutting uses a subprogram (referring to the last section of this chapter) to design a cutting pattern, and then executes G68 and G69 to accomplish the mirror-effect cutting, as shown in Fig 3-34.

G 68 and G69 are used as a single program block in application. The sign of the X-coordinates behind the G68 block is inverted (+ changes to -, - changes to +) by executing G68 while the Y-coordinates are not affected. The sign of the Y-coordinates behind the G69 block is inverted by executing G69 while the X-coordinates are not affected. Therefore, all you need to do to cut the pattern of Fig 3-34 is to write a subprogram for pattern 1, then execute G68 and G69 for cutting patterns 2, 3, and 4. The program is written as follows:

M98 P___	-- Cut pattern 1 (P___ subprogram code)
G68	-- Invert the sign of X-coordinates behind G68 block

M98 P___	-- Cut pattern 2 (P___ subprogram code)
G69	-- Invert the sign of Y-coordinates behind G69 block
M98 P___	-- Cut pattern 3 (P___ subprogram code)
G68	-- Invert the sign of X-coordinates behind G68 block
M98 P___	-- Cut pattern 4 (P___ subprogram code)
G69	-- Invert the sign of Y-coordinates behind G69 block.
M02	

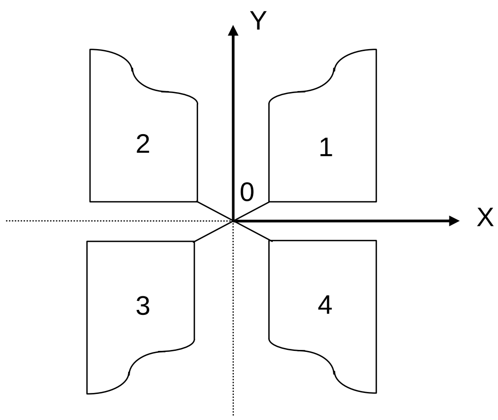


Fig 3-34 Mirror-effect Cutting

Note that G68 and G69 are modal G-codes.

Whenever G68 or G69 is executed: ＋X --> －X or ＋Y-->－Y

When G68 or G69 is executed next time: －X --> ＋X,或 －Y-->＋Y

Thus, if only patterns 1 and 2 need to be cut, G68 must be executed again to restore the sign of X-coordinates, as described below:

M98 P___	-- Cut pattern 1 (P___ subprogram code)
G68	-- Invert the sign of X-coordinates behind G68 block
M98 P___	-- Cut pattern 2 (P___ subprogram code)
G68	-- Invert the sign of X-coordinates behind G68 block
M02	

The RESET key can also cancel mirror-effect cutting. During mirror-effect cutting, the "X-MIRROR" or "Y- MIRROR " is shown at the top of the CRT screen. The display disappears when the mirror-effect cutting is canceled.

3.20 Absolute and Incremental Coordinate Settings, G90, G91

The absolute and incremental coordinate can be set via the following two approaches:

1. Mode -- Use G90 and G91 commends to specify a mode.
2. Incremental bit-code -- Use U.V.W commands to specify an incremental bit-code. (Refer to Chapter II)

Mode specification format:

G90 Absolute coordinates setting

G91 Incremental coordinates setting

The absolute coordinates system is the default power-on of the H4CL-M Series. Use G90 and G91 to set the absolute or incremental coordinates in the program. The incremental bit-code U,V,W are only valid in the G90 status. They are invalid in the G91 status. X, Y, Z stand for incremental coordinates in the G91 status.

Ex 1: Setting absolute coordinates (Fig. 3-35)

N1 G90

N2 G1 X20.000 Y15.000 ....P0 to P1

N3 X35.000 Y25.000 ....P1 to P2

N4 X60.000 Y30.000 ....P2 to P3

Ex 2: Setting incremental coordinates (Fig 3-35)

N1 G91

N2 G1 X20.000 Y15.000 ....P0 to P1

N3 X15.000 Y10.000 ....P1 to P2

N4 X25.000 Y5.000 ....P2 to P3

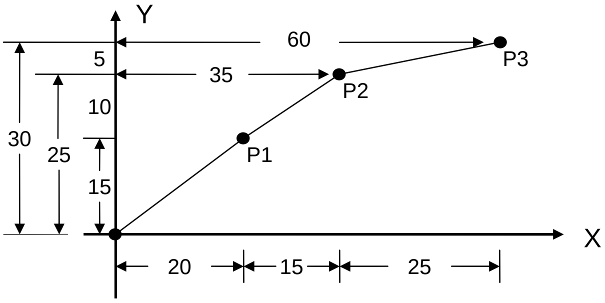


Fig 3-35 G90, G91 Example

3.21 Canned Cycle Functions (H4CL-M only), G81~G89, G80

These G-code commands are for the H4CL-M milling machine only, and NOT for other HUST CNC controllers.

H4CL-M provides a number of canned cycle cutting functions for processing. They form a command group and are executed using a specified G-code. The H4CL-M series provides several canned cycle functions to simplify program design. The cutting sequence controlled by the canned cycle command group of H4CL-M is illustrated in Fig. 3-36 below.

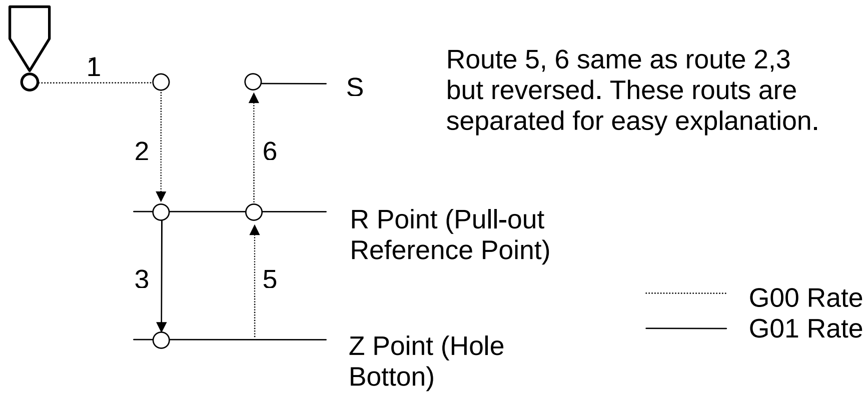


Fig 3-36 Canned Cycle Cutting Sequence

1. Fast positioning to the start (S) point on X-Y plane.
2. Fast positioning to the reference point for drilling start (R) along Z-axis.
3. Hole drilling to at the bottom (Z) along the Z-axis.
4. Mechanical action at the hole bottom – tool waits or spindle rotation reverses.
5. Drill bit retracted to R-point. The moving speed depends on the command specified.
6. Move back to the start point S at G00 feed-rate.

When applying the canned cycle function, M03 is used for normal spindle rotation, M04 is used for reverse spindle rotation, and M05 is used for spindle stop.

Basic Format For Canned Cycle Functions:

G90 or G91

G98

G81~G89 X____Y____Z____P____Q____R____F____K____

G80 or G00 or G01

Explanation

X, Y	:	Specify absolute or incremental coordinates for the hole.
Z	:	Specify absolute or incremental depth or coordinates for the hole.
P	:	Dwell at the hole bottom. Unit: ms; i.e. 1000 stands for one second.
Q	:	G83 amount of feed for each cut, in μm.
R	:	Specify the absolute or incremental coordinates of R-point. R is the reference point of feeding/retraction.
F	:	Feed-rate setting.
K	:	Processing repetition setting.

During the drilling operation, parameters such as specified drilling mode (such as G81, G82, and so on), feeding/retraction reference point, and hole depth or coordinates (Z) are mode codes. They will not change before other command codes of the same group are set up. The single block command of each basic format with respect to the canned cycle function is described in detailed below.

3.22 G90 or G91-Absolute or Incremental Coordinates Setting

These commands were described in the previous section. In the canned cycle program, R and Z are coordinates relative to the zero point of Z-axis in the absolute coordinates system, while Z is an incremental coordinate relative to R-point in the incremental coordinates system. Though the R and Z coordinates remain unchanged in the program, their displacement coordinates are different (Fig. 3-37) when executing different commands (G90/G91).

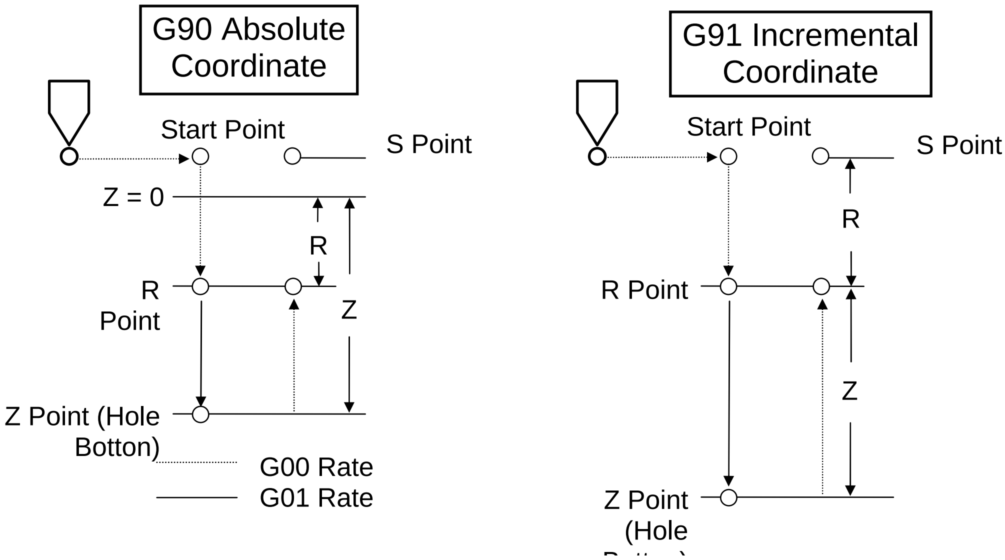


Fig 3-23 G90 and G91 Application

3.23 G98 or G99 – Cutting Feed-rate Setting

G98 : Feed-rate per minute, mm/min

G99 : Feed-rate per revolution, mm/rev

The feed rate (F) of the H4CL-M Series is defined by G98 and G99. G98 is the power-on default setting. The conversion equation of both commands is:

Fm = Fr * S

Fm : Feed-rate per minute, mm/min

Fr : Feed-rate per revolution, mm/rev

S : Spindle rpm, rev/min

3.24 G80, G81~G89 -- Canned Cycle Commands

Definition of each parameter for canned cycle commands has been described above. The work-piece cutting application of G80~G89 is tabulated in Table 3-38.

G-code	Application	Drill Rate	Action at Bottom	Retraction Rate
G80	Cycle canceled	---	-----	---
G81	Drilling Canned Cycle	G01 Feed rate	---	G00 Fast

G82	Drilling Canned Cycle	G01 Feed rate	Dwell	G00 Fast
G83	Deep Hole Drilling (peck drill)	G01 Feed rate	---	G00 Fast
G85	Boring Canned Cycle	G01 Feed rate	---	G01 Rate
G86	Boring Canned Cycle	G01 Feed rate	Spindle stop	G01 Fast
G89	Boring Canned Cycle	G01 Feed rate	Dwell	G01 Rate

Table 3-38 G80~G89 Work-piece Cutting Application

3.25 G80 Cancellation of Canned Cycle

All canned cycle commands are cancelled by executing G80, G00 or G01.

3.26 G81 Drilling Canned Cycle

Format:

G81 X____Y____Z____R____K____F____

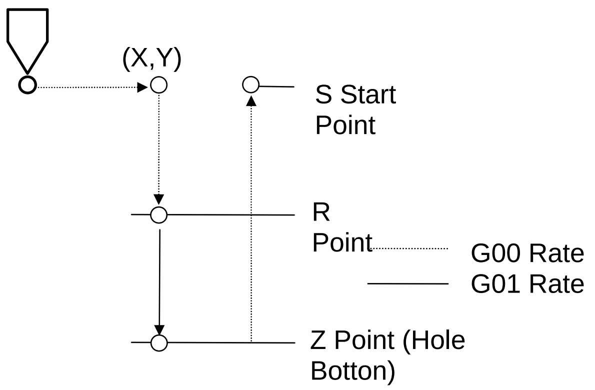


Fig 3-39 G81 Drilling Canned Cycle

3.27 G82 Drilling Canned Cycle

Format:

G82 X____Y____Z____P____R____K____F____

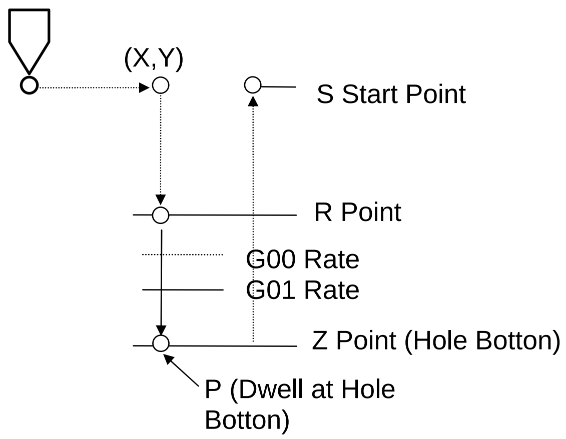


Fig 3-41 G82 Drilling Canned Cycle

The difference between G81 and G82 is that G82 has a wait time (P) before retraction when the drill bit reaches the bottom. The wait time (P) is input as an integer in milliseconds.

3.28 G83 Deep Drilling Canned (peck drill) Cycle

Format:

G83 X____Y____Z____R____Q____K____F____

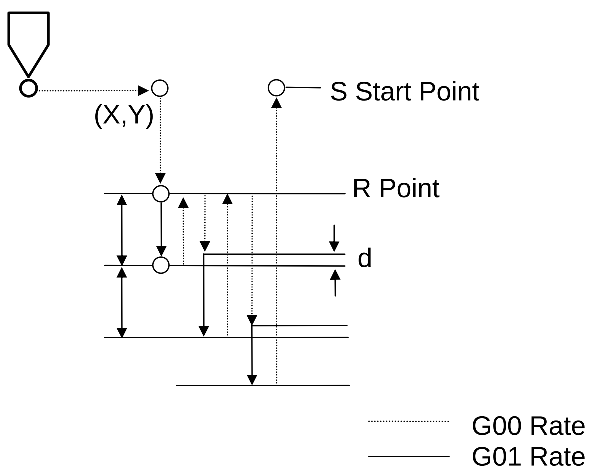


Fig 3-42 G83 Deep Drilling Canned (peck drill) Cycle

In Fig. 3-42, Q is the depth of each drilling and d is the reservation for the change of the feed-rate from G00 to G01 after the second feeding. This data is set in MCM parameter #282. (The d value can be changed in the graphics input form.)

3.29 G84 Tap Cutting Canned Cycle

G84 X(U)___,Y(V)___,Z(W)___,R___,Q___,F___

X, Y Specify absolute coordinates for the hole.

U, V Specify incremental coordinates for the hole.

Z Specify absolute depth for the hole bottom.

Specify incremental depth for the hole bottom.

R Specify start point of feeding/retraction.

Q Specify each feed amount. **(If Q is not specified, the tool feeds directly to the end without stopping.)**

F Specify the feeding amount per revolution.

MCM 281<>0, MCM273 setting as the taping retraction amount.

MCM 281=0, Return to R whenever a feeding action is completed.

The spindle positioning function is required for tap cutting. So I11 is the home of the spindle.

☆ **Before the tap cutting, make sure the spindle setting, parameter, and feedback are correct~**

☆ **If no spindle encoder feedback is available, don't use this function~**

Parameters for 4 axes + spindle (with an inverter or servo motor): (spindle connected to the D/A port) & Parameters for 3 axes + inverter spindle: (spindle connected to the D/A axis)

☐	Spindle speed	MCM 225
☐	Spindle encoder pulses	MCM 224
☐	Spindle speed for tap cutting	R132

Parameters for 3 axes + 1 servo spindle: (spindle connected to A axis)

☐	A axis resolution	MCM124、125
☐	Spindle speed	MCM 225

- ☐ Spindle encoder pulses MCM 224
- ☐ Spindle speed for tap cutting R132

3.30 G85 Boring Canned Cycle

Format:

G85 X____Y____Z____R____K____F____

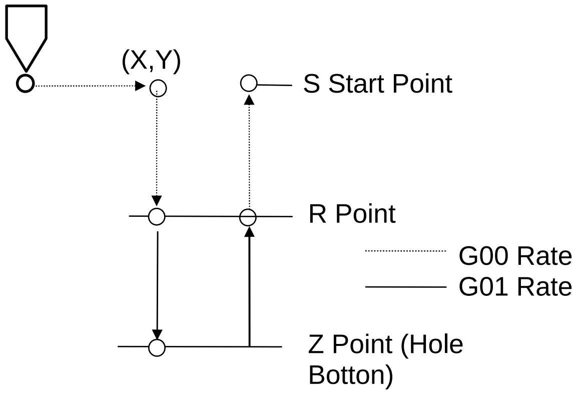


Fig 3-43 G85 Boring Canned Cycle

3.31 G86 Boring Canned Cycle (Spindle Stop at Hole Bottom)

G86 X____Y____Z____R____K____F____

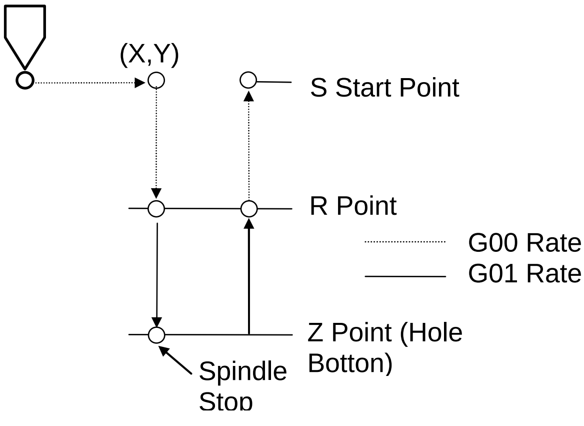


Fig 3-44 G86 Boring Canned Cycle

The difference between G85 and G86 is that the G86 spindle stops before retraction when the drill bit reaches the hole bottom.

3.32 G89 Boring Canned Cycle with Dwell at Hole Bottom

Format:

G89 X____Y____Z____R____P____K____F____

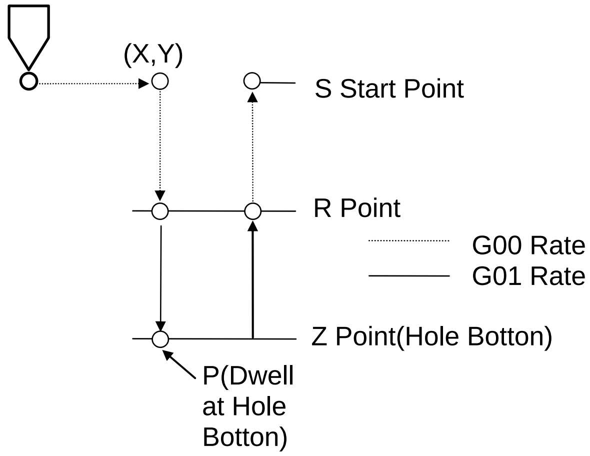


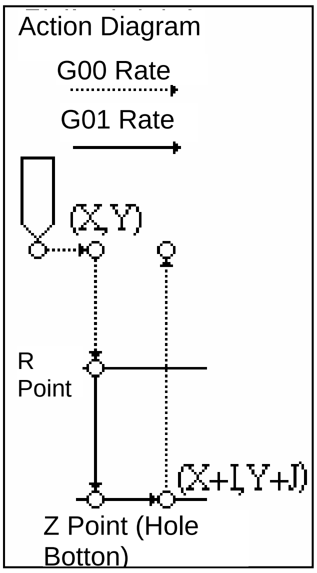
Fig 3-45 G89 Boring Canned Cycle

The difference between G85 and G89 is that G89 has a wait time (P) before retraction when the drill bit reaches bottom. The wait time (P) is input as an integer in milliseconds.

3.33 G22 Linear Groove Milling (Only available in absolute mode)

Format:

G22 X___ Y___ Z____ R___ I____ J____ F____



- X Start point of coordinate X
- Y Start point of coordinate Y
- Z Grooving depth
- R Height of outer part
- I The X-axis incremental coordinate with an end point relative to the start point.
- J The Y-axis incremental coordinate with an end point relative to the start point.
- F Grooving speed

Explanation:

1. G00 X(x) Y(y)
2. G00 Z(r)
3. G01 Z(z) F(f)
4. G01 X(x+I) Y(y+j)
5. G00 Z(r)

3.34 G23 Arc Groove Milling (Only available in absolute mode)

Format:

G23 X___Y___Z___R___I___J____K___T___F____

X Start point coordinate

Y Start point coordinate

Z Groove depth

R Height of outer part

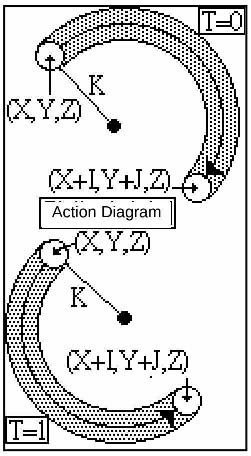
I X-axis incremental coordinates with an end point relative to the start point.

J Y-axis incremental coordinates with an end point relative to the start point.

K Radius of circle

T Grooving type (0~1)

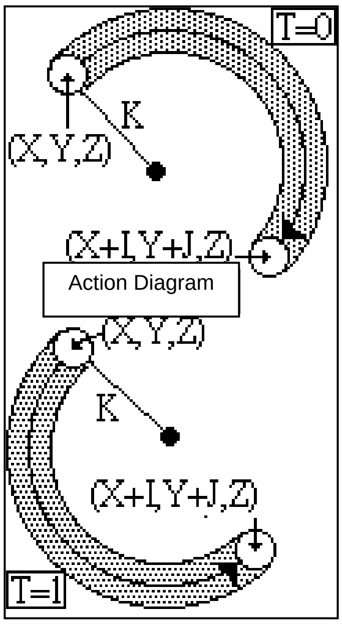
F Grooving speed



Explanation:

1. G00 X(x) Y(y)
2. G00 Z(r)
3. G01 Z(z) F(f)
4. T=0; G02 U(i) V(j) R(k) F(f)

   T=1; G03 U(i) V(j) R(k) F(f)
5. G00 Z(r)



The gray area shows the cutting trajectory.

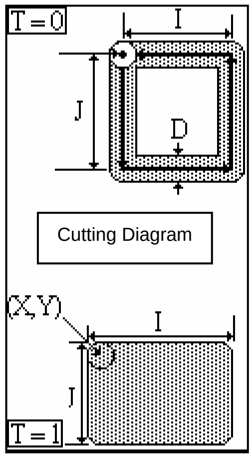
PS. R-value is positive when an arc less than 180-degrees is cut.

R-value is negative when an arc greater than 180-degrees is cut.

3.35 G24 Square Groove Milling (Only available in absolute mode)

Format:

G24 X___Y___Z___R___I___J____D___T___F____



X Start point coordinate

Y Start point coordinate

Z Groove depth

R Height of outer part

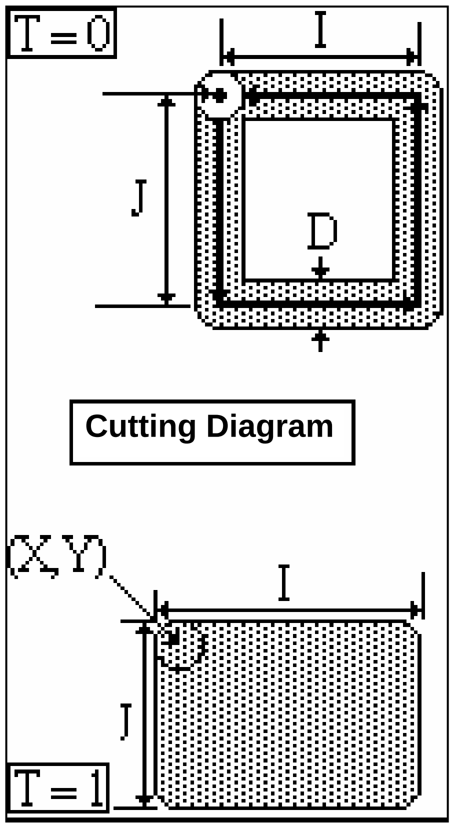
I Groove width

J Groove length

D Tool radius

T Groove type (0~1)

F Groove Speed

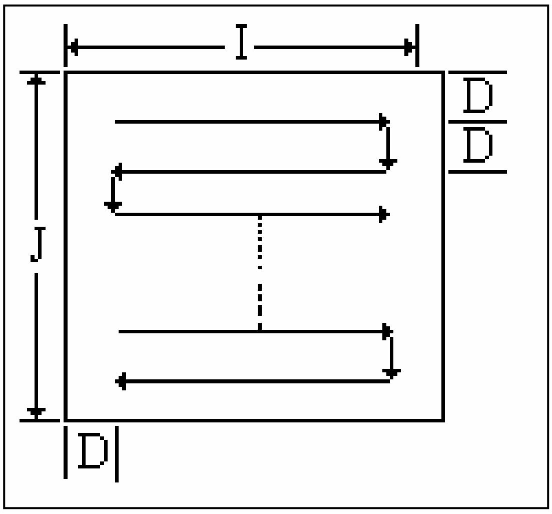


Explanation:

T=0;

1. G00 X(x) Y(y)
2. G00 Z(r)
3. G01 Z(z)
4. G01 U(i)
5. G01 V( )
6. G01-U(i)
7. G01-V(j)
8. G00 Z(r)

T=1;

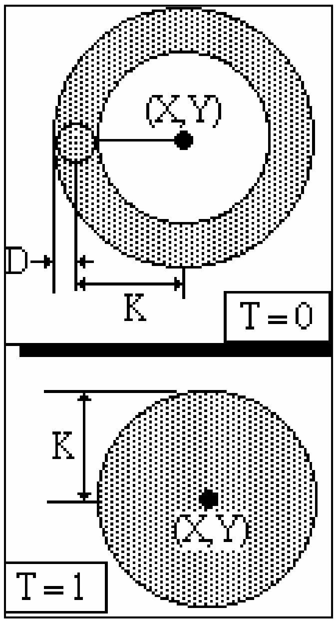


As shown in the above figure, an inner square is cut in a S-shaped groove-milling manner. Then cut again along the side to remove the part that is not cut during the S-shaped groove milling process.

3.36 G25 Round Groove Milling (Only available in absolute mode)

Format:

G24 X___Y___Z___R___K___D___T___F____



X	Center coordinate
Y	Center coordinate
Z	Groove depth
R	Height of outer part
K	Radius of circle
D	Tool diameter
T	Groove type
F	Groove speed

Explanation:

T=0;

1. G00 X(x-k) Y(y)
2. G00 Z(r)

3. G01 Z(z) F(f)

4. G02 I(k) J(0) R(k) F(f)

5. G00 Z(r)

<u>T=1;</u>

1. G00 X(x-k) Y(y)

2. G00 Z(r)

3. G01 U(d) F(f)

4. G01 Z(z) F(f)

5. G02 I(k) J(0) R(k) F(f)

6. IF (k>d) THEN {[k=k-d]and[goto N3]}

7. G00 Z(r)

3.37 Special Canned Cycle

The special canned cycle should go with the canned cycle commands G81~G89.

Before using the special canned cycle, the commands should be used to specify the hole processing data.

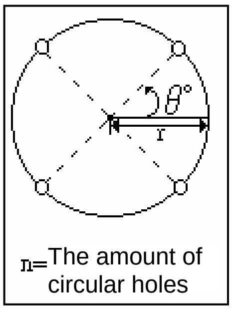
If no hole processing data is specified by executing the canned cycle commands, the special canned cycle command only provides positioning functions without drilling.

Tools traveling between holes at the highest speed (G00).

3.38 G34 Circular Drilling Canned Cycle

Format:

G34 X___Y___ I___J____K___F____



X Center coordinates

Y Center coordinates

I Radius of circle r

J Angle of the first hole –θ

K The amount of circular holes – n

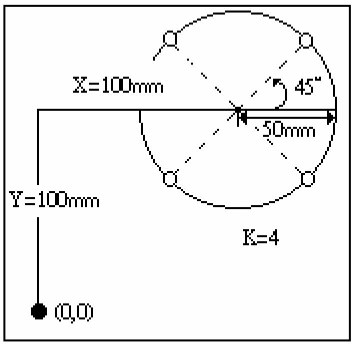
F Drilling speed

Example:

N001 G81 Z-100. R-50.K0 F100 ←Definition of drilling resource

N002 G34 X100. Y100. I50. J45. K4

N003 G80 ←Canned cycle command canceled



3.39 G35 Angular Linear Drilling Canned Cycle

Format:

G35 X___Y___ I___J____K___F____

X Start point coordinate

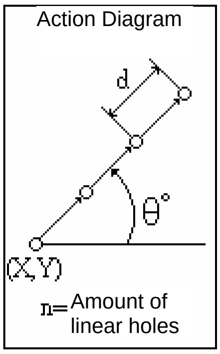
Y Start point coordinate

I Drilling distance d

J Angle – θ

K The amount of linear holes – n

F Drilling speed

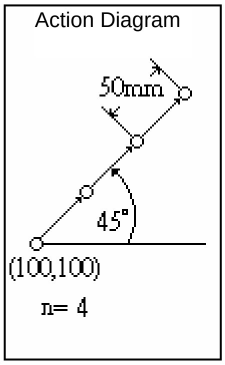


Example:

N001 G81 Z-100. R-50.K0 F100 ←Definition of drilling data

N002 G35 X100. Y100. I50. J45. K4

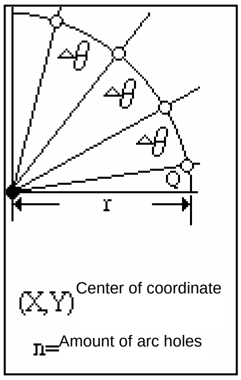
N003 G80   Canned cycle command canceled



3.40 G36 Arc Drilling Canned Cycle

Format:

G36 X___Y___ I___J____P____K___F____



X Center coordinate

Y Center coordinate

I Circle radius

J Angle of the first hole – θ

P Angle of each drilling

K The number of arc holes

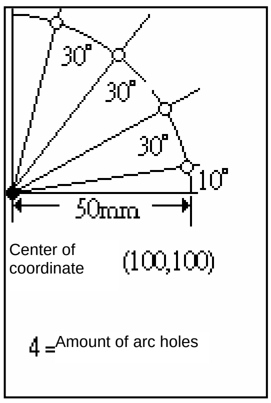
F F Drilling speed

Example:

N001 G81 Z-100. R-50.K0 F100 ←Definition of drilling data

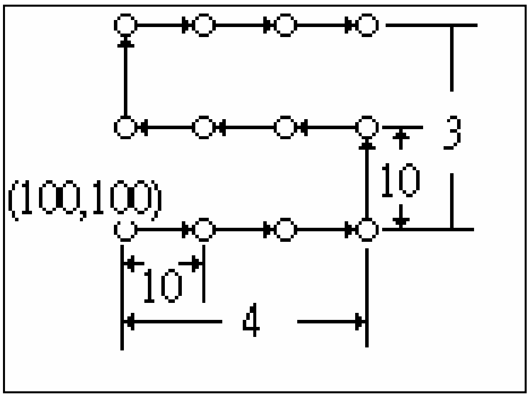
N002 G36 X100. Y100. I50. J10.P30. K4

N003 G80 ←Canned cycle command canceled



3.41 G37 Grid Drilling Canned Cycle

G37 X___Y___ I___P___J____K___F____



X Start point coordinate

Y Start point coordinate

I X-axis space

P Drilling Numbers of X-axis

J Y-axis space

K Drilling Numbers of Y-axis

F Number of grid holes

Example:

N001 G81 Z-100. R-50.K0 F100 ←Definition of drilling data

N002 G37 X100. Y100. I10.P4 J10. K3

N003 G80 ←Canned cycle command canceled

3.42 Auxiliary Function, M-code, S-code

The auxiliary function M-code (referred to as M-code) consists of a capital letter M followed by a 2-digit number. The M-code ranges 00~99 and each code represents a different action. The following M-codes are used by H4CL-M system and no customers are allowed access.

M00 Program Stop.

When the program runs to this point, all processing actions stop, including spindle and coolant. Press the "CYCST" key to restart the program from where it stopped.

M01 Optional Stop.

See more details in Sec.6 of Chap 8.

M02 Program End.

M30 Program Finished.

The program finishes at this point and returns to the start point.

M98 Subprogram Call.

M99 Subprogram End.

Except for the above M-codes that cannot be changed, customers may define other M-codes in the PLC if required. Examples of some general settings are shown below. These examples are also parts of the H4CL-M standard PLC Ladder.

M03 Spindle rotation in normal direction.

M04 Spindle rotation in reversed direction.

M05 Spindle rotation stops.

M08 Coolant ON.

M09 Coolant OFF.

The auxiliary function S-code is used to control the rpm of the spindle. The maximum setting is S999999.

Ex: S1000 means that the spindle rotates at 1000 rev/min

3.43 Subprogram

If there are some program or command groups requiring repeated execution, you can store these program or command groups in memory as a subprogram. This can simplify the design of the program and make the structure of the main program more succinct. The subprogram can be executed during automatic operation and a subprogram can call another subprogram.

3.43.1 Subprogram Structure

The structure of the subprogram is pretty much the same as the main program except that the subprogram ends with M99.

PROGRAM 05	. . . . .	Subprogram code
	. . . . .	Program content
	. . . . .	Program content
M99	. . . . .	Program end

If the subprogram is not called by the main program but executed by directly pressing "CYCST", it stops after executing 8,000,000 times.

3.43.2 Execution of Subprogram

Format:

M98 P____L____

P : Subprogram code

L : Execution times of the subprogram. If not specified, the subprogram executes only once.

Ex: M98 P05 . . . . . . Execute subprogram No 5 once.

M98 P05 L3 . . . . . . Execute subprogram No 5 three times.

Stepwise Call: The main program calls the first subprogram, and the first subprogram calls the second subprogram in turn. The H4CL-M Series controller provides a maximum of 5 levels stepwise call, as shown in the following figure:

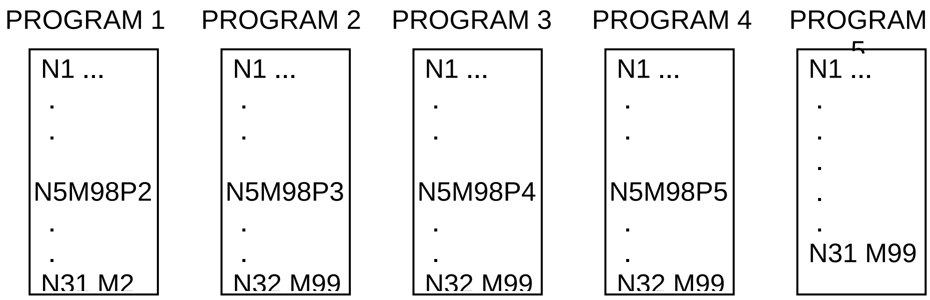


Fig 3-46 Subprogram Stepwise Call

The M98 and M99 block settings shall not contain any displacement, such as X…, Z… .

3.44 Customized Program Group [MACRO] Command, G65

G65 is used for basic and logic operations for some variables and executes the program branching function after analyzing a specified variable. G65 is available to the main program and subprogram. A group of G65 can be formed as an independent program group with the same structure as the subprogram. For definition of each operator with respect to a program group command, refer to Table 3-4.

G65 Format:

G65 Lm P#i A#j B#k

L, P, A, B : G65 codes are unchangeable.

m : Operation code as defined in Table 3-3.

Ex.: L2 stands for addition (+) and L3 stands for subtraction (-).

#i : Functions.

1. P#i is the location to store the result of mathematical operations.
2. Pi is the program serial number for line feed when a function is deemed as valid.

#j : Variable name 1. This function represents a variable number a constant.

CASE1: A#j : j represents a variable number in the range 1 -- 9999.

CASE2: Aj :j represents a constant in the range -9999999 -- 9999999.

Please note that the format "Aj" has no "#".

#k : Variable name 2. This function represents a variable number or a constant.

CASE1: A#j, j represents a variable number ranging 1 -- 9999.

CASE2: Aj, j represents a constant ranging from “-“ 9999999 to “--” 9999999.

Please note that the format "Aj" has no "#".

Variable Explanations:

1. Variable #i

   #1~#9999 : User defined variables,

   The can be stored when the power is turned off.

   #10000 and above: These are read-only controller system variables. No changes are allowed.

2. All variables (#i, #j, #k) can only contain integers. No decimal should be used. #i must be positive, while #j and #k can be positive or negative. A negative integer indicates that the sign of the value contained in that variable is inverted.

Ex. 1: #2 = 99

G65 L01 P#1 A-#2 ; #1 = -#2 = -99。

Ex. 2: #2 = 25, #3 = 5

G65 L04 P#1 A#2 B-#3 ; #1 = #2× -#3 = -125。

3. If the content value of #j and #k is entered as a constant, it must be an integer (max 7 digits, + or -). The input unit is depending on the decimal format of the G65 command. Refer to Sec. 6.5 for details.

Decimal Point	1 (6/1 format)	2 (5/2 format)	3 (4/3 format)	4 (3/4 format)
Unit	100 μm	10 μm	1 μm	0.1 μm
Ex.: 250 entered	25000 μm	2500 μm	250 μm	25 μm

Table 3-4 Mathematical Operator Definitions For HUST G65 Command

G-code	L-code	Operator Definition	Mathematical Definitions
G65	L01	Equal or Substitution,	#i = #j
G65	L02	Addition	#i = #j + #k
G65	L03	Subtraction	#i = #j - #k
G65	L04	Multiplication	#i = #j x #k
G65	L05	Division	#i = #j / #k
G65	L06	Place Data into Variables	#i = #j
G65	L07	Copy Variables	
G65	L11	Logic OR,	#i = #j .OR. #k
G65	L12	Logic AND,	#i = #j .AND. #k
G65	L13	Logic XOR,	#i = #j .XOR. #k
G65	L14	ROL, rotate left	
G65	L15	ROR, rotate right	
G65	L16	LSL, move left	
G65	L17	LSR, move right	
G65	L21	Subduplicate	#i = √#j
G65	L22	Absolute	#i = |#j|
G65	L23	Complement	#i = #J - trunc(#j/#k) x #k trunc:(Disregard values less than 1)
G65	L26	Combined Mul/Div Operation	#i = (#i x #j) / #k
G65	L31	Sin	#i = #j x Sin(#k)
G65	L32	Cos	#i = #j x Cos(#k)
G65	L33	Tangent (Tan)	

G65	L34	Arctangent (Tan^{-1})	
G65	L50	Obtain Data in Register	#i = #j
G65	L51	Obtain I-Bit data	#i = #j
G65	L52	Obtain O-Bit data	#i = #j
G65	L53	Obtain C-Bit data	#i = #j
G65	L54	Obtain S-Bit data	#i = #j
G65	L55	Obtain A-Bit data	#i = #j
G65	L56	Obtain Counter Data	#i = #j
G65	L60	Register Setting	#i = #j
G65	L66	Counter Setting	#i = #j
G65	L80	Unconditional Branching	Go To n; program goes to block number 'n'
G65	L81	Conditional Branching 1	If #j = #k, Go To n
G65	L82	Conditional Branching 2	If #j #k, Go To n
G65	L83	Conditional Branching 3	If #j > #k, Go To n
G65	L84	Conditional Branching 4	If #j < #k, Go To n
G65	L85	Conditional Branching 5	If #j #k, Go To n
G65	L86	Conditional Branching 6	If #j #k, Go To n
G65	L99	User Defined Error Signal	Error signals display = i+50 (i=1~49)

Note: The range of computation is from (–9999.999) to (+9999.999).

<u>Mathematical Operation Examples</u> (See Table 3-4)

1. Equal or Substitution

G65 L1 P#i A#j ; #i = #j

Ex. 1: #10 initial value=0, for #10 = 150

Command	: G65 L1 P#10 A150
Result	: #10 = 150

Ex. 2: #10 initial value=0, #5 initial value=1200, for #10 = #5

Command	: G65 L1 P#10 A#5
Result	: #10 = 1200

Ex. 3: #10 initial value=0, #5 initial value=1200, for #10 = -#5

Command	: G65 L1 P#10 A-#5
Result	: #10 = -1200

2. Addition

G65 L2 P#i A#j B#k ; #i = #j + #k

Ex. 1: #10 initial value=99, #5 initial value=1200, for #1 = #10 + #5

Command	: G65 L2 P#1 A#10 B#5
Result	: #1 = #10 + #5 = 1299

Ex. 2: #10 initial value=99, for #10 = #10 + 1

Command	: G65 L2 P#10 A#10 B1
Result	: #10 = #10 + 1 = 100

3. Subtraction

G65 L3 P#i A#j B#k ; #i = #j - #k

Ex. 1: #10 initial value=1200, #5 initial value=99, for #1 = #10 - #5

Command	: G65 L3 P#1 A#10 B#5
Result	: #1 = #10 - #5 = 1101

Ex. 2: #10 initial value=99, for #10 = #10 - 1

Command	: G65 L2 P#10 A#10 B1
Result	: #10 = #10 - 1 = 98

4. Multiply

G65 L4 P#i A#j B#k ; #i = #j × #k

The result of multiplication should be in the range of -9999.999~+9999.999. Otherwise, the system operation results in error.

Ex. 1: #4 initial value=10, #30 initial value=25, for #10= #4 × #30

Command : G65 L4 P#10 A#4 B#30

Result : #10 = #4 × #30 = 250

Ex. 2: #4 initial value=100000, #30 initial value=25000,
for #10 = #4 × #30

Command : G65 L4 P#10 A#4 B#30

Result : #10 = ?????

(HUST H4CL-M controller cannot handle the multiplied values greater than 9999.999.)

5. Division

G65 L5 P#i A#j B#k ; #i = #j / #k

Results less than 1 are disregarded.

Ex. 1: #4 initial value=130, #30 initial value=25, for #10= #4 / #30

Command : G65 L5 P#10 A#4 B#30

Result : #10 = #4 / #30 = 5 (130/25 = 5.2)

Ex. 2: #4 initial value=10, for #10 = #4 / 30

Command : G65 L5 P#10 A#4 B30

Result : #10 = #4 / 30 = 0

6. Place Data into Variables

G65 L6 P# i A#j B#k ; # i …. #(i+k) = # j

Ex. 1: initial value #10=100, #11=20, #13=50, #5=99
for #10 = #11 = #12 = #13 = #14 = #5

Command : G65 L6 P#10 A#5 B5

Result : #10 = #11 = #12 = #13 = #14 = #5 = 99

Ex. 2: For #10 …..#(10+N-1) = 100, N = #3 = 4

Command : G65 L6 P#10 A100 B#3

Result : #10 = #11 = #12 = #13 = 100

7. Copying Variables

G65 L7 P#i A#j B#k ; #i = #j; #(i+1)=#(j+1) ….

If #i plus 900000 ; #(#i) =#j; #(#i)+1=#(j+1)

Note: 0 < #k < 1024

Ex. 1: Copy #10 ….. #20 to #125…. #135

Command	: G65 L7 P#125 A#10 B11
Result	: #125=#10, #126=#11, #127=#12, #128=#13 #129=#14, #130=#15, #131=#16, #132=#17 #133=#18, #134=#19, #135=#20

Ex. 2: Copy #1 ….. #5 to #256…. #260

Initial value	: #256 = 101, #1 = 301
Command	: G65 L7 P#256 A#1 B5
Result	: #256 = #1 = 301, #257 = #2, #258 =#3, #259 = #4, #260 = #5

Ex. 3: Copy #1 ….. #5 to #101…. #105

Initial value	: #256 = 101, #101 = 121、#1 = 301
Command	: G65 L7 P#900256 A#1 B80
Result	: #101 = #1 = 301, #102 = #2, #103 =#3, #104 = #4, #105 = #5

8. Logic OR

G65 L11 P#i A#j B#k ; #i = #j .OR. #k

Ex. 1: For #10 = #5 .OR. #20, #5 = 12, #20=100

Command	: G65 L11 P#10 A#5 B#20
Result	: #10 = 12 .OR. 100 = 108

Ex. 2: For #10 = #10 .OR. 10, #10 = 15

Command	: G65 L11 P#10 A#10 B10
Result	: #10 = 15 .OR. 10 = 15

9. Logic AND

G65 L12 P#i A#j B#k ; #i = #j .AND. #k

Ex. 1: For #10 = #5 .AND. #20, #5 = 12, #20=100

Command	: G65 L12 P#10 A#5 B#20

Result : #10 = 12 .AND. 100 = 4

Ex. 2: For #10 = #10 .AND. 10, #10 = 15

Command : G65 L12 P#10 A#10 B10

Result : #10 = 15 .AND. 10 = 10

10. Logic XOR

G65 L13 P#i A#j B#k ; #i = #j .XOR. #k

Ex. 1: For #10 = #5 .XOR. #20, #5 = 4, #20=100

Command : G65 L13 P#10 A#5 B#20

Result : #10 = 4 .XOR. 100 = 96

Ex. 2: For #10 = #10 .XOR. 10, #10 = 15

Command : G65 L11 P#10 A#10 B10

Result : #10 = 15 .XOR. 10 = 5

11. ROL (Rotate Left)

G65 L14 P#i A#j B#k

In a 16-bit (Bit15 to Bit0) rotation, Bit15 shifts to Bit0 when rotating to the left. Where calculation exceeds 16 bits, the bits after Bit15 are disregarded.

Bit	15	14	…	…										2	1	0
	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x

Ex. 1: Initial value #10 = 49152

Command : G65 L14 P#12 A#10 B1 (ROL once)

Result : #12 = 32769

Bit	15	14	…	…										2	1	0
	1	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0

Bit	15	14	…	…										2	1	0
	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1

1

Ex. 2: Initial value #10 = 7

Command : G65 L14 P#12 A#10 B1 (ROL once)

Result : #12 = 14

Bit	15	14	…	…										2	1	0
	0	0	0	0	0	0	0	0	0	0	0	0	0	1	1	1

⇩

Bit	15	14	…	…										2	1	0
	0	0	0	0	0	0	0	0	0	0	0	0	1	1	1	0

0

Ex. 3: Initial value #10 = -2

Command : G65 L14 P#12 A#10 B1 (ROL once)

Result : #12 = -3

Bit	15	14	…	…										2	1	0
	1	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0

⇩

Bit	15	14	…	…										2	1	0
	1	1	1	1	1	1	1	1	1	1	1	1	1	1	0	1

1

12. ROR (Rotate Right)

G65 L15 P# i A#j B#k

In a 16-bit (Bit15 to Bit0) rotation, Bit0 shifts to Bit15 during right rotation. Where calculation exceeds 16 bits, bits after Bit15 are disregarded.

Bit	15	14	…	…										2	1	0
	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x

Ex. 1: Initial value #10 = 3

Command : G65 L15 P#12 A#10 B1 (ROR once)

Result : #12 = 32769

Bit	15	14	…	…										2	1	0
	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	1

⇩

Bit	15	14	…	…										2	1	0
	1	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1

1

Ex. 2: Initial value #10 = 6

Command : G65 L15 P#12 A#10 B1 (ROR once)

Result : #12 =3

Bit	15	14	…	…										2	1	0
	0	0	0	0	0	0	0	0	0	0	0	0	0	1	1	0

⇩

Bit	15	14	…	…										2	1	0
	0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	1

0

13. LSL (Move Left)

G65 L16 P#i A#j B#k

Bit	15	14	…	…										2	1	0
	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x

Ex. 1: Initial value #10 = 13

Command : G65 L16 P#12 A#10 B2 (LSL twice)

Result : #12 = 52

Bit 15 14 … … 2 1 0

0	0	0	0	0	0	0	0	0	0	0	0	1	1	0	1

Bit 15 14 … … 2 1 0

0	0	0	0	0	0	0	0	0	0	1	1	0	1	0	0

14. LSR (Move Right)

G65 L17 P#i A#j B#k

Bit 15 14 … … 2 1 0

x	x	x	x	x	x	x	x	x	x	x	x	x	x	x	x

Ex. 1: Initial value #10 = 13

Command : G65 L17 P#12 A#10 B2 (LSR twice)

Result : #12 = 3

Bit 15 14 … … 2 1 0

0	0	0	0	0	0	0	0	0	0	0	0	1	1	0	1

Bit 15 14 … … 2 1 0

0	0	0	0	0	0	0	0	0	0	0	0	0	0	1	1

15. Subduplicate

G65 L21 P#i A#j ; $\#i = \sqrt{\#j}$

Results less than 1 are disregarded.

Ex. 1: For $\#10 = \sqrt{\#5}$

4 MCM Parameter Settings

4.1 MCM Parameter Settings

The MCM parameter setting function allows the user to define the system constants of the controller according to mechanical specifications and machining conditions. The correct and proper setting of these constants is important in the operation of the mechanical system and fabrication of the work-piece. Make sure that the setting is correct. Press Reset to restart the machine when the MCM parameter is set successfully.

The MCM Parameter Setting List is shown on the following page. You can configure MCM parameters with reference to this list.

How to Read and Change MCM parameters:

(1) Change with the “SPACE” button

1. Press RESET and double-click I/O Parameter to enter MCM mode. The first MCM page (i.e. MCM #1~10) appears on the LCD screen as shown in Fig. 4-1.

2. Use Page↑/Page↓ to switch to the next or previous page. 10 parameters are displayed on each page.

3. Move the cursor to the parameter to be changed using the Cursor↑ or Cursor↓ key, enter a new value, and press the Enter key to finish the change.

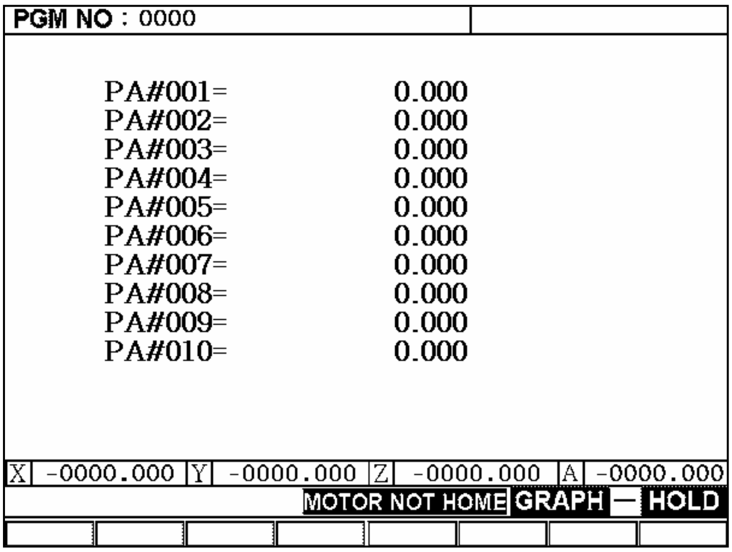


Fig 4-1

(2) Change via Upload from RS232C:
Use the transmission software (HCON) to send parameters to the PC for saving as a text file. Change the parameters with PE2, HE, or other

document processing software and transmit them back to the CNC. The trasnmission software also provides real-time online editing functions.

Clear all MCM parameters to default settings.

Double-click $\boxed{\frac{\text{Auto}}{\text{MDI}}}$ to enter the MDI mode and execute G10 P1000 command.

MCM Parameter Setting List

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
1	0	mm	G54 X-axis 1st work coordinates setting	
2	0	mm	G54 Y-axis 1st Work coordinates setting	
3	0	mm	G54 Z-axis 1st work coordinates setting	
4	0	mm	G54 A-axis 1st work coordinates setting	
5	0	mm	Reserved	
6	0	mm	Reserved	
7	0	mm	G55 X-axis 2nd work coordinates setting	
8	0	mm	G55 Y-axis 2nd work coordinates setting	
9	0	mm	G55 Z-axis 2nd work coordinates setting	
10	0	mm	G55 A-axis 2nd work coordinates setting	
11	0	mm	Reserved	
12	0	mm	Reserved	
13	0	mm	G56 X-axis 3rd work coordinates setting	
14	0	mm	G56 Y-axis 3rd work coordinates setting	
15	0	mm	G56 Z-axis 3rd work coordinates setting	
16	0	mm	G56 A-axis 3rd work coordinates setting	
17	0	mm	Reserved	
18	0	mm	Reserved	
19	0	mm	G57 X-axis 4th work coordinates setting	
20	0	mm	G57 Y-axis 4th work coordinates setting	
21	0	mm	G57 Z-axis 4th work coordinates setting	
22	0	mm	G57 A-axis 4th work coordinates setting	
23	0	mm	Reserved	
24	0	mm	Reserved	
25	0	mm	G58 X-axis 5th work coordinates setting	
26	0	mm	G58 Y-axis 5th work coordinates setting	
27	0	mm	G58 Z-axis 5th work coordinates setting	
28	0	mm	G58 A-axis 5th work coordinates setting	
29	0	mm	Reserved	
30	0	mm	Reserved	
31	0	mm	G58 X-axis 6th work coordinates setting	
32	0	mm	G58 Y-axis 6th work coordinates setting	
33	0	mm	G58 Z-axis 6th work coordinates setting	
34	0	mm	G58 A-axis 6th work coordinates setting	
35	0	mm	Reserved	
36	0	mm	Reserved	
37	0	mm	X-axis 1st tool length compensation setting	
38	0	mm	Y-axis 1st tool length compensation setting	
39	0	mm	Z-axis 1st tool length compensation setting	
40	0	mm	A-axis 1st tool length compensation setting	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
41	0	mm	Reserved	
42	0	mm	Reserved	
43	0	mm	1st tool radius compensation setting	
44	0	mm	X-axis 2nd tool length compensation setting	
45	0	mm	Y-axis 2nd tool length compensation setting	
46	0	mm	Z-axis 2nd tool length compensation setting	
47	0	mm	A-axis 2nd tool length compensation setting	
48	0	mm	Reserved	
49	0	mm	Reserved	
50	0	mm	2nd tool radius compensation setting	
51	0	mm	X-axis 3rd tool length compensation setting	
52	0	mm	Y-axis 3rd tool length compensation setting	
53	0	mm	Z-axis 3rd tool length compensation setting	
54	0	mm	A-axis 3rd tool length compensation data setting	
55	0	mm	Reserved	
56	0	mm	Reserved	
57	0	mm	3rd tool radius compensation setting	
58	0	mm	X-axis 4th tool length compensation setting	
59	0	mm	Y-axis 4th tool length compensation setting	
60	0	mm	Z-axis 4th tool length compensation setting	
61	0	mm	A-axis 4th tool length compensation setting	
62	0	mm	Reserved	
63	0	mm	Reserved	
64	0	mm	4th tool radius compensation setting	
65	0	mm	X-axis 5th tool length compensation setting	
66	0	mm	Y-axis 5th tool length compensation setting	
67	0	mm	Z-axis 5th tool length compensation setting	
68	0	mm	A-axis 5th tool length compensation setting	
69	0	mm	Reserved	
70	0	mm	Reserved	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
71	0	mm	5th tool radius compensation setting	
72	0	mm	X-axis 6^{th} tool length compensation setting	
73	0	mm	Y-axis 6^{th} tool length compensation setting	
74	0	mm	Z-axis 6^{th} tool length compensation setting	
75	0	mm	A-axis 6^{th} tool length compensation setting	
76	0	mm	Reserved	
77	0	mm	Reserved	
78	0	mm	6th tool radius compensation setting	
79	0	mm	X-axis 7^{th} tool length compensation setting	
80	0	mm	Y-axis 7^{th} tool length compensation setting	
81	0	mm	Z-axis 7^{th} tool length compensation setting	
82	0	mm	A-axis 7^{th} tool length compensation setting	
83	0	mm	Reserved	
84	0	mm	Reserved	
85	0	mm	7th tool radius compensation setting	
86	0	mm	X-axis 8^{th} tool length compensation setting	
87	0	mm	Y-axis 8^{th} tool length compensation setting	
88	0	mm	Z-axis 8^{th} tool length compensation setting	
89	0	mm	A-axis 8^{th} tool length compensation setting	
90	0	mm	Reserved	
91	0	mm	Reserved	
92	0	mm	8th tool radius compensation setting	
93	0		Master/Slave Mode Setting 0=None, 1=X-axis, 2=Y-axis, 3=Z-axis, 4=A=axis, 256=Non-Stop between Blocks	
94	0	mm	G28 X-axis 1st ref. point setting	
95	0	mm	G28 Y-axis 1st ref. point setting	
96	0	mm	G28 Z-axis 1st ref. point setting	
97	0	mm	G28 A-axis 1st ref. point setting	
98	0	mm	Reserved	
99	0	mm	Reserved	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
100	0	mm	G30 X-axis 2nd ref. point setting	
101	0	mm	G30 Y-axis 2nd ref. point setting	
102	0	mm	G30 Z-axis 2nd ref. point setting	
103	0	mm	G30 A-axis 2nd ref. point setting	
104	0	mm	Reserved	
105	0	mm	Reserved	
106	0	mm	X-axis backlash compensation setting Range:0~9.999 mm.	
107	0	mm	Y-axis backlash compensation setting. Range:0~9.999 mm.	
108	0	mm	Z-axis backlash compensation setting. Range:0~9.999 mm.	
109	0	mm	A-axis backlash compensation setting Range:0~9.999 mm.	
110	0	mm	Reserved	
111	0	mm	Reserved	
112	1000	mm/min	X-axis, JOG speed & power-on G01 speed setting	
113	1000	mm/min	Y-axis, JOG speed setting	
114	1000	mm/min	Z-axis, JOG speed setting	
115	1000	mm/min	A-axis, JOG speed setting	
116	1000	mm/min	Reserved	
117	1000	mm/min	Reserved	
118	100	Pulse	X-axis resolution denominator setting (Encoder pulse per eevolution)	
119	100	μm	X-axis resolution numerator setting (Ball screw pitch)	
120	100	Pulse	Y-axis resolution denominator setting (Encoder pulse per revolution)	
121	100	μm	Y-axis resolution numerator setting (Ball screw pitch)	
122	100	Pulse	Z-axis resolution denominator setting (Encoder pulse per revolution)	
123	100	μm	Z-axis resolution numerator setting (Ball screw pitch)	
124	100	Pulse	A-axis resolution denominator setting (Encoder pulse per revolution)	
125	100	μm	A-axis resolution numerator setting (Ball screw pitch)	
126	100	Pulse	Reserved	
127	100	μm	Reserved	
128	100	Pulse	Reserved	
129	100	μm	Reserved	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
130	0		X-axis homing direction when tool returning to machine origin. 0=positive, 1=negative.	
131	0		Y-axis homing direction when tool returning to machine origin. 0=positive, 1=negative	
132	0		Z-axis homing direction when tool returning to machine origin. 0=positive, 1=negative	
133	0		A-axis homing direction when tool returning to machine origin. 0=positive, 1=negative	
134	0		Reserved	
135	0		Reserved	
136	2500	mm/min	X-axis returning to machine origin, first velocity	
137	2500	mm/min	Y-axis returning to machine origin, first velocity	
138	2500	mm/min	Z-axis returning to machine origin, first velocity	
139	2500	mm/min	A-axis returning to machine origin, first velocity	
140	2500	mm/min	Reserved	
141	2500	mm/min	Reserved	
142	40	mm/min	X-axis encoder homing grid speed when tool returning to machine origin.	
143	40	mm/min	Y-axis encoder homing grid speed when tool returning to machine origin.	
144	40	mm/min	Z-axis encoder homing grid speed when tool returning to machine origin.	
145	40	mm/min	A-axis encoder homing grid speed when tool returning to machine origin.	
146	40	mm/min	Reserved	
147	40	mm/min	Reserved	
148	10000	mm/min	X-axis maximum feed rate setting	
149	10000	mm/min	Y-axis maximum feed rate setting	
150	10000	mm/min	Z-axis maximum feed rate setting	
151	10000	mm/min	A-axis maximum feed rate setting	
152	10000	mm/min	Reserved	
153	10000	mm/min	Reserved	
154	0		X-axis, direction of motor rotation, 0=positive, 1=negative	
155	0		Y-axis, direction of motor rotation, 0=positive, 1=negative	
156	0		Z-axis, direction of motor rotation, 0=positive, 1=negative	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
157	0		A-axis, direction of motor rotation, 0=positive, 1=negative	
158	0		Reserved	
159	0		Reserved	
160	4		X-axis, encoder pulse multiplication factor, 1, 2, or 4	
161	4		Y-axis, encoder pulse multiplication factor, 1, 2, or 4	
162	4		Z-axis, encoder pulse multiplication factor, 1, 2, or 4	
163	4		A-axis, encoder pulse multiplication factor, 1, 2, or 4	
164	4		Reserved	
165	4		Reserved	
166	100	msec	G00 accel./decel. time constant setting, linear (4~512)	
167	100	msec	G01 accel./decel. time constant setting, linear(10~1024)	
168	9600		RS232C baud rate setting	
169	0		Program execution times (M02, M30, M99)	
170	0		Program execution counter limit (maximum 9,999,999)	
171	9999999	mm	X-axis software over travel limit (+)	
172	9999999	mm	Y-axis software over travel limit (+)	
173	9999999	mm	Z-axis software over travel limit (+)	
174	9999999	mm	A-axis software over travel limit (+)	
175	9999999	mm	Reserved	
176	9999999	mm	Reserved	
177	-9999999	mm	X-axis software over travel limit (-)	
178	-9999999	mm	Y-axis software over travel limit (-)	
179	-9999999	mm	Z-axis software over travel limit (-)	
180	-9999999	mm	A-axis software over travel limit (-)	
181	-9999999	mm	Reserved	
182	-9999999	mm	Reserved	
183	0	mm	X-axis machine origin shift value	
184	0	mm	Y-axis machine origin shift value	
185	0	mm	Z-axis machine origin shift value	
186	0	mm	A-axis machine origin shift value	
187	0	mm	Reserved	
188	0	mm	Reserved	
189			Reserved, do not use!	
190	0		X-axis program coordinates clearing when encountering M02, M30, M99	
191	0		Y-axis program coordinates clearing when encountering M02, M30, M99	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
192	0		Z-axis program coordinates clearing when encountering M02, M30, M99	
193	0		A-axis program coordinates clearing when encountering M02, M30, M99	
194	0		Reserved	
195	0		Reserved	
196	1		X-axis incr./abs. program command, 0=incremental, 1=absolute	
197	1		Y-axis incr./abs. program command, 0=incremental, 1=absolute	
198	1		Z-axis incr./abs. program command, 0=incremental, 1=absolute	
199	1		A-axis abs. program command, 1=absolute	
200	1		Reserved	
201	1		Reserved	
202			Reserved, do not use!	
203	64	Pulse	X-axis position gain, standard=64	
204	64	Pulse	Y-axis position gain, standard=64	
205	64	Pulse	Z-axis position gain, standard=64	
206	64	Pulse	A-axis position gain, standard=64	
207	64	Pulse	Reserved	
208	64	Pulse	Reserved	
209	10	Pulse	X-axis break-over point of position gain, standard=10	
210	10	Pulse	Y-axis break-over point of position gain, standard=10	
211	10	Pulse	Z-axis break-over point of position gain, standard=10	
212	10	Pulse	A-axis break-over point of position gain, standard=10	
213	10	Pulse	Reserved	
214	10	Pulse	Reserved	
215	0		PLC R000~R199 data saved/not saved during power failure. 0=No, 256=Yes(R00~R199)	
216	1000.000	mm	Servo homing grid length when tool returning to home, X-axis.	
217	1000.000	mm	Servo homing grid length when tool returning to home, Y-axis.	
218	1000.000	mm	Servo homing grid length when tool returning to home, Z-axis.	
219	1000.000	mm	Servo homing grid length when tool returning to home, A-axis.	
220			Reserved, do not use!	
221			Reserved, do not use!	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
222	0		Accel./decel. type. 0=Linear, 1=S curve	
223	100	msec	G99 accel./decel. time	
224	4096	Pulse	Spindle encoder pulses	
225	3000	rpm	Spindle motor rpm at output voltage=10V	
226	100	msec	Spindle accel./decel. time	
227	0		Start number for automatic generation of program block numbers	
228	0		Increment of numbers during automatic generation of program block numbers	
229	100		Denominator of feed-rate multiplier when in MPG test mode	
230	100		Numerator of feed-rate multiplier when in MPG test mode	
231	0	0/1	X-axis encoder homing grid direction when tool returning to home	
232	0	0/1	Y-axis encoder homing grid direction when tool returning to home	
233	0	0/1	Z-axis encoder homing grid direction when tool returning to home	
234	0	0/1	A-axis encoder homing grid direction when tool returning to home	
235	0	0/1	Reserved	
236	0	0/1	Reserved	
237	0	Pulse	Servo-error	
238	64	msec	MPG accel./decel. time (4 ... 512)	
239	0	Rad/sec	G02 & G03 FEED-LIMIT	
240	0		0/256 Metric/ imperial system	
241	100		X-axis MPG hand-wheel resolution denominator setting	
242	100		X-axis MPG hand-wheel resolution numerator setting	
243	100		Y-axis MPG hand-wheel resolution denominator setting	
244	100		Y-axis MPG hand-wheel resolution numerator setting	
245	100		Z-axis MPG hand-wheel resolution denominator setting	
246	100		Z-axis MPG hand-wheel resolution numerator setting	
247	100		A-axis MPG hand-wheel resolution denominator setting	
248	100		A-axis MPG hand-wheel resolution numerator setting	
249	100		Reserved	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
250	100		Reserved	
251	100		Reserved	
252	100		Reserved	
253	0		X is the linear axis when X-axis=0 X is the rotating axis when X-axis ≠0	
254	0		Y is the linear axis when Y-axis=0 Y is the rotating axis when Y-axis ≠0	
254	0		Y is the linear axis when Y-axis=0 Y is the rotating axis when Y-axis ≠0	
255	0		Z is the linear axis when Z-axis=0 Z is the rotating axis when Z-axis ≠0	
256	0		A is the linear axis when A-axis=0 A is the rotating axis when A-axis ≠0	
257	0		Reserved	
258	0		Reserved	
259	3		Arc cutting error.(ideal value=1)	
260	360000		Display of coordinates per servo spindle rotation	
261 : 281			Reserved	
282	2.000	mm	Mill mode: Drilling tool retraction clearance	
283			for system use	
284			System ID	
285	0	msec	X-axis accel./decel. time setting	
286	0	msec	Y-axis accel./decel. time setting	
287	0	msec	Z-axis accel./decel. time setting	
288	0	msec	A-axis accel./decel. time setting	
289			Reserved	
290			Reserved	
291	0		X-axis ball screw pitch error compensation setting	
292	0		Y-axis ball screw pitch error compensation setting	
293	0		Z-axis ball screw pitch error compensation setting	
294	0		A-axis ball screw pitch error compensation setting	
295	0		Reserved	
296	0		Reserved	
297	20000	mm	X-axis segment length of pitch error compensation	
298	20000	mm	Y-axis segment length of pitch error compensation	

MCM Parameter No.	Factory Default Settings	Unit	Description	Remarks
299	20000	mm	Z-axis segment length of pitch error compensation	
300	20000	mm	A-axis segment length of pitch error compensation	
301	20000	mm	Reserved	
302	20000	mm	Reserved	
303-342	0		X-axis 40 segments compensation	
343-382	0		Y-axis 40 segments compensation	
383-422	0		Z-axis 40 segments compensation	
423-462	0		A-axis 40 segments compensation	
463-542	0		Reserved	

Note: Press PAGE↑ or PAGE↓ once to skip 10 parameters.

4.2 Description of MCM Parameters

In this section the decimal format of parameters is described based on the 4/3 format.

The work coordinates of MCM parameters #1~36 are set by G54~G59. That is to set the work origin of the work coordinates, the machine coordinates of the work origin are relative to the machine coordinates with the machine origin as the zero point.

1. G54 X-axis 1st work coordinates setting.
   Format=□.□□□ Unit: mm

2. G54 Y-axis 1st Work coordinates setting.
   Format=□.□□□Unit: mm

3. G54 Z-axis 1st work coordinates setting.
   Format=□.□□□Unit: mm

4. G55 A-axis 1st work coordinates setting.
   Format=□.□□□Unit: mm

7. G56 X-axis 2nd work coordinates setting.
   Format=□.□□□Unit: mm

8. G56 Y-axis 2nd work coordinates.
   Format=□.□□□Unit: mm

9. G56 Z-axis 2nd work coordinates setting.
   Format=□.□□□Unit: mm

10. G57 A-axis 2nd work coordinates setting.
    Format=□.□□□Unit: mm

13. G58 X-axis. 3rd work coordinates setting.
    Format=□.□□□Unit: mm

14. G58 Y-axis 3rd work coordinates setting.
    Format=□.□□□Unit: mm

15. G58 Z-axis 3rd work coordinates setting.
    Format=□.□□□Unit: mm

16. G59 A-axis 3rd work coordinates setting.
    Format=□.□□□ Unit: mm

19. G59 X-axis 4th work coordinates setting.
Format=□.□□□Unit: mm

20. G59 Y-axis 4th work coordinates setting.
Format=□.□□□Unit: mm

21. G59 Z-axis 4th work coordinates setting.
Format=□.□□□Unit: mm

22. G59 A-axis 4th work coordinates setting.
Format=□.□□□Unit: mm

25. G59 X-axis 5th work coordinates setting.
Format=□.□□□ Unit: mm

26. G59 Y-axis 5th work coordinates setting.
Format=□.□□□Unit: mm

27. G59 Z-axis 5th work coordinates setting.
Format=□.□□□ Unit: mm

28. G59 A-axis 5th work coordinates setting.
Format=□.□□□ Unit: mm

31. G59 X-axis 6th work coordinates setting.
Format=□.□□□ Unit: mm

32. G59 Y-axis 6th work coordinates setting.
Format=□.□□□Unit: mm

33. G59 Z-axis 6th work coordinates setting.
Format=□.□□□Unit: mm

34. G59 A-axis 6th work coordinates setting.
Format=□.□□□Unit: mm

37. X-axis 1st tool length compensation.
Format=□.□□□Unit: mm

38. Y-axis 1st tool length compensation.
Format=□.□□□Unit: mm

39. Z-axis 1st tool length compensation.
Format=□.□□□Unit: mm

40. A-axis 1st tool length compensation.
Format=□.□□□Unit: mm

43. Radius compensation
Format=□.□□□Unit: mm

44. X-axis 2nd tool length compensation.
Format=□.□□□Unit: mm

45. Y-axis 2nd tool length compensation.
Format=□.□□□Unit: mm

46. Z-axis 2nd tool length compensation.
Format=□.□□□Unit: mm

47. A-axis 2nd tool length compensation.
Format=□.□□□Unit: mm

48. Radius compensation
Format=□.□□□Unit: mm

49. X-axis 3rd tool length compensation.
Format=□.□□□Unit: mm

50. Y-axis 3rd tool length compensation.
Format=□.□□□Unit: mm

51. Z-axis 3rd tool length compensation.
Format=□.□□□Unit: mm

52. A-axis 3rd tool length compensation.
Format=□.□□□Unit: mm

53. Radius compensation
Format=□.□□□Unit: mm

54. X-axis 4th tool length compensation.
Format=□.□□□Unit: mm

55. Y-axis 4th tool length compensation.
Format=□.□□□Unit: mm

56. Z-axis 4th tool length compensation
Format=□.□□□Unit: mm

57. A-axis 4th tool length compensation.
Format=□.□□□Unit: mm

58. Radius compensation
Format=□.□□□Unit: mm

59. X-axis 5th tool length compensation.
Format=□.□□□Unit: mm

60. Y-axis 5th tool length compensation.
Format=□.□□□Unit: mm

61. Z-axis 5th tool length compensation .
Format=□.□□□Unit: mm

62. A-axis 5th tool length compensation.
Format=□.□□□Unit: mm

63. Radius compensation.
Format=□.□□□Unit: mm

64. X-axis 6th tool length compensation.
Format=□.□□□Unit: mm

65. Y-axis 6th tool length compensation.
Format=□.□□□Unit: mm

66. Z-axis 6th tool length compensation.
Format=□.□□□Unit: mm

67. A-axis 6th tool length compensation.
Format=□.□□□Unit: mm

78. Radius compensation
Format=□.□□□Unit: mm

79. X-axis 7th tool length compensation.
Format=□.□□□Unit: mm

80. Y-axis 7th tool length compensation.
Format=□.□□□Unit: mm

81. Z-axis 7th tool length compensation.
Format=□.□□□Unit: mm

82. A-axis 7th tool length compensation.
Format=□.□□□Unit: mm

85. Radius compensation
Format=□.□□□Unit: mm

86. X-axis 8th tool length compensation.
Format=□.□□□Unit: mm

87. Y-axis 8th tool length compensation.
Format=□.□□□Unit: mm

88. Z-axis 8th tool length compensation.
Format=□.□□□Unit: mm

89. A-axis 8th tool length compensation.
Format=□.□□□Unit: mm

92. Radius compensation
Format=□.□□□Unit: mm

93. Master/Slave mode setting.
Format=□.□□□ (Default=0)

0 = Set servomotor accel /decel type to CNC standard
1 = Set to Master/Slave mode, X=Master, Y-axis, Z-axis, A axis
2 = Set to Master/Slave mode, Y=Master, X-axis, Z-axis, A axis
3 = Set to Master/Slave mode, Z=Master, X-axis, Z-axis, A axis.
4 = Set to Master/Slave mode, A=Master, X-axis, Y-axis, Z axis.
256= Non-stop Operation among blocks

MCM parameters #94~#105 are for reference point settings. The setting of each reference point is relative to machine coordinates with the machine origin as the zero point.

94. G28 X-axis 1st reference point setting.

95. G28 Y-axis 1st reference point setting.

96. G28 Z-axis 1st reference point setting.

97. G28 A-axis 1st reference point setting.

100. G30 X-axis 2nd reference point setting.

101. G30 Y-axis 2nd reference point setting.

102. G30 Z-axis 2nd reference point setting.

103. G30 A-axis 2nd reference point setting.
Format=□.□□□ (Default=0.000) Unit: mm

106. X-axis backlash compensation setting.

107. Y-axis backlash compensation setting.

108. Z-axis backlash compensation setting.

109. A-axis backlash compensation setting.
Format=□.□□□ (Default=0) Unit: mm

112. X-axis JOG speed, & power-on G01 speed setting, unit: mm/min

113. Y-axis, JOG speed setting, unit: mm/min

114. Z-axis, JOG speed setting, unit: mm/min

115. A-axis, JOG speed setting, unit: mm/min
Format=□□□□ (Default=1000)

118. X-axis resolution denominator setting.

119. X-axis resolution numerator setting.

120. Y-axis resolution denominator setting.

121. Y-axis resolution numerator setting.

122. Z-axis resolution denominator setting.

123. Z-axis resolution numerator setting.

124. A-axis resolution denominator setting.

125. A-axis resolution numerator setting.

<u>The numerator or denominator of the resolution</u> is set based on the specifications of the mechanical axial gearing (such as ball screw) and the encoder pulses of the servomotor. <u>No change of the setting</u> is allowed without authorization.

$$\text{Resolution} = \frac{\text{Ball screw pitch}}{\text{Motor encoder} \times \text{multiplication}} \times \text{GR}$$

#118~125 #160~163

Ex1:
X-axis = normal linear axis (MCM#253 = 0), ball screw pitch = 5.000 mm
Motor ENCODER = 2500 pulses, multiplication factor = 4 (MCM#160 = 4)
Gear Ratio= 5:1(Servomotor rotates 5 turns = ball screw Rotates 1 turn)

$$\text{Resolution} = \frac{5000}{2500 \times 4} \times \frac{1}{5}$$

$$= \frac{1}{10}$$

X-axis resolution denominator (MCM#118) =10

X-axis resolution numerator (MCM#119) =1

Ex2:
Y-axis = rotating axis (MCM#254 = 1), angle per rotation = 360.000°
Motor encoder=2500 pulses, multiplication factor=4 (MCM#161 = 4)
Gear Ratio= 5:1 **(**Servomotor rotates 5 turns = Y-axis Rotates 1 turn)

$$\text{Resolution} = \frac{360000}{2500 \times 4} \times \frac{1}{5}$$

$$= \frac{36}{5}$$

Y-axis resolution denominator (MCM#120) =5
Y-axis resolution numerator (MCM#121) =36

Note: when resolution is below 1/100, the software OT limit must be within the range from –999999 to 999999. Otherwise an error message displays, which cannot be removed.

Example: Assuming MCM#118=400 MCM#119=2, and X-axis resolution $<$ 1/100, the software OT limit of X-axis (i.e.#171) must be less than 999999 and #177 must be greater than –999999.

130. X-axis homing direction when tool returns to machine origin.
131. Y-axis homing direction when tool returns to machine origin.
132. Z-axis homing direction when tool returns to machine origin.
133. A-axis homing direction when tool returns to machine origin.
Format=□, (Default=0)

Setting = 0, tool returns to machine origin in the positive direction.
Setting = 1, tool returns to machine origin in the negative direction

136. X-axis returning to machine origin, first velocity.
137. Y-axis returning to machine origin, first velocity.
138. Z-axis returning to machine origin, first velocity.
139. A-axis returning to machine origin, first velocity.
Format =□□□□ (Default=2500) Unit: mm/min

142. X-axis encoder homing grid velocity when tool returns to machine origin.
143. Y-axis encoder homing grid velocity when tool returns to machine origin.
144. Z-axis encoder homing grid velocity when tool returns to machine origin.
145. A-axis encoder homing grid velocity when tool returns to machine origin.
Format=□□□□ (Default=40) Unit: mm/min

HUST H4CL-M Series controller provides three velocities for an axis to return to the machine origin (Fig 7-2)

First velocity: The velocity of X, Y, Z, and A is set respectively in MCM#136~139, while the direction is set in MCM#130~133.

Second velocity: The second velocity of X, Y, and Z is set to 1/4 of the first velocity when it drops to 0. The direction is set in MCM#231~234.

Third velocity: This velocity is the encoder homing grid velocity and set in MCM#142~145. The direction is set in #231~234.

When the tool returns to home, the machine moves to the **limit switch** at the first velocity and the length of the **limit switch must be greater than the distance required for deceleration.** Otherwise the machine will run over the limit switch and result in a homing error.

The equation to calculate the length of the limit switch is:
the length of limit switch ≥ (FDCOMxACC)÷60000

Note: ① FDCOM = Homing velocity 1, (MCM #136~#139,)
② ACC = G01 accel./decel. time (MCM #167)
③ 60000 msec (60 secs Ｘ 1000 = 60000 msec)

Ex.: FDCOM, the homing velocity 1 = 3000 mm / min
ACC, accel/decel time = 100 ms
Minimal Length of Limit Switch = (3000 x 100) ÷ 60000 = 5 mm

When C063=1 (1-pulse), it commands the controller to do homing operation.
R232 =1, X-axis homing operation, R232=2, Y-axis homing operation,
R232=4 Z-axis homing operation, R232=8, A-axis homing operation,
R232=15, X, Y, Z, A simultaneous homing operation.

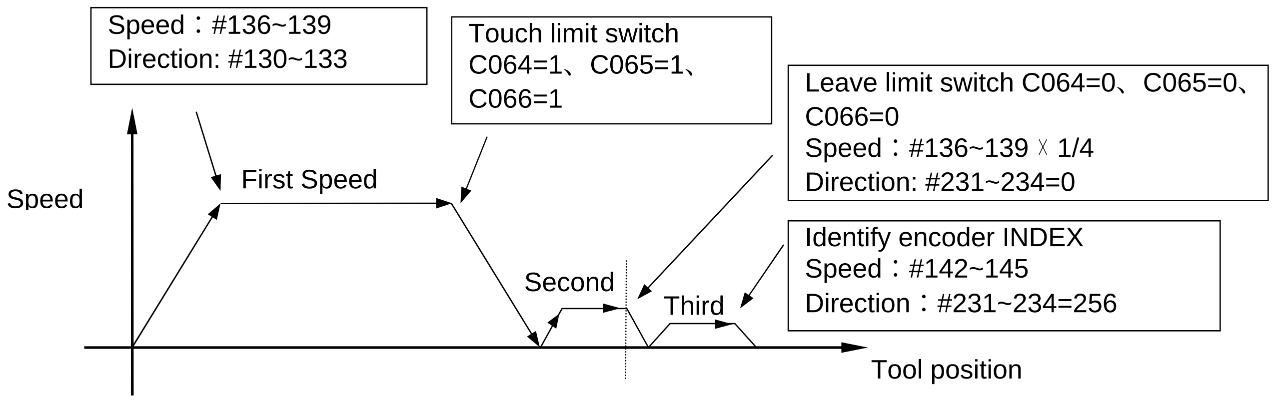


Fig 4-2 (A) Homing Speed and Homing Grid Direction

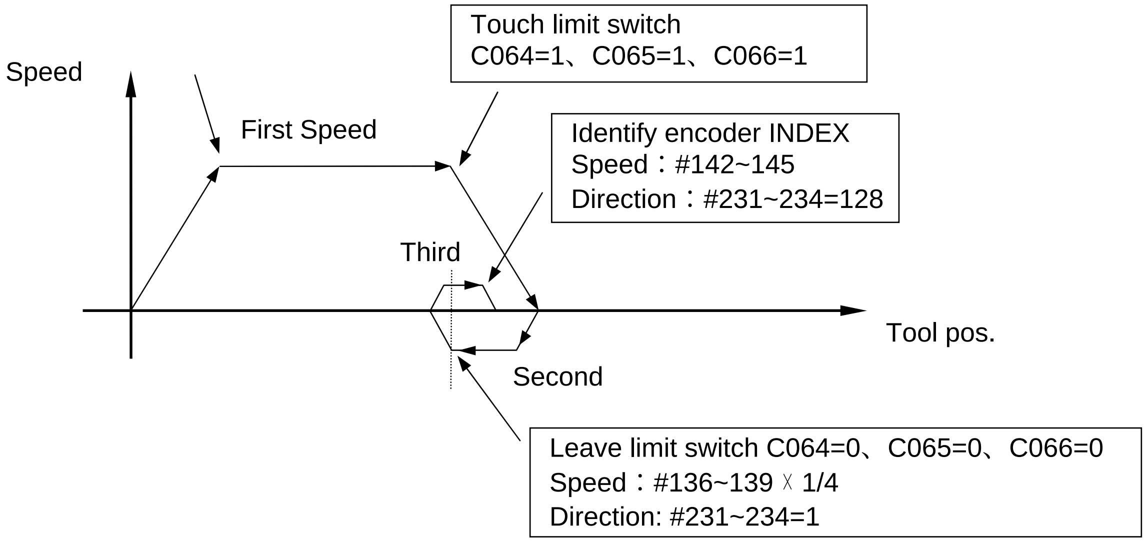


Fig 4-2 (B) Homing Speed and Homing Grid Direction

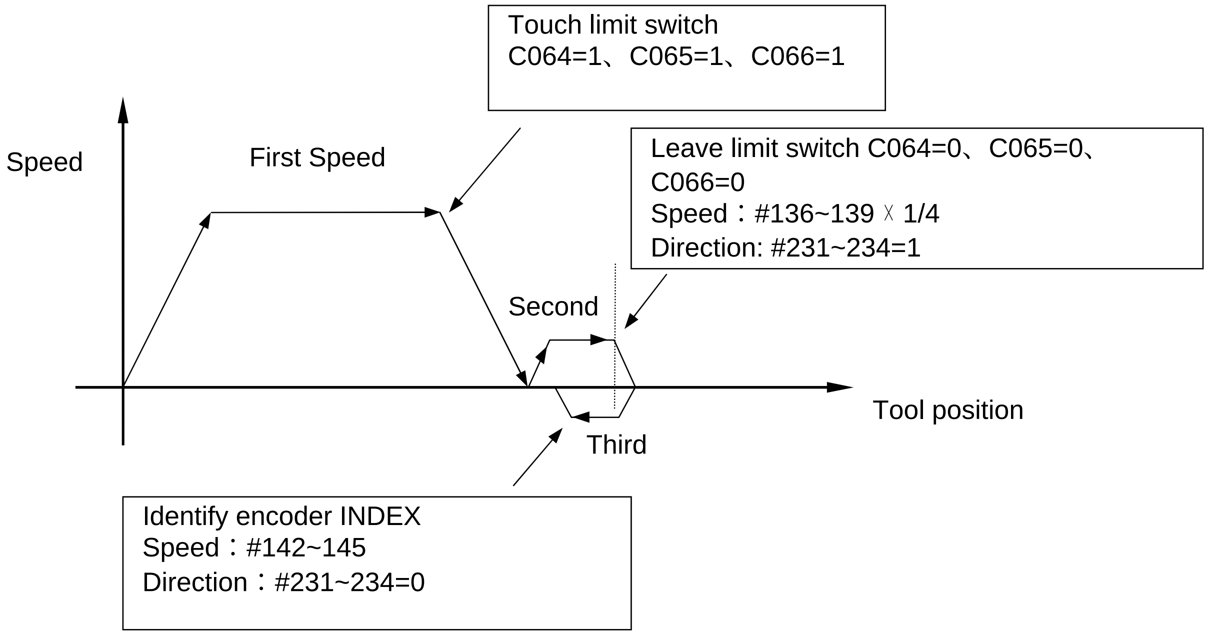


Fig 4-2 (C) Homing Speed and Homing Grid Direction

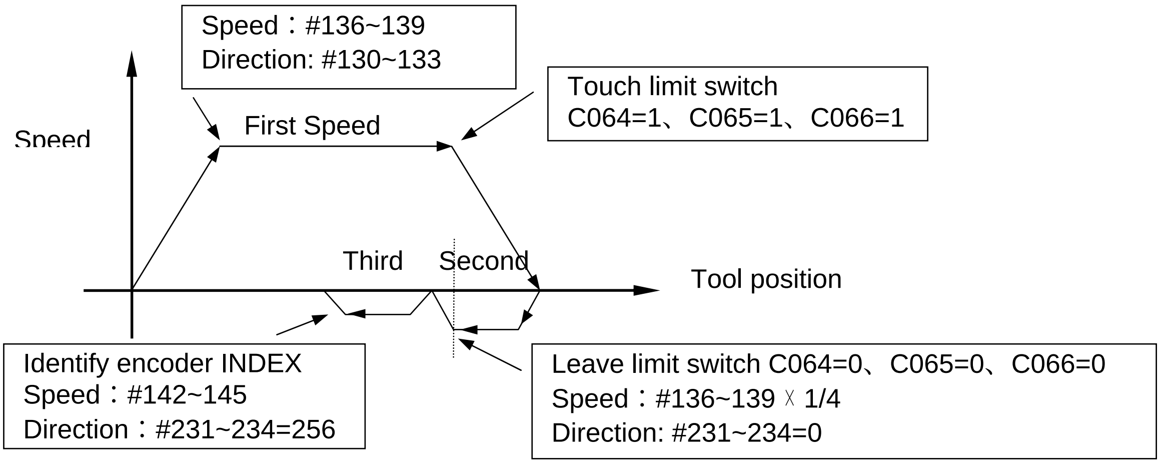


Fig 4-2 (D) Homing Speed and Homing Grid Direction

148. X-axis maximum feed rate setting.
149. Y-axis maximum feed rate setting.
150. Z-axis maximum feed rate setting.
151. A-axis maximum feed rate setting.

Format=□□□□□ (Default =10000) Unit: mm/min
Note: the setting is an integer format.

If you set the Z-axis as 5000, which means that your highest feed rate will be 5000 mm per minute.

The feed speed limit can be calculated from the following equation:

Fmax = 0.95 x RPM (the rated rpm of the servomotor) x Pitch (ball screw pitch) ÷ GR

(Recommended value)

Ex: The highest rotation speed of the servomotor on X-axis is 3,000 rpm, the ball screw pitch is 5mm.(Servo motor rotations:5; ball screw rotations:1)
Fmax = 0.95 x 3000 x 5 ÷ 5= 2850
Therefore, it is recommended to set MCM #148=2850.

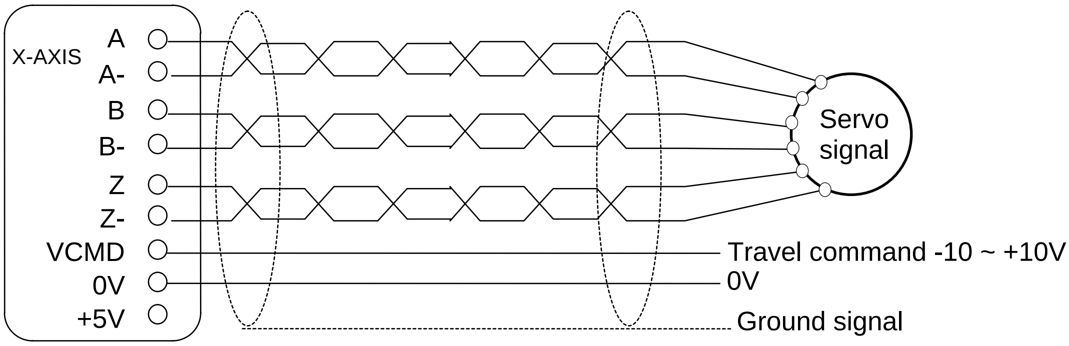
154. X-axis, direction of motor rotation, .
155. Y-axis, direction of motor rotation, .
156. Z-axis, direction of motor rotation, .
157. A-axis, direction of motor rotation, .
Format=□ (Default=0)

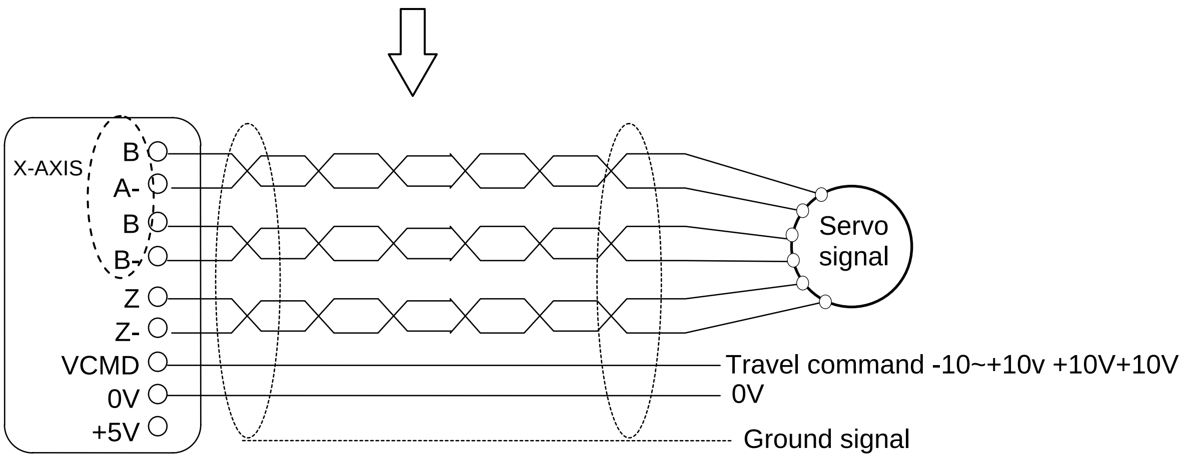
Setting = 0, Motor rotates in the positive direction. (CW)
Setting = 1, Motor rotates in the negative direction. (CCW)

Design of the machine varies among manufacturers, and the position of the servomotor on the ball screw varies too. MCM #154~157 can be used to adjust the direction if the machine moves in opposite after the servomotor is installed. The parameters will affect the direction of the machine coordinate

> Note: the explanation below is not related to the setting of the motor direction but important to the wiring of the machine.
>
> Since design of the driver varies among manufacturers, the position signal encoder of a new driver might not be able to fit with the controller. In this case, the servomotor will become **divergent (motor rotates at high rpm)** because of the error of feedback. For this reason, Be sure to separate the servomotor and the machine until the accuracy of motor rotation direction is confirmed. When motor **divergence** occurs, change the position of A and (A-) with B and (B-)

Example:





160. X-axis, encoder pulse multiplication factor
161. Y-axis, encoder pulse multiplication factor
162. Z-axis, encoder pulse multiplication factor

163. A-axis, encoder pulse multiplication factor
Format=□ (Default=4)

Setting = 1, encoder pulses multiplied by 1.
Setting = 2, encoder pulses multiplied by 2.
Setting = 4, encoder pulses is multiplied by 4.

Only one of the four values can be used for MCM #160~163.

> Note:
>
> The setting of the multiplication factor is dependent on the rigidness of the structure. The vibration of the motor during feeding of the power to the structure suggests that the rigidity of the structure is too high. This can be corrected by reducing the multiplication factor of the axis concerned.

Example: The servomotor encoder pulses at 2000 per rotation. When MCM #161 is set to 2, the pulses of Y-axis are 2000 x 2 = 4000.

166. G00 accel/decel time constant setting.
Format=□□□ (Default=100) Unit: Millisecond (msec.)
Setting range: 4~512 ms.

167. G01 accel./decel. time constant setting.
Format=□□□ (Default=100) Unit: Millisecond (msec.)
Setting range: 10 ~ 1024 ms. 100 ms is the recommended for G00 and G01.
If MCM #222 setting = 0: G00 linear accel/decel.
If MCM #222 setting = 1, G00 "S" curve accel/decel. In this case, the setting remains same as the linear type, but the actual accel/decel time is the double of the setting.

168. RS232C baud rate setting.
Format=□□□□(Default = 38400)

Set RS232C baud rate to one of the followings :
4800,9600,19200,38400,57600
The default baud rate is 38400, which stands for 38400 bits per second.
In addition to the baud rate, configure the following parameters for RS232C:

Parity	-- Even
Stop Bits	-- 2 bits
DATA BITS	– 7 BITS

169. Program execution times (M02, M30, M99)
Format=□□□□□□□(Default=0)

Record how many times the current program has been executed. If PLC C030 =1 (refer to PLC Wiring Manual), one (1) should be added to the

counter every time when the program runs in M02, M30 or M99.
The counter limit will be compared to the MCM #170 value. The program stops running when the two values are equal.

170. Program execution counter limit (M02, M30, M99).
Format=□□□□□□□(Default=0)
Maximum value = 9,999,999.

171. X-axis software OT Limit (+)
172. Y-axis Software OT Limit (+)
173. Z-axis software OT Limit (+)
174. A-axis software OT Limit (+).
Format=□□□□.□□□ (Default=+9999.999) Unit: mm

The set value is the positive distance between software OT and the machine origin.

177. X-axis software OT Limit in (-)
178. Y-axis software OT Limit in (-)
179. Z-axis software OT Limit in (-)
180. A-axis software OT Limit in (-)
Format=□□□□.□□□ (Default=-9999.999) Unit: mm/min
The set value is the negative distance between software OT and the machine origin).

The concept and description of over travel limit:

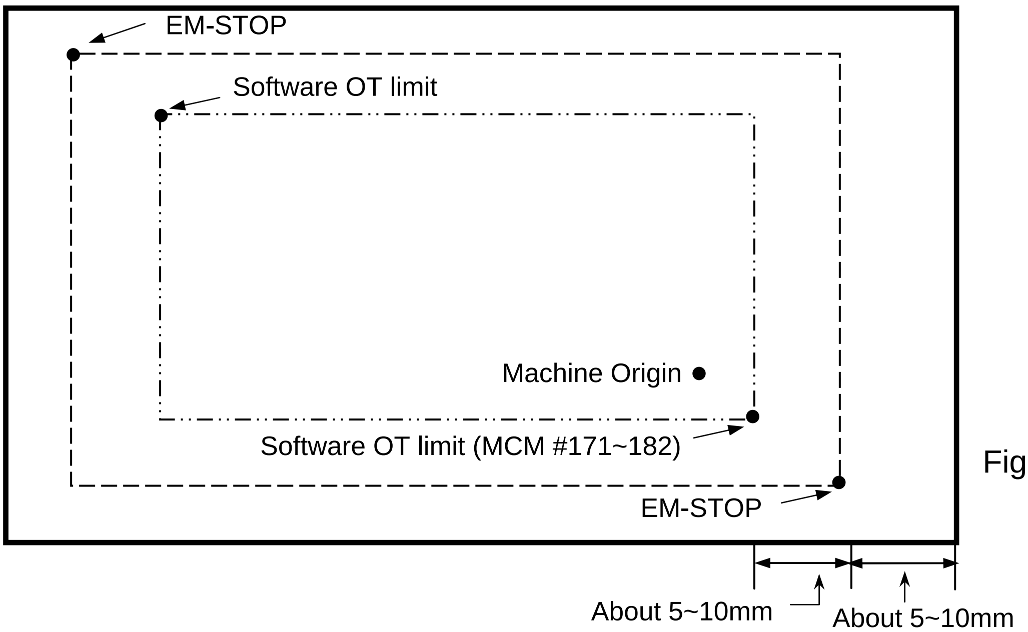


Fig 4-3

Note:The distance between the software OT Limit and EM-STOP is about 5~10mm

183. X-axis machine orgin shift value
184. Y-axis machine orgin shift value
185. Z-axis machine orgin shift value
186. A-axis machine orgin shift value
Format=□□□□.□□□ (Default=0.000) Unit: mm
The distance between the machine origin and shift value.

189. Reserved, do not use.

190. X-axis program coordinate clearing when encountering M02, M30, M99
191. Y-axis program coordinate clearing when encountering M02, M30, M99.
192. Z-axis program coordinate clearing when encountering M02, M30, M99
193. A-axis program coordinate clearing when encountering M02, M30, M99
Format=□□□□(Default=0)

0 = Program coordinates are not cleared when encountering M02, M30, M99.
1= Program coordinates are cleared when encountering M02, M30.
2 = Program coordinates are cleared when encountering M99.
3 = Program coordinates are cleared when encountering M02, M30, M99.

196. X-axis incremental/absolute X program command
Format=□(Default=1)

Settings 0=X is an incremental coordinate and U is invalid.
1=X is an absolute coordinate and U is an incremental coordinate.

197. Y-axis incremental/absolute Y program command
Format=□(Default=1)

Settings 0=Y is an incremental coordinate and V is invalid.
1=Y is an absolute coordinate and V is an incremental coordinate.

198. Z-axis incremental/absolute Z program command
Format=□(Default=1)

Settings 0=Z is an incremental coordinate and W is invalid.
1=Z is an absolute coordinate and W is an incremental coordinate.

199. A-axis absolute program command. No incremental command is possible.
G90 and G91 must apply.

*Description 1: After the parameter is configured, execute **G01** X***,Y***,Z*** F***. The program will execute axial motions according to the pre-set incremental or absolute coordinates.
The incremental address codes of X, Y, Z are U, V, W. Unlike X, Y, Z, no change between incremental and absolute coordinates is possible for A, as it has no incremental address code. In this case, G90 and G91 must apply.

*Description 2: When incremental address codes are used, the X, Y, and Z axes must be set to 1 as the absolute coordinate to give U, V, and W commands.

*Description 3: If a four-axis absolute or incremental change is required in conjunction with G90, G91, the absolute or incremental coordinate of X, Y, Z, and A blocks will be determined by G90, G91 no matter whether the parameter are set to absolute or incremental values.

Mode specification format:

G90 Absolute coordinates setting
G91 Incremental coordinates setting

1. G90 Absolute Coordinates Setting:
When the G90 command is given, the X, Y, Z, A are absolute coordinates and all axial blocks feed based on these absolute coordinates (Ex. 1). The incremental address codes U, V, W are applicable to G90. When X, Y, Z are changed to U, V, W, the X, Y, X axial feeding becomes incremental, while A remains feeding based on the absolute coordinates. If the program contains G90, it must also contain G91 to terminate G90. Otherwise, restart is required to terminate G90.

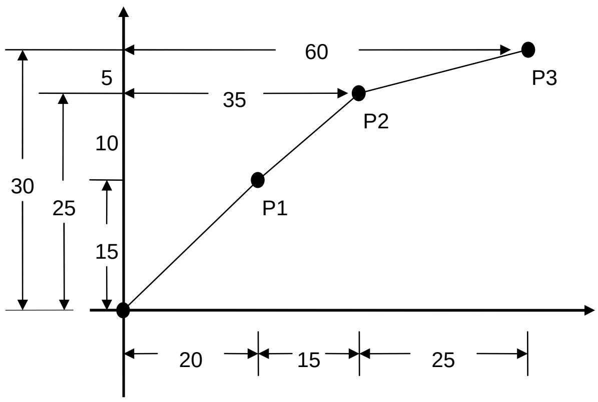
Ex 1: G90 absoulte coordinate settting

```
N1 G90
N2 G1 X20.000 Y15.000      ....   P0 to P1
N3 X35.000 Y25.000         ....   P1 to P2
N4 X60.000 Y30.000         ....   P2 to P3
```

2. G91 Incremental Coordinate Setting:
When the G91 command is given, the X, Y, Z, A are incremental coordinates and all axial blocks will feed based on these incremental coordinates (Ex. 2). X, Y, Z represent increments in the G91 mode, so no U, V, W are required when giving the G91 command. **When U, V, W are used, no axial motion will be executed.** If the program contains G91, it must also contain G90 to terminate G91. Otherwise, restart is required to terminate G91.

Ex 2: G91 incremental coordinate setting

N1 G91
N2 G1 X20.000 Y15.000 .... P0 to P1
N3 X15.000 Y10.000 .... P1 to P2
N4 X25.000 Y5.000 .... P2 to P3



202. Reserved, do not use.

203. X-axis position gain setting

204. Y-axis position gain setting

205. Z-axis position gain setting

206. A-axis position gain setting
Format=□□□(Default=64)
Setting range 8~640

MCM #203~206 are used to set the position gain and 64 is the recommended value. The position gain is important for smooth motor operation. The value should not be changed without permission after it is set up.

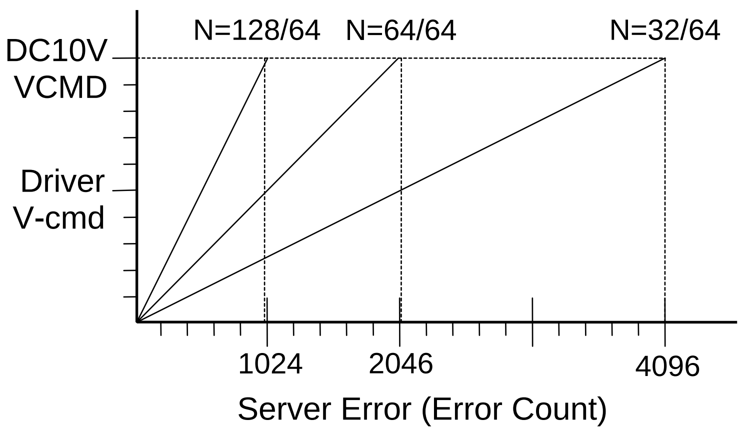


Fig 4-4 Relationship between the Driver V-command and Servo Error

Formula of the position gain and HUST H4CL-M V-command:

$$\text{Position Gain} = \frac{\text{Setting}}{64}$$

$$\text{CNC Controller V-cmd} = \text{GAIN} * \text{Servo Encoder Feedback Error} * \left(\frac{10V}{2048}\right)$$

The HUST controller provides a closed-circuit system. The servo-error is the difference in pulses between the position command and actual motor encoder pulses. The control unit will adjust its V-command appropriately based on this error. Note that the position gain setting has a great effect on the servo stability of the system and the servo response. If:

$\text{Servo-error} > 4096$, a message of "Error 2" will be generated.

In this case, change the parameters in MCM #203~206 and press the "**Reset**" key. If the problem persists, check the motor or wiring and make sure they are properly connected

Adjustment for smooth motor operation:(recommended procedure)

(1) Adjust the servo driver (refer to the driver operating manual)

(2) Adjust the encoder pulse multiplication factor (1,2,4) in MCM #160 163. Normally, the servo error changes between 0 and 1 when the motor is locked. If the servo error changes between 4 and 5, adjust the MCM #160~163 amplicaton factor of; i.e. 4 --> 2 or 2 --> 1.

(3) Adjust the position gain settings in MCM #203~206.

209. X-axis break-over point of position gain, standard value
210. Y-axis break-over point of position gain, standard value
211. Z-axis break-over point of position gain, standard value
212. A-axis break-over point of position gain, standard value
Format=□□□(Default = 10)

Explanation:
The proper setting of this break-over point of position gain ensures the smooth start of the servomotor.
When the servo error is smaller than the value of MCM #209~212, the position gain is 64. Otherwise, the position gain will be calculated based on the value of MCM #203~206.

The value of MCM #209~212 depends on the frictional force of the motor. Generally, if the frictional force is high, the value must be small and vice versa.

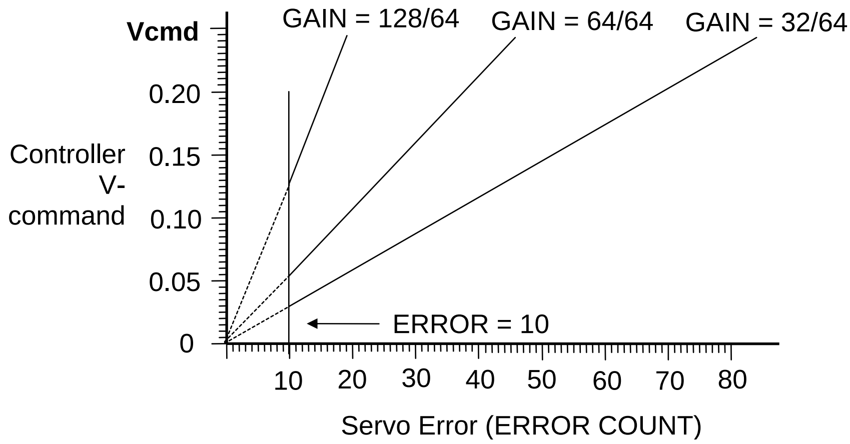


Fig 4-5 Break-over Point of Position Gain

215. PLC R000~R199 data saved/not saved during power failure
Format=□ (Default=0)
Setting = 256, Save   Setting = 0, Not save.

216. Servo homing grid length when tool returns to home, X-axis

217. Servo homing grid length when tool returns to home, Y-axis

218. Servo homing grid length when tool returns to home, Z-axis

219. Servo homing grid length when tool returns to home, A-axis

Format=□□□□.□□□(Default=1000.000)

The max. length for the servomotor to locate the grid signal.

Example:
If the length of the X-axis servomotor rotating 3/4 turn = 5.000 mm, then MCM216 = 5.200
If the length of the Y-axis servomotor rotating 3/4 turn = 5.000 mm, then MCM217 = 5.200
If the length of the Z-axis servomotor rotating 3/4 turn = 5.000 mm, then MCM218 = 5.200
If the length of the A-axis servomotor rotating 3/4 turn = 5.000 mm, then MCM219 = 5.200

※ If the servo cannot locate the grid even though it exceeds the set range, the error message ERR 15 displays.

220. Reserved, do not use

221. Reserved, do not use

222. Accel/decel type.
Format = □ (Default=0)

Setting = 0, Linear   Setting = 1, "S" curve

223. G99 accel/decel time per rotation
Format=□□□ (Default=100), Unit: msec.
Setting Range: 4 ~ 1024 ms.

224. Spindle encoder pulses setting
Format=□□□□ (Default: 4096)

(1) If the spindle is mounted at the X-axis end, then
setting value = specifications of the encoder pulses Ｘ amplification factor of MCM #160.

(2) If the spindle is mounted at the Y-axis end, then
setting value = specifications of encoder pulses Ｘ amplification factor of MCM #161.

(3) If the spindle is mounted at the Z-axis end, then
setting value = specifications of encoder pulses Ｘ amplification factor of MCM #162.

225. Spindle motor rpm at output voltage = 10V
Format=□□□□ (Default=3000), Unit: RPM

226. Spindle accel./decel. time in spindle mode
Format=□□□□ (Default=0), Unit: msec.

227. Start number for automatic generation of program block numbers
Format: S=□ the start number of program block numbers (Default=0)

228. Increment numbers during automatic generation of program block numbers
Format: D=□ the increment between block numbers (Default=0)
If D = 0, the block number is NOT generated automatically.
The block number is generated automatically when you press the INSERT key in the EDIT or TEACH mode. When you press the RESET key, the block number will be recounted from the configured parameter of MCM #128, #129.

Ex: S=0, D=5
Sequence of the automatic generation of block numbers: 5,10,15,20,25, ect.

229. The denominator of feed-rate when in MPG test mode
Format =□□□□□ (Default=100)

230. The numerator of feed-rate when in MPG test mode

Format =□□□□□ (Default=100)

During MPG TEST mode, if the MPG feed-rate in the MPG test mode is not fast as required, set #229 and 230 to increase the speed by multiplying the MPG feed-rate with the ratio of the parameters #229 and #230. <u>The value of #229 and #230 should be within 5-digits and should not be set to 0.</u>

231. X-axis encoder homing grid direction when tool returns to home

232. Y-axis encoder homing grid direction when tool returns to home

233. Z-axis encoder homing grid direction when tool returns to home

234. A-axis encoder homing grid direction when tool returns to home
Format: =□ (Default=0)

Example: The X-axis encoder homing grid direction when tool returns to home (MCM #231) is described as follows:

Setting=0: When the encoder returns from the X-axis to Home, speed 2 and 3 is moving away from the limit switch to search the HOME GRID, and the direction is same as speed 1. See fig.7-2 (A)

Value=1: When the encoder is returns from the X-axis to Home, speed 2 is moving away from the limit switch to search the HOME GRID, and the direction is same as speed 1. See fig.7-2 (B)

Value=128: When the encoder is returning from the X-axis to Home, speed 2 moves away from the limit switch to search the HOME GRID, and the direction is opposite to speed 1. See fig.7-2 (C)

Value=256: When the encoder is returning from the X-axis to Home, speed 2 and 3 moves away from the limit switch to search the HOME GRID, and the direction is opposite to speed 1. See fig.7-2 (D)

237. Servo error
Format=□ (Default=0), Unit: Pulse
In some cases, the servomotor lays behind (Servo Error Count) after NC has given the V-command during execution of the positioning action. This parameter determines the range (i.e. pulses) of the servo error that allows the NC to execute the next action.
Setting=0 The function is off
Setting=1 The function is on no matter how many pulses there are.

238. MPG accel/decel time
Format=□□□ (Default=64), Unit: msec.
Setting range: 4~512 ms.
When the hand-wheel is used in the JOG mode, the accel/decel time of the motor = MCM #236.

239. G02 & G03 FEED-LIMIT

Format=□□□□□ (Default=0), Unit:Rot./sec
Setting=0, no limit
The parameter 239 limits the maximum feed rate for arc cutting commands G02 and G03

240. Metric/ Imperial system
Format=□□□(Default = 0)
Setting = 0, Metric System Measurements.
Setting = 1, Imperial System Measurements.

241. X-axis MPG hand-wheel resolution denominator setting

242. X-axis MPG hand-wheel resolution numerator setting
Format=□□□□(Default value=100). See the following examples.

243. Y-axis MPG hand-wheel resolution denominator setting

244. Y-axis MPG hand-wheel resolution numerator setting
Format=□□□□ (Default value=100). See the following examples.

245. Z-axis MPG hand-wheel resolution denominator setting

246. Z-axis MPG hand-wheel resolution numerator setting
Format=□□□□ (Default value=100). See the following examples.

247. A-axis MPG hand-wheel resolution denominator setting

248. A-axis MPG hand-wheel resolution numerator setting
Format=□□□□(Default value=100). See the following examples.

Ex: The pulses when opeartion of the MPG hand-wheel starts=MCM #241=100

Feed distance =MCM #242=100
Hand-wheel multiplication factor R222=100 (PLC setting)
Hand-wheel moves one block=100 pulses
When the hand-wheel moves one block, the feed distance on X-axis
= 100 x (100÷100)
=0.1 mm
= 100 μm.

Ex: The pulses when opeartion of the MPG hand-wheel starts=MCM #243=200
Feed distance=MCM #244=500
Hand-wheel multiplication factor R222=100 (PLC setting)
Hand-wheel moves one block=100 pulses
When the hand-wheel moves one block, the feed distance on Y axis
= 100 x (500÷200)
=0.250mm

= 250 μm 。

253. Setting X-axis as the rotating axis

254. Setting Y-axis as the rotating axis

255. Setting Z-axis as the rotating axis

256. Setting A-axis as the rotating axis
Format=□, Default=0
Setting = 0, linear axis.
Setting = 1, rotating axis.

259. Arc cutting error
Format:□□□□
Default＝1
Setting range: 1~32

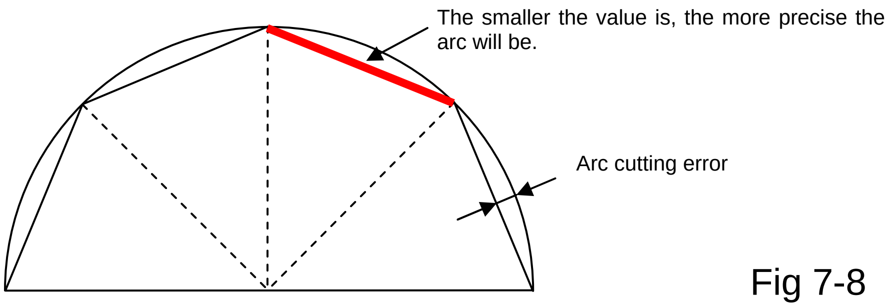


Fig 7-8

For arc cutting, the ideal cutting path is an arc, but actually the motor moves along the arc cord (a straight line). Therefore, the cutting error must be taken into account when an arc cutting is required.

The smaller the value is, the more precise the arc will be. The ideal value is 1.

However, sometimes the motor cannot operate if the value is not greater than 1, resulting in a greater cutting error.

260 : 281 Reserved

282 For system use

283 For system use

284 System ID

285. X-axis accel./decel. time setting

286. Y-axis accel./decel. time setting

287. Z-axis accel./decel. time setting

288. A-axis accel./decel. time setting
Format=□□□□ (Default=0), Unit: msec
A/D time (4~3072)

<u>MCM #291~542: Ball screw pitch error compensation</u>

<u>The machine origin is the reference point</u> for HUST H4CL-M ball screw pitch error compensation.

291. X-axis ball screw pitch error compensation setting

292. Y-axis ball screw pitch error compensation setting

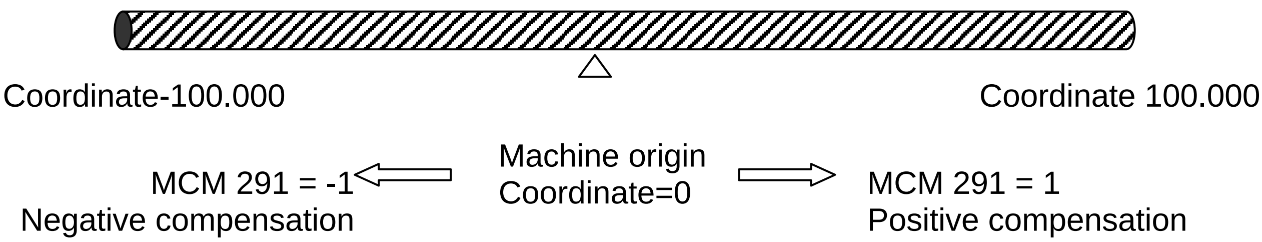
293. Z-axis ball screw pitch error compensation setting

294. A-axis ball screw pitch error compensation setting
Format=□ (Default=0)

Setting=0, compensation cancel
Setting=-1, negative compensation
Setting=1, positive compensation

X-axis	Y-axis	Z-axis	A-axis	Explanation
0	0	0	0	Compensation cancel
-1	-1	-1	-1	The tool is at machine origin; negative compensation
1	1	1	1	The tool is at machine origin; positive compensation

Ex.1: #291=-1, #291=1



297. X-axis segment length of pitch error compensation

298. Y-axis segment length of pitch error compensation

299. Z-axis segment length of pitch error compensation

300. A-axis segment length of pitch error compensation
Format=□.□□□ (Default=0), Unit=mm

Axis	MCM# for Segment Length	Segment Length	Max. Number of Segments
X	MCM#303~#342	20~480mm	40
Y	MCM#343~#382	20~480mm	40
Z	MCM#383~#422	20~480mm	40
A	MCM#423~#462	20~480mm	40

1. The segment length of compensation is the total length of the ball-screw divided by the number of segments.

Ex.: The total length of the X-axis ball screw is 1m(1000mm), which is to be divided into 10 compensation segments.

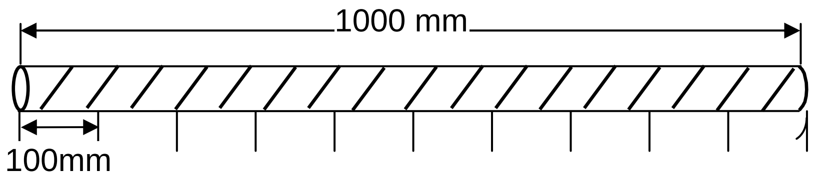


Fig 7-9

The average length of each segment is 100mm and the setting of MCM #297 X=100.000. The compensation of each segment is determined by MCM #303~342.

2. If the length of a compensation segment is less than 20 mm, it will be set to 20 mm.

3. HUST H4CL-M uses an average compensation approach and sets up 8 points for each segment length as a basis for compensation. The compensation of each point is 1/8 of the parameters in MCM #303~342. The value of the compensation is an integer with µm as its unit. The remainder of 1µm will be added into the next point for compensation.

Ex.: The X-axis is divided into 10 segments for compensation and the average length of each segment is 100mm. Compensation setting: MCM #303 = 0.026mm; the average compensation of each point = 0.026/8=0.00325mm. The compensation of eight points in the first segment is described in the following table:

Compensation point	Tool position mm	Avg. comp. of each point	Actual comp. of each point	Aggregated compensation
1	12.5	0.00325	0.003	0.003
2	25	0.00325	0.003	0.006
3	37.5	0.00325	0.003	0.009
4	50	0.00325	0.004	0.013
5	62.5	0.00325	0.003	0.016
6	75	0.00325	0.003	0.019
7	87.5	0.00325	0.003	0.022
8	100	0.00325	0.004	0.026

303~542. X, Y, Z, A axes 40 segments compensation
Format=□.□□□ (Default=0), Unit=mm
The maximum number of segments for each axis is 40.

The compensation value is incremental, either positive or negative. If the number of segments is less than 40, all other parameters must be set to zero.

Ex.: There are only 10 compensation segments. Therefore, compensation from #11 to #40 (#313 ~342 for X-axis, #353 ~382 for Y-axis, #393 ~422 for Z-axis, and #433~462 for A-axis) must be set to zero (0).

303~342. Segmentation compensation for X-axis ball screw pitch error (unit: mm)

MCM#	Default	Compensation
303	0	
304	0	
305	0	
306	0	
307	0	
308	0	
309	0	
310	0	
311	0	
312	0	
313	0	
314	0	
315	0	
316	0	
317	0	
318	0	
319	0	
320	0	
321	0	
322	0	
323	0	
324	0	
325	0	
326	0	
327	0	
328	0	
329	0	
330	0	
331	0	
332	0	
333	0	
334	0	
335	0	
336	0	
337	0	
338	0	
339	0	
340	0	
341	0	
342	0	

343~382. Segmentation compensation for Y-axis ball screw pitch error (unit: mm)

MCM#	Default	Compensation
343	0	
344	0	
345	0	
346	0	
347	0	
348	0	
349	0	
350	0	
351	0	
352	0	
353	0	
354	0	
355	0	
356	0	
357	0	
358	0	
359	0	
360	0	
361	0	
362	0	
363	0	
364	0	
365	0	
366	0	
367	0	
368	0	
369	0	
370	0	
371	0	
372	0	
373	0	
374	0	
375	0	
376	0	
377	0	
378	0	
379	0	
380	0	
381	0	
382	0	

383~422. Segmentation compensation for Z-axis ball screw pitch error (unit: mm)

MCM#	Default	Compensation
383	0	
384	0	
385	0	
386	0	
387	0	
388	0	
389	0	
390	0	
391	0	
392	0	
393	0	
394	0	
395	0	
396	0	
397	0	
398	0	
399	0	
400	0	
401	0	
402	0	
403	0	
404	0	
405	0	
406	0	
407	0	
408	0	
409	0	
410	0	
411	0	
412	0	
413	0	
414	0	
415	0	
416	0	
417	0	
418	0	
419	0	
420	0	
421	0	
422	0	

423~462. Segmentation compensation for A-axis ball screw pitch error (unit: mm)

MCM#	Default	Compensation
423	0	
424	0	
425	0	
426	0	
427	0	
428	0	
429	0	
430	0	
431	0	
432	0	
433	0	
434	0	
435	0	
436	0	
437	0	
438	0	
439	0	
440	0	
441	0	
442	0	
443	0	
444	0	
445	0	
446	0	
447	0	
448	0	
449	0	
450	0	
451	0	
452	0	
453	0	
454	0	
455	0	
456	0	
457	0	
458	0	
459	0	
460	0	
461	0	
462	0	

5 Connections

5.1 Connection System Introduction

This connection manual explains the electrical connections and system structure of the H4CL-M Series numerical control unit to ensure an adequate connection between the numerical unit and machine. This manual is intended for users with basic electrical knowledge.

Fig. 5-1 shows the command devices externally connected to the H4CL-M Series.

They are categorized as follows:

(1) The control unit can be connected to a PC via the RS232 interface.

(2) A PLC ladder application is built-in.

(3) The system can control both AC servo and stepper motors.

(4) The system provides 24 DI / DO 16 points. Also included are connections for the MPG, spindle encoder and skip sensor.

(5) In addition to the standard DI / DO, 12 DI / 16 DO points can be added via the serial port. **Total 36 DI / 32 DO points (Optional)**

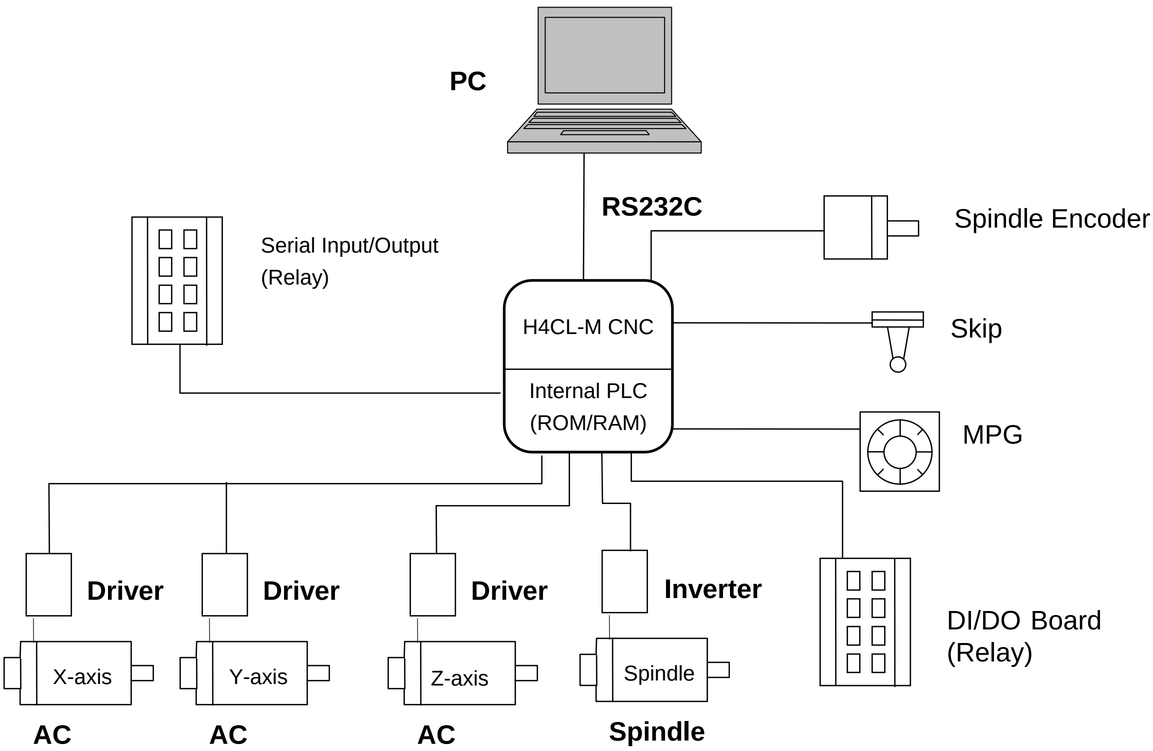


Fig. 5-1 Applications of the H4CL-M Series

5.2 System Installation

5.2.1 Operating Environment

The control unit used for the H4CL-M Series must be installed under the following conditions. Any failure to observe these conditions may lead to abnormal operation.

*	Ambient temperature	
	Operation	－0°C~ 45°C
	Storage or transportation	－-20°C ~ 55°C
*	Temperature variation	－Max. 1.1°C / min.
*	Humidity	
	Normal	－< 80% RH.
	Short time	－Max. > 95% RH.
*	Vibration limit	
	Operation	－max. 0.075 mm at 5 HZ.
*	Noise	
	Operation	－up to 2000 V / 0.1$\times 10^{-6}$ sec. per 0.01 second.

* Others

  Please consult us when the system is installed in an environment where the dust, coolant, or organic solution could be a problem.

5.2.2 Notes on the Control Unit Case Design

* A sealed case with dustproof design should be used for the control unit and MDI panel.
* The internal temperature of the case should not be 10°C higher than the ambient temperature.
* The cable inlet/outlet should be properly sealed.
* To avoid noise interference, the cable, each unit, AC power and CRT should be mounted at least more than 100mm apart. In a magnetic field environment, this distance should be increased to at least more than 300mm.
* Refer to the operator’s manual of the servo driver for the mounting information about the servo driver.

5.2.3 Case Thermal Design

The internal temperature of the case should not be 10°C higher than the ambient temperature. The main factors are the heat source and heat transfer area. For customers, the heat source is more uncontrollable than the heat transfer area. The allowable internal temperature rise of a case is shown as follows:

(1) With a cooling fan, the allowable temperature rise is 1°C /6 W/1 m^2.

(2) Without a cooling fan, the allowable temperature rise is 1°C /4 W/1 m^2.

This means that a case with a heat transfer area of 1 m^2 and a cooling fan should have an allowable internal temperature rise of 1°C if a heating unit of 6 W (4W without a cooling fan) is installed in the case. The heat transfer area of a case is acquired by subtracting its overall surface area from its area in contact with the floor.

Example 1: (With a cooling fan)

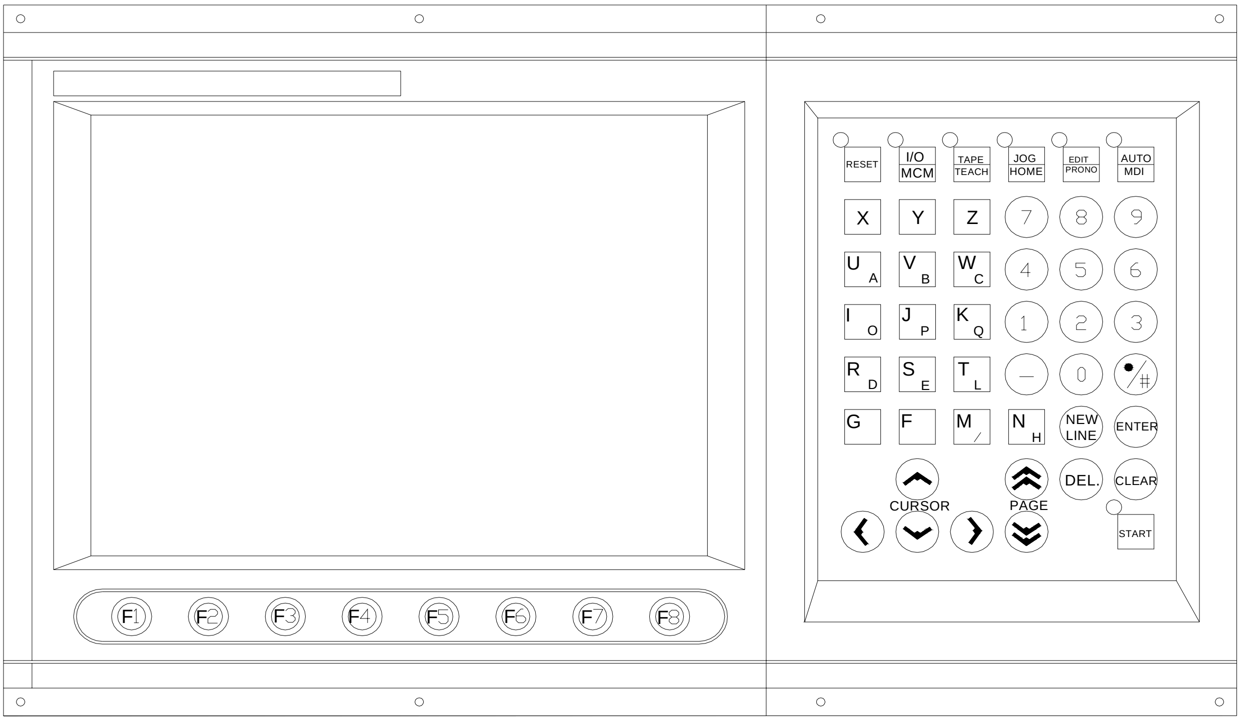
Heat Transfer Area = 2 square meters.
Allowable Internal Temperature Rise = 10°C
Maximum Allowable Heat Source = 6W x 2 x 10 = 120W.
Other cooling devices, such as the cooling fin, must be incorporated for calculation if the heat source is more than 120W.

Example 2: (Without a cooling fan)

Heat Transfer Area = 2 square meters.
Allowable Internal Temperature Rise = 10°C
Maximum Allowable Heat Source = 4W x 2 x 10 = 80W
Other cooling devices, such as the cooling fin, must be incorporated for calculation if the heat source is more than 80W.

5.2.4 H4CL-M External Dimensions

* H4CL-M Panel (Including the MDI Panel)



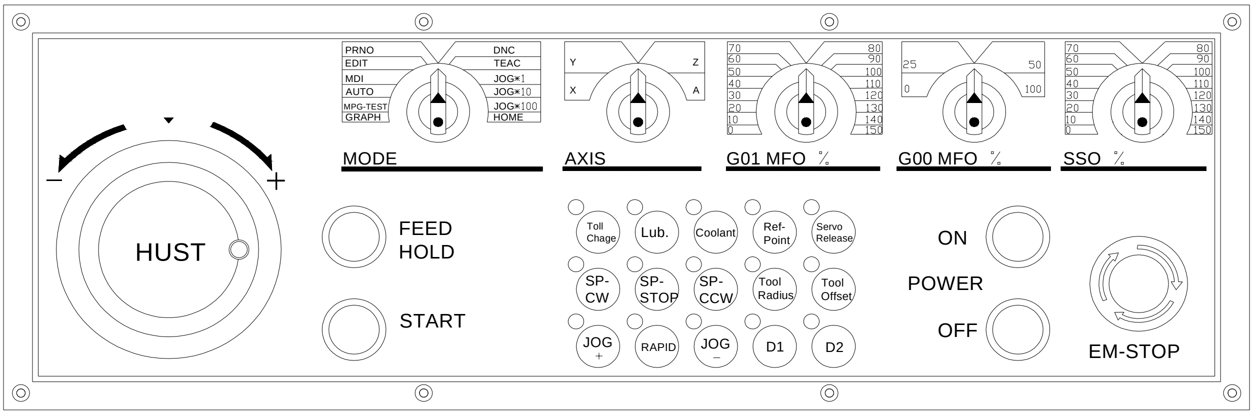


Fig. 5-2 H4CL-M Panel (Including the MDI Panel)

* H4CL-M CPU Main board Connectors (Rear View)

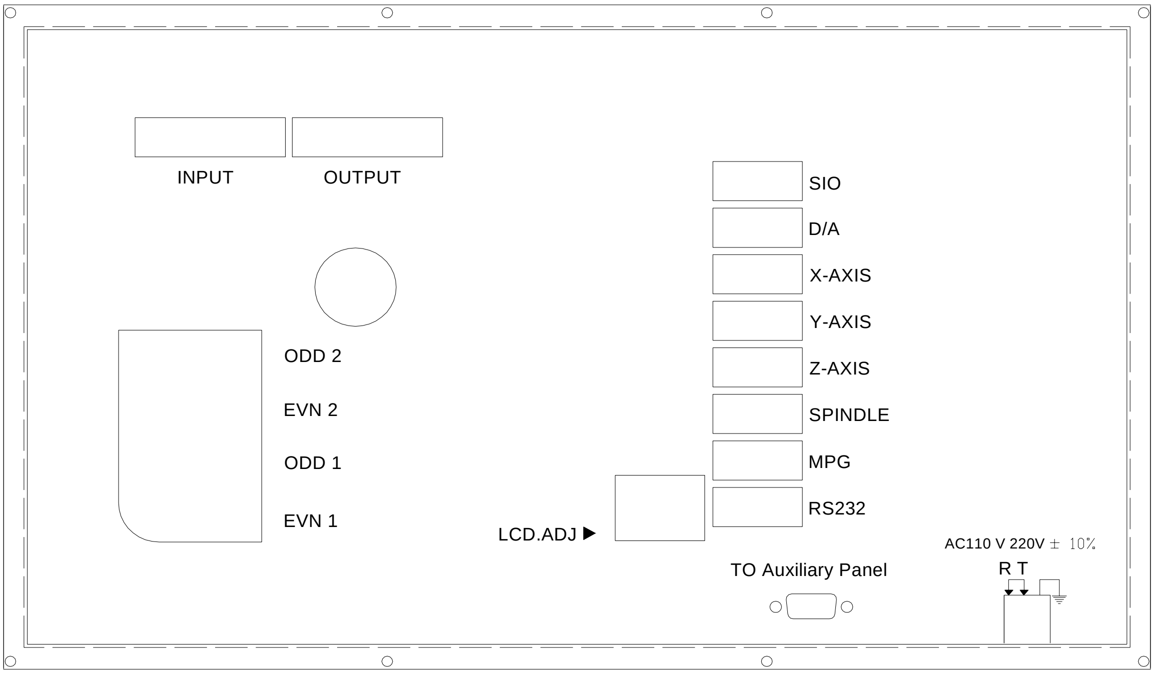


Fig. 5-3 H4CL-M CPU Main board Connectors

- **H4CL-M Series Case Dimensions**

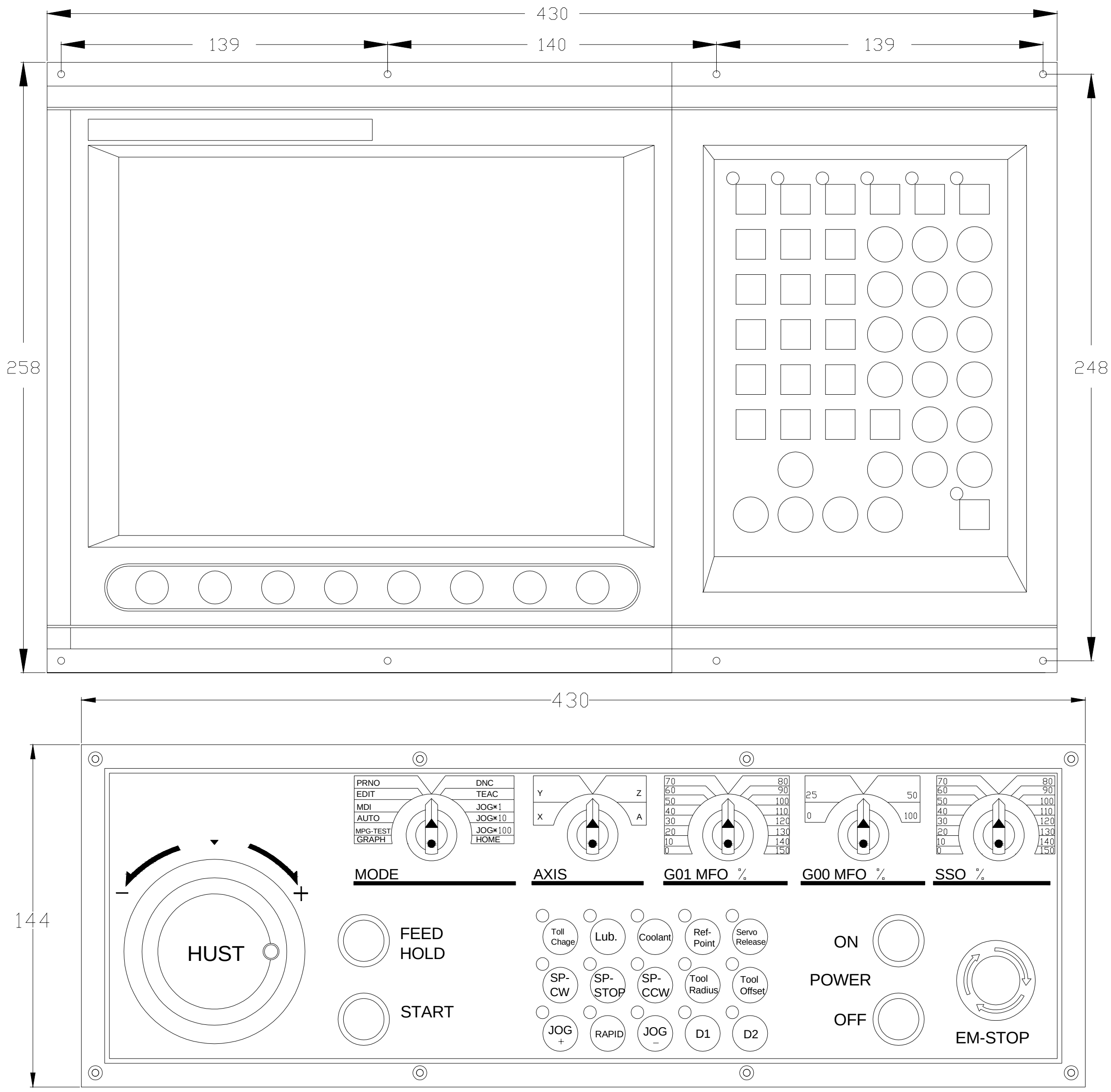


Fig. 5-4 H4CL-M Series Control Unit Case Dimensions

* **H4CL-M Series Control Unit Case Dimensions (Top View)**

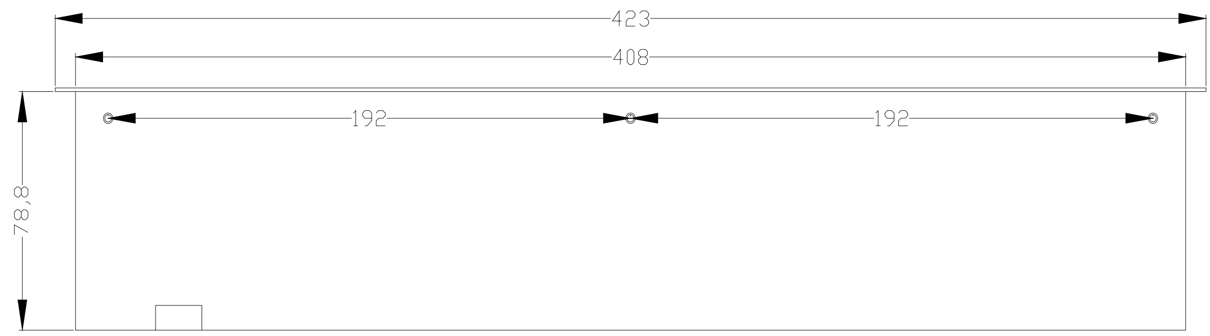


Fig. 5-5 H4CL-M Series Control Unit Case Dimensions (Top View)

* H4CL-M Series Cutout Dimensions

- Main Control Unit Cutout Diagram

Fig. 5-6 Control Unit Cutout Dimensions

- MDI Panel Cutout Dimensions

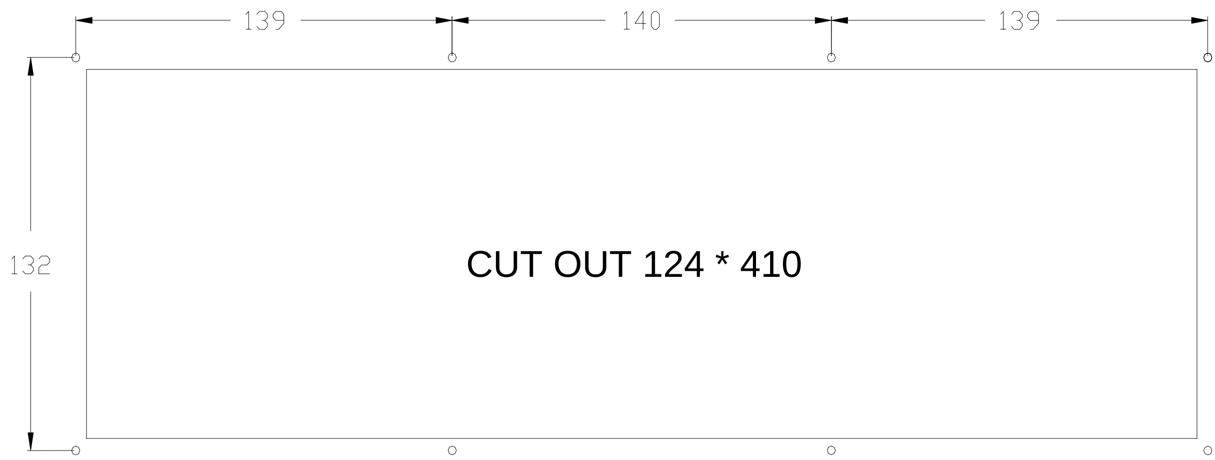


Fig. 5-7 MDI Panel Cutout Dimensions

5.3 Input/Output (I/O) Interface Connections

5.3.1 Input Board / Output Board (Terminal Block Type)

*** NPN standard input board – 24 IN.**

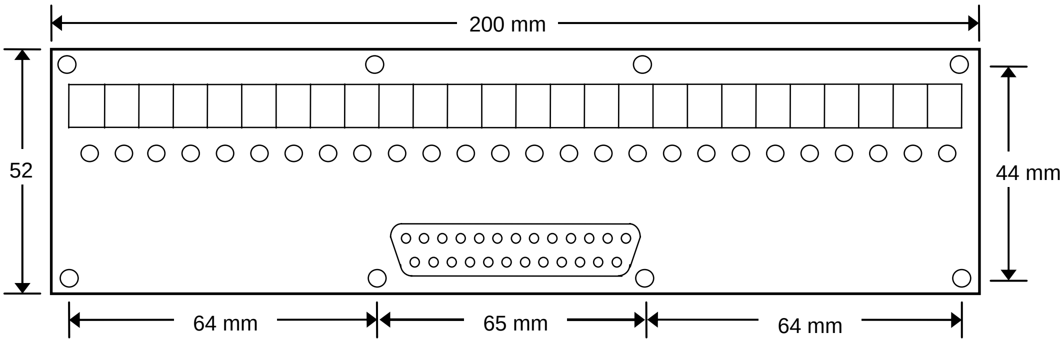


Fig. 5-8 NPN type 1 Input Board

*** NPN type standard output board – 16 Outputs.**

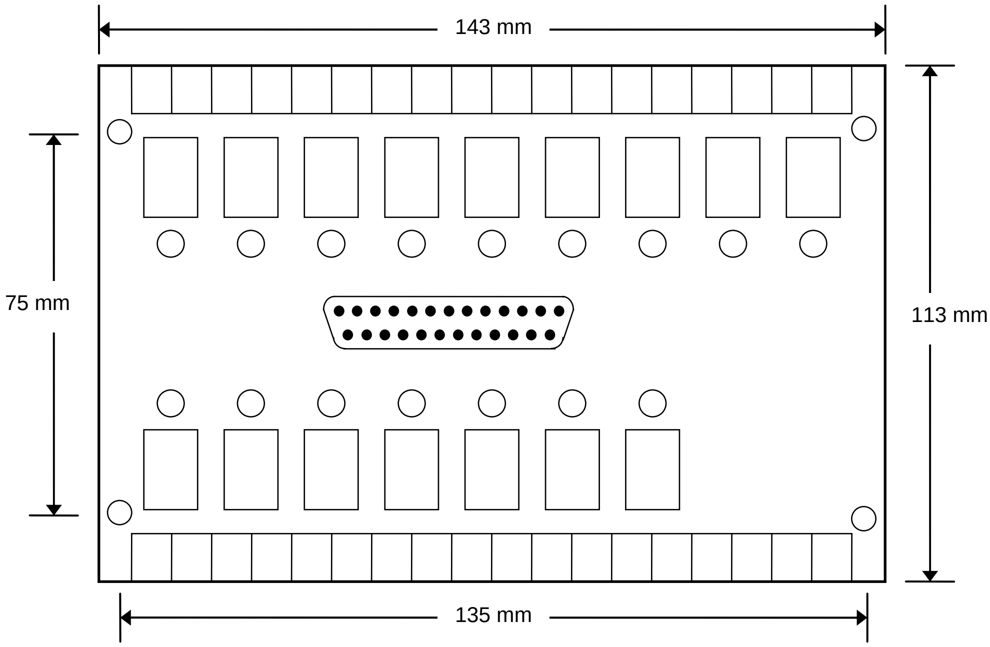


Fig. 5-9 NPN type 1 Output Board

5.3.2 Input Board / Output Board (CE Standard)

*** Input Board**

The standard NPN input board provides an interface of 24 input points. When the signal is correctly received, the corresponding indicator will “illuminate”, otherwise, check if your part program or connections are correct. Note: Voltages will exist at the +24V terminal of each input when the +24V power supply begins operation.

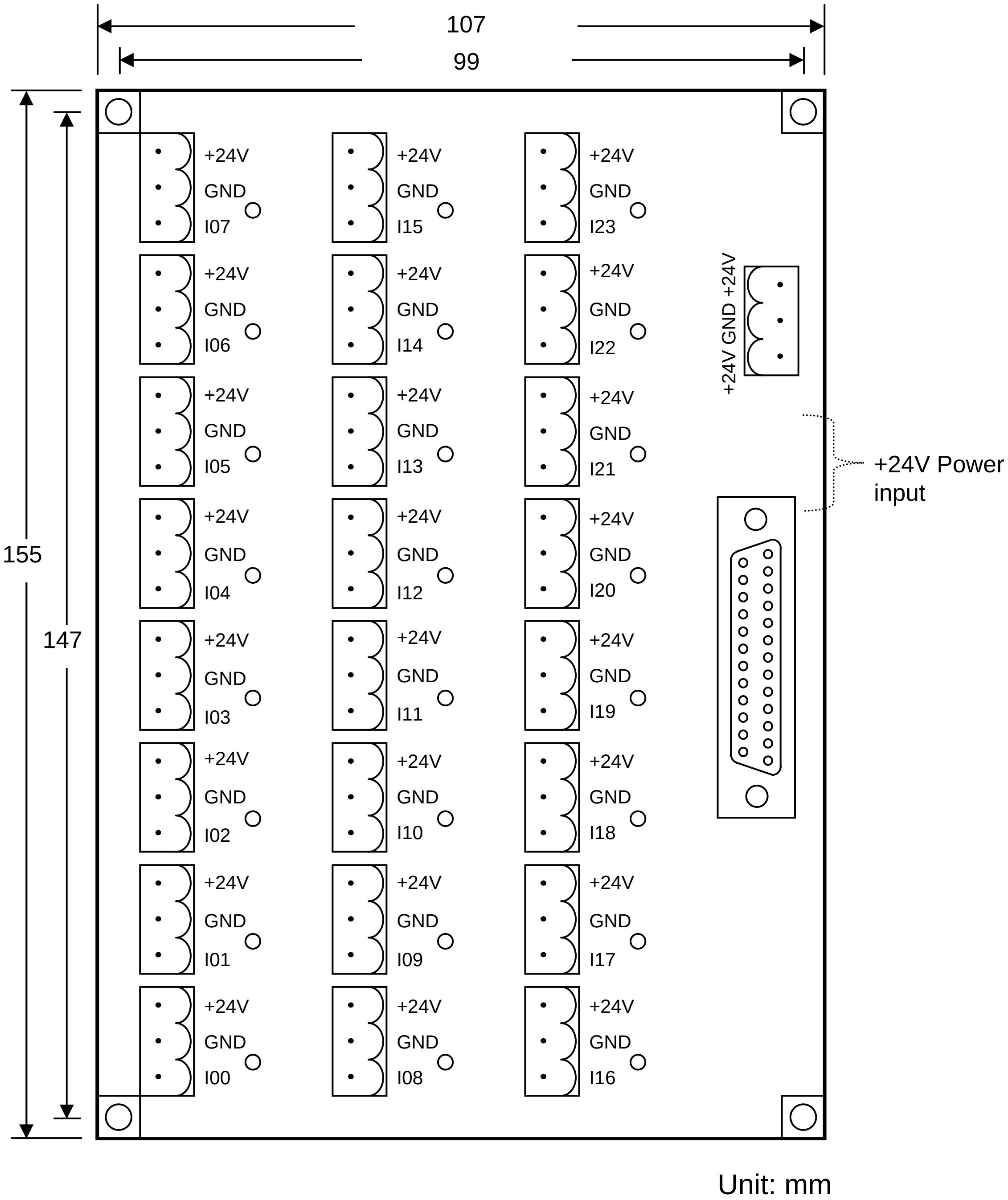
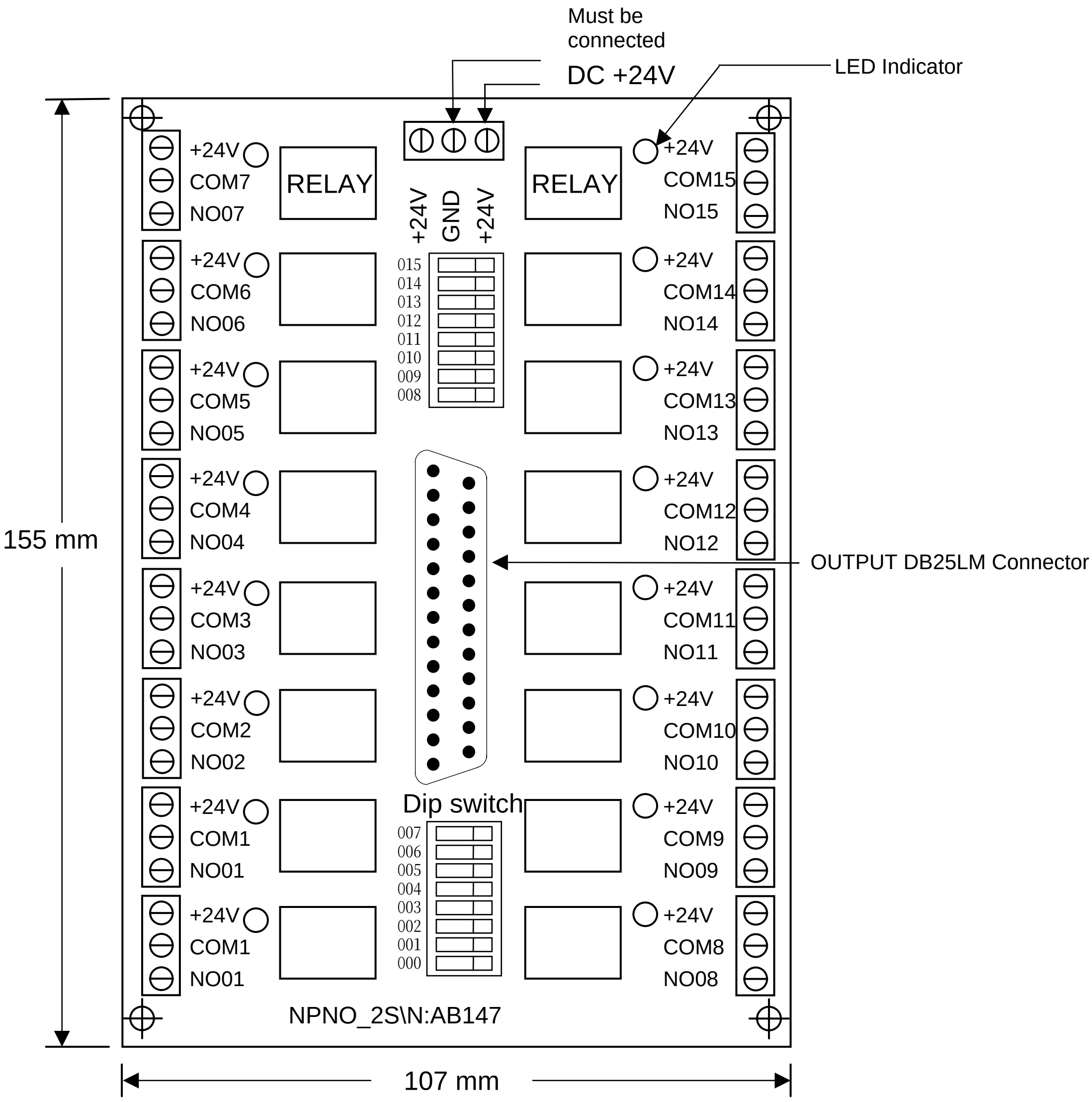


Fig. 5-10 Input Board

* Output Board

The standard NPN output board provides an interface of 16 output points. During output, the corresponding indicator will illuminate. If the DIP switch is set to “OFF”, the COM and NO are the general switch contact. If the DIP switch is set to “ON”, the COM is directly connected to the 24V ground. Therefore, when the solenoid valve and DC motor at the NO is driven via 24V signal ground; the DIP should only be set to “ON”. In this case, it is not necessary to connect the COM with the 24V ground.



Note: NO = Normal open contact.

COM = Common contact.

Fig. 5-11 Output Board

5.3.3 I/O Interface Connector Pin Assignment

I/O connections and pin assignments are described in this section.

The I/O connector designation and pin assignments are shown in Table 5-1 and Fig. 5-12. Note that there are 24 input points and 16 output points.

Table 5-1 I/O Connector Designation and Pin Assignment

HUST CNC	Connector Designation	I/O Pin Assignment
H4CL-M	Output	O 000 ~ O 015
	Input	I000 ~ I023

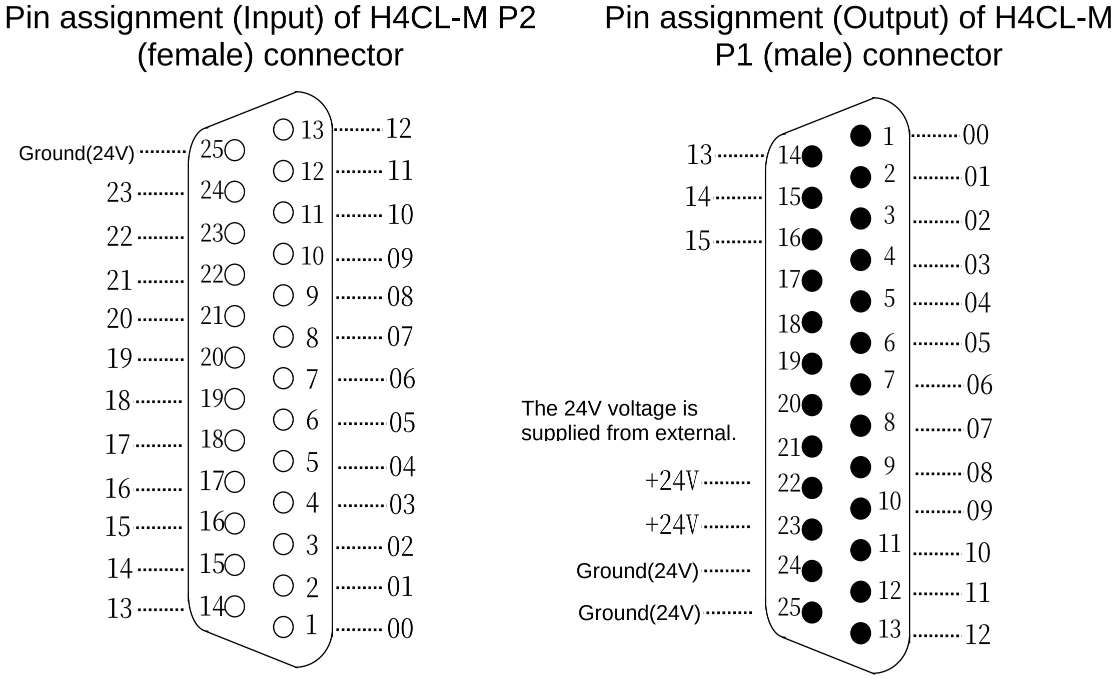


Fig. 5-12 I/O Connector Pin Assignment (NPN-type)

I/O signals are directly connected to the input board and I/O points on the relay board when these boards are used. I/O signals will be connected to output and input connectors when the input board and relay board are not used. The output points of the H4CL-M Series are transistor circuit of the open collector.

The H4CL-M Series should be fed with the +24V power supply from the output board. The +24 power supply is not provided.

*** The +24V power supply should be used for the output terminal when no output points are used. The input terminal will not receive signals when the +24V power supply is not used for the input terminal.**

5.3.4 Input Signals

The input signal is transmitted to the control unit from an external machine through the button, limit switch, relay board contact, or proximity switch. The H4CL has 24 input points.

*** Input Signal Specifications:**

Input voltage: 0 V.

Input current: 8 mA.

*** Input Signal Connection Diagram (direct input to the control unit)**

The input signal is directly connected to the input points (the input connectors of the H4CL-M) of the numerical control unit when the input board is not used.

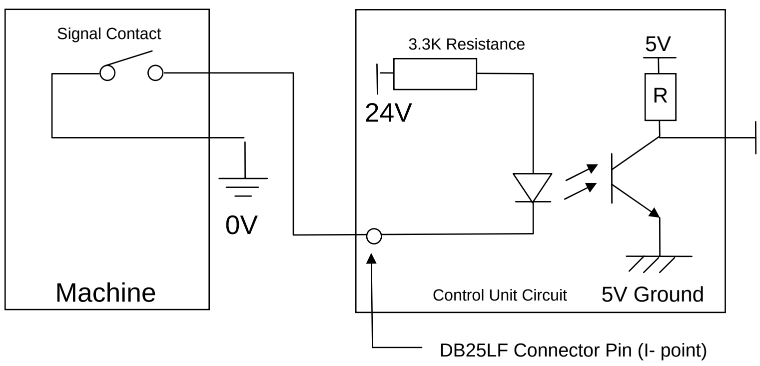


Fig. 5-13 Input Signal Connection (direct input to the control unit)

* **Input Signal Connection Diagram (input to the control unit via the input board)**

Refer to Fig. 5-7 and 5-9 for the input pin assignment if the input board is used.

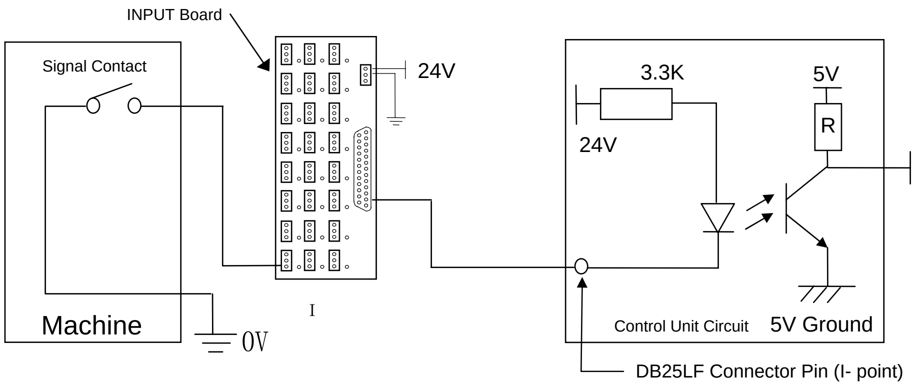


Fig. 5-14 Input Signal Connection (input to the control unit via the input board)

5.3.5 Output Signals

The output signal is transmitted to the external machine from the control unit. The output circuit of the HC4L-M is a transistor circuit that is used to drive the relay or LED of the machine.

* **Output Signal Specification**

(a) Each contact on the output relay board can withstand up to AC 250V and 1A.

(b) Each contact of the H4CL-N transistor circuit supplies up to 24V and 100mA when the output relay board is not used.

* **Output Signal Connection (direct input to the machine from the control unit)**

The output signal will be directly connected to the output points (the output connectors of the H4CL-M) of the numerical control unit when the output relay board is not used

When the H4CL-M control unit is directly connected to an inductive load, such as a relay on the machine, the inductive load should be connected to a spark killer in parallel that should be as close to the load as possible. (See. Fig 5-15)

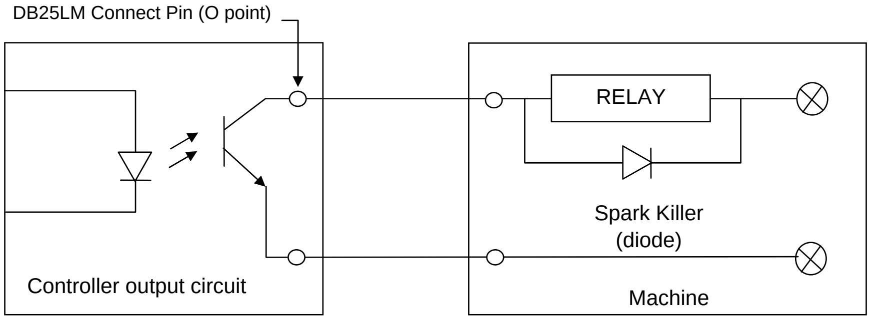


Fig. 5-15 Output Line (without the output relay board)

* **Output Signal Connection (the signal is transmitted to the machine via the output relay board)**

Refer to Fig. 5-8 and 5-10 for the output pin assignment if the output board is used.

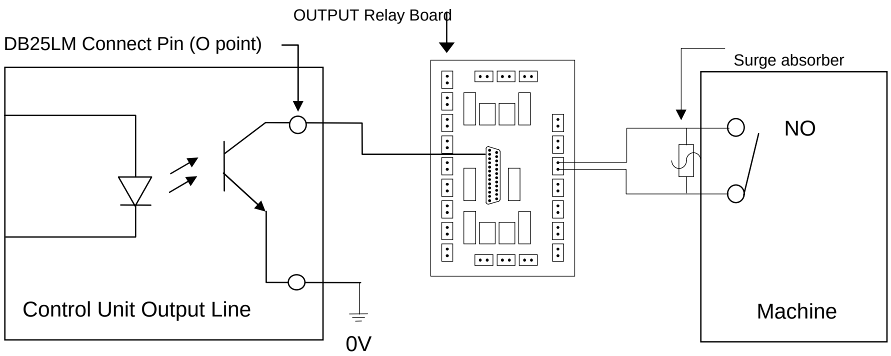


Fig. 5-16 Output Line (to the machine via the output relay board)

5.4 Connection Diagram

5.4.1 Connector Type

The connector types on the back of the H4CL-M control unit are listed below. Each connector symbol is followed by a letter of either M (for male) or F (for female).

DB9L ： 9-pin connector.

DB25L ： 25-pin connector.

TBxx ： Terminal connector and its pin is represented with “XX”.

5.4.2 Connector Designation

The H4CL-M connector designation in marked on the back of the control unit case and their corresponding types are listed below:

Connector Name	Connector Designation	Connector Type
Standard Output Interface	Output	DB25LM (Male)
Standard Input Interface	Input	DB25LF (Female)
D/A Spindle Analog Output	D / A	DB15LF (Female)
X-axis Servo	X-axis	DB9LF (Female)
Y-axis Servo	Y-axis	DB9LF (Female)
Z-axis Servo	Z-axis	DB9LF (Female)
Spindle	Spindle	DB9LF (Female)
MPG	MPG	DB9LM (Male)
I/O (expandable)	SIO	DB15LM (Male)
RS232C Interface Connector	RS232	DB9LF (Female)

Table 5-2 H4CL-M Connector Designation and Type

5.4.3 H4CL-M Connection (Y-shaped terminal)

* H4CL-M Main Connection Diagram

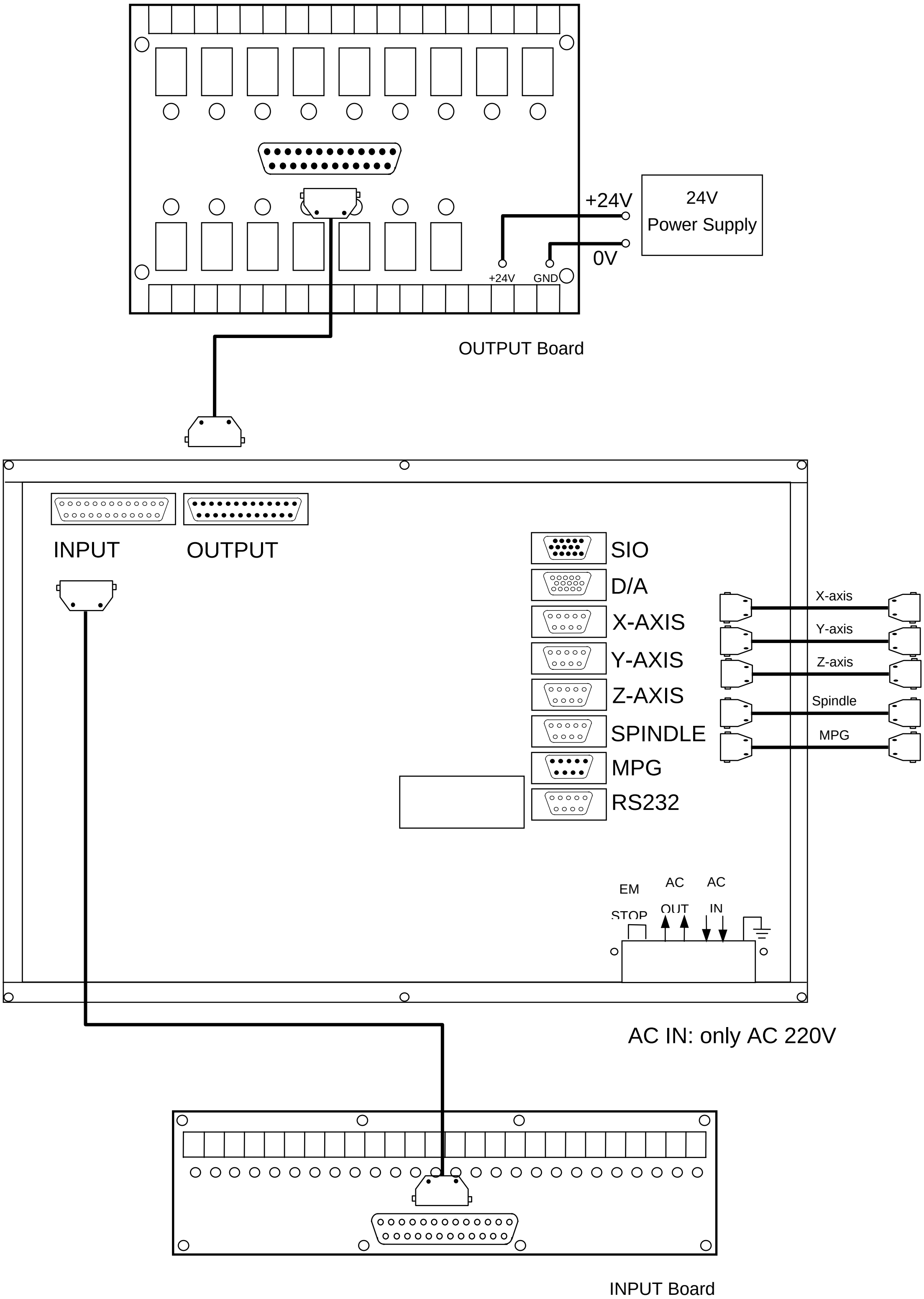


Fig. 5-17

* Emergency-Stop Line-1

It is recommended to connect as Fig. 5-18. In doing so, the software and hardware will be controlled in series control and the user can press the emergency button to turn off the Servo-On even if any abnormity is found in the software.

(One end of the idle contact used for output is connected to the ground through emergency stop button and limit switch. The other end of the idle contact is connected to the relay to control all SERVO ON contacts of the servo motor.)

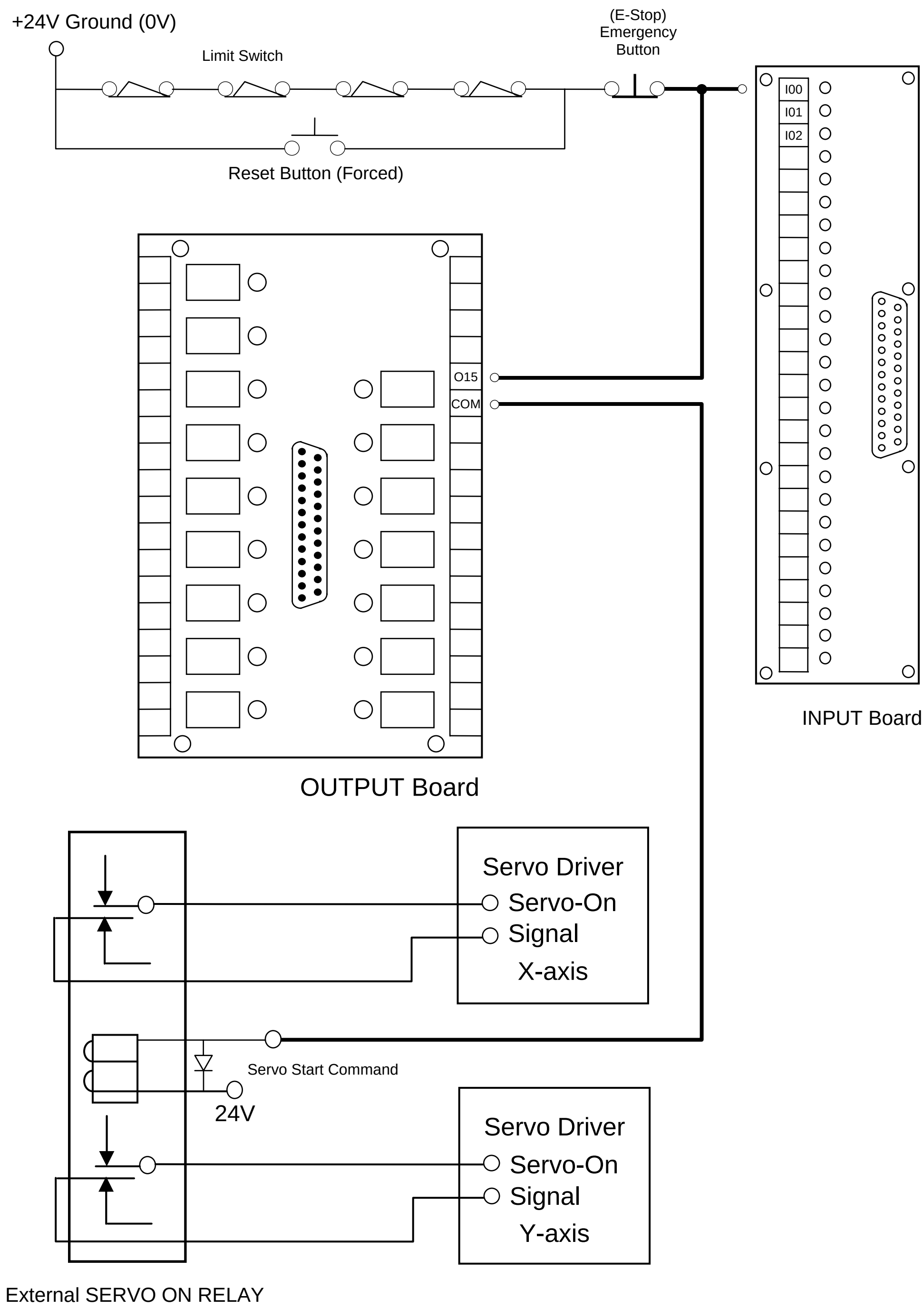


Fig. 5-18

* Emergency-Stop Line-2

Fig.5-19 is a simplified connection diagram.

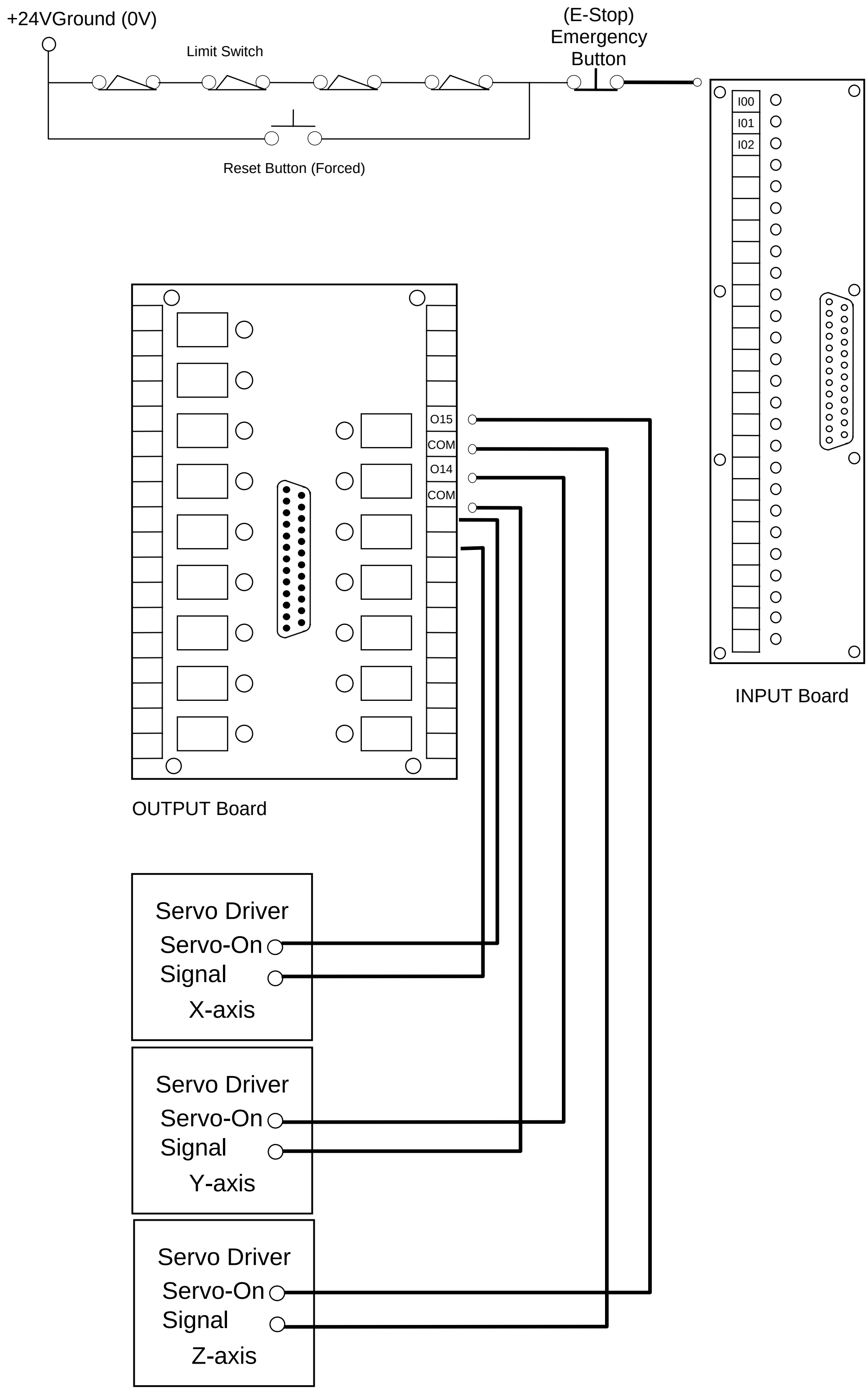
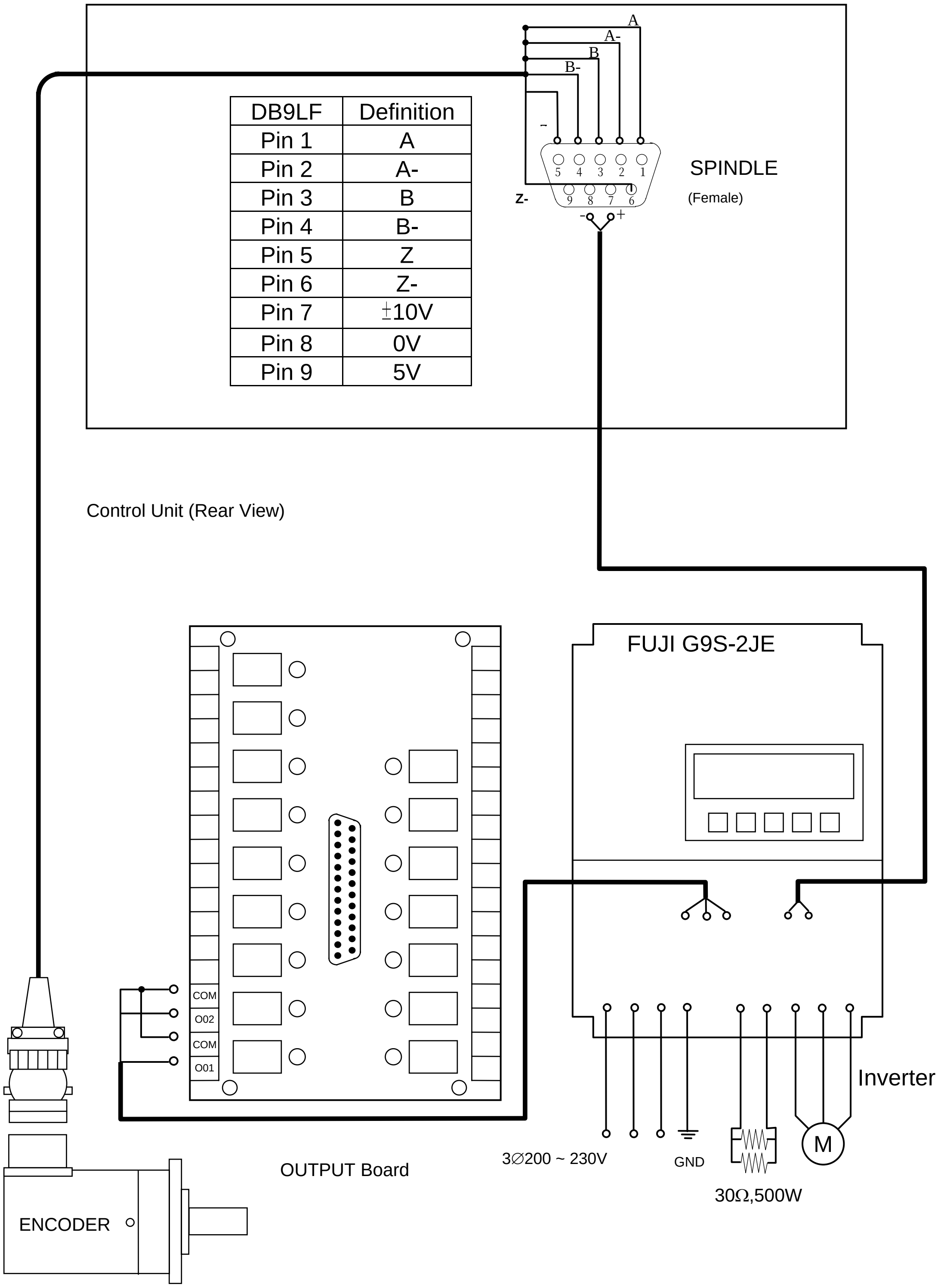


Fig. 5-19

*   H4CL-M Spindle Connection



Note: It is recommended to connect 24V for the contacts on the output board.

If a part (such as oil and water pumps, and motor) requiring 110V or 220V voltage needs to be powered on, an additional relay is powered via the output board.

Fig. 5-20

5.4.4 H4CL-M Connection (CE Standard)

* H4CL-M Main Connection Diagram

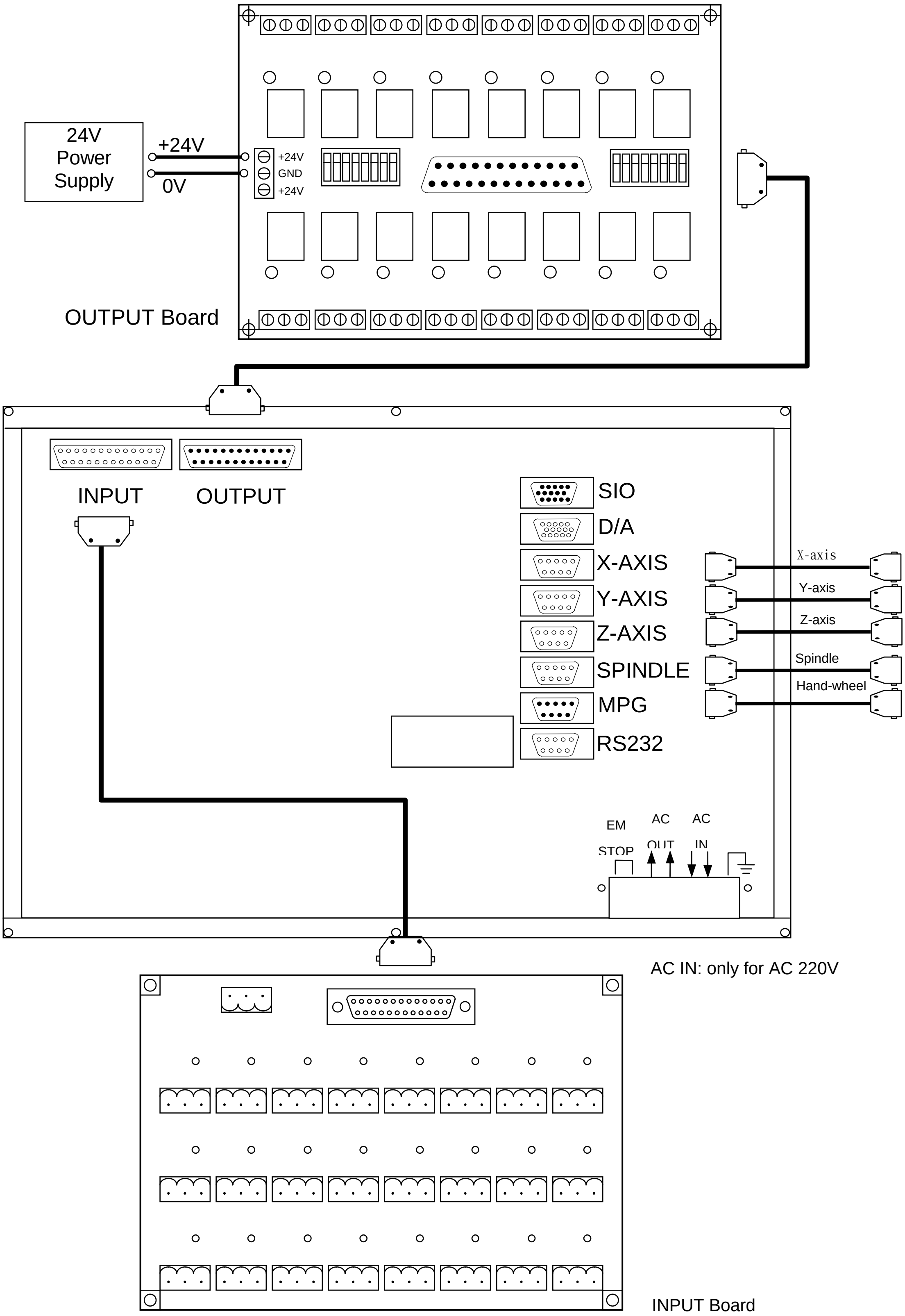


Fig. 5-21

*** Emergency-Stop Line-1**

It is recommended to connect as Fig. 5-22. When the software and hardware is connected in series, press the emergency button to turn off the servo even if the software is abnormal.

(One end of the idle contact used for output is connected to the ground through emergency stop button and limit switch and its other end is connected to the relay to control all SERVO ON contacts of the servo motor.)

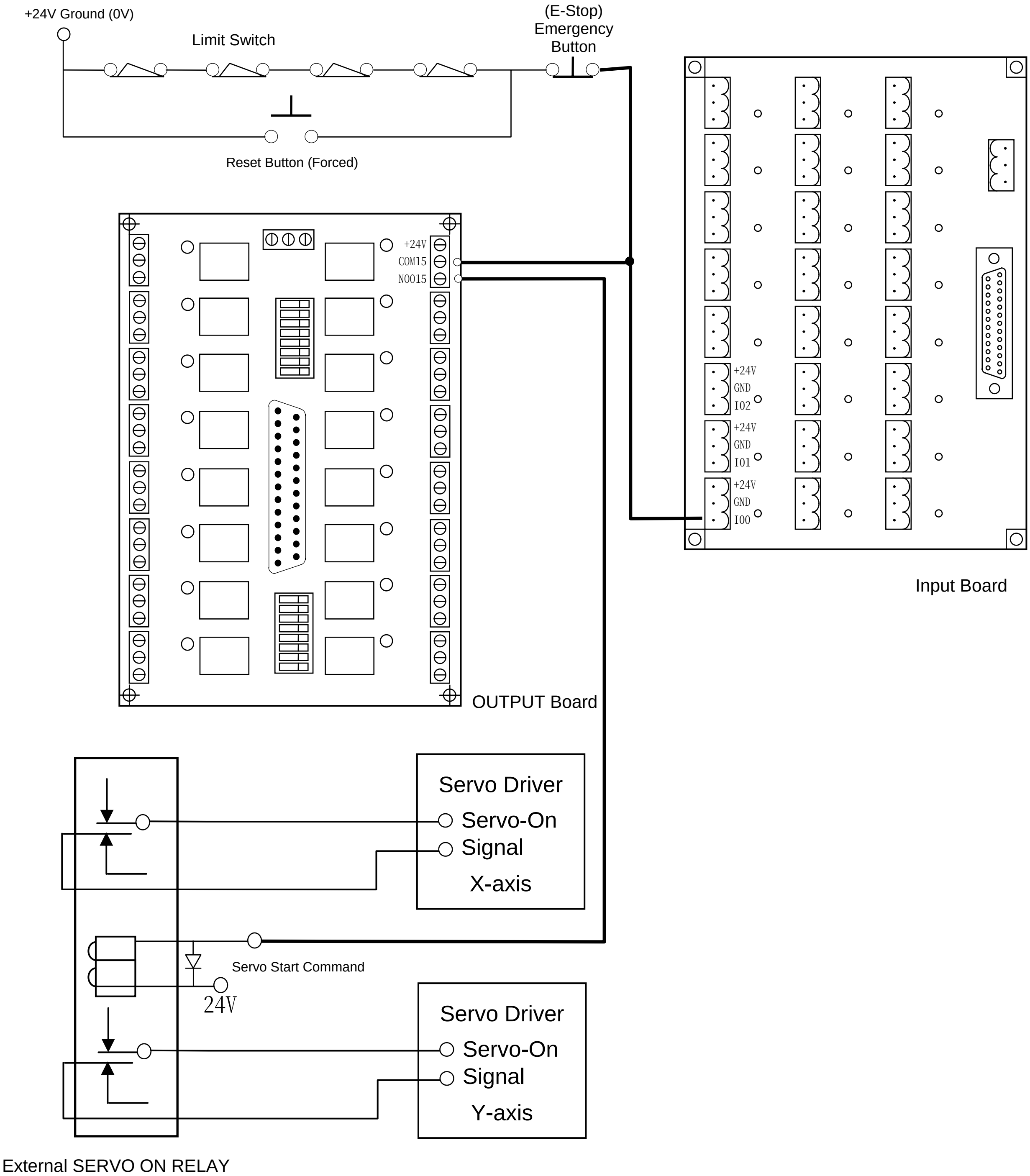


Fig. 5-22

* Emergency-Stop Line-2

Fig.5-23 is a simplified connection diagram.

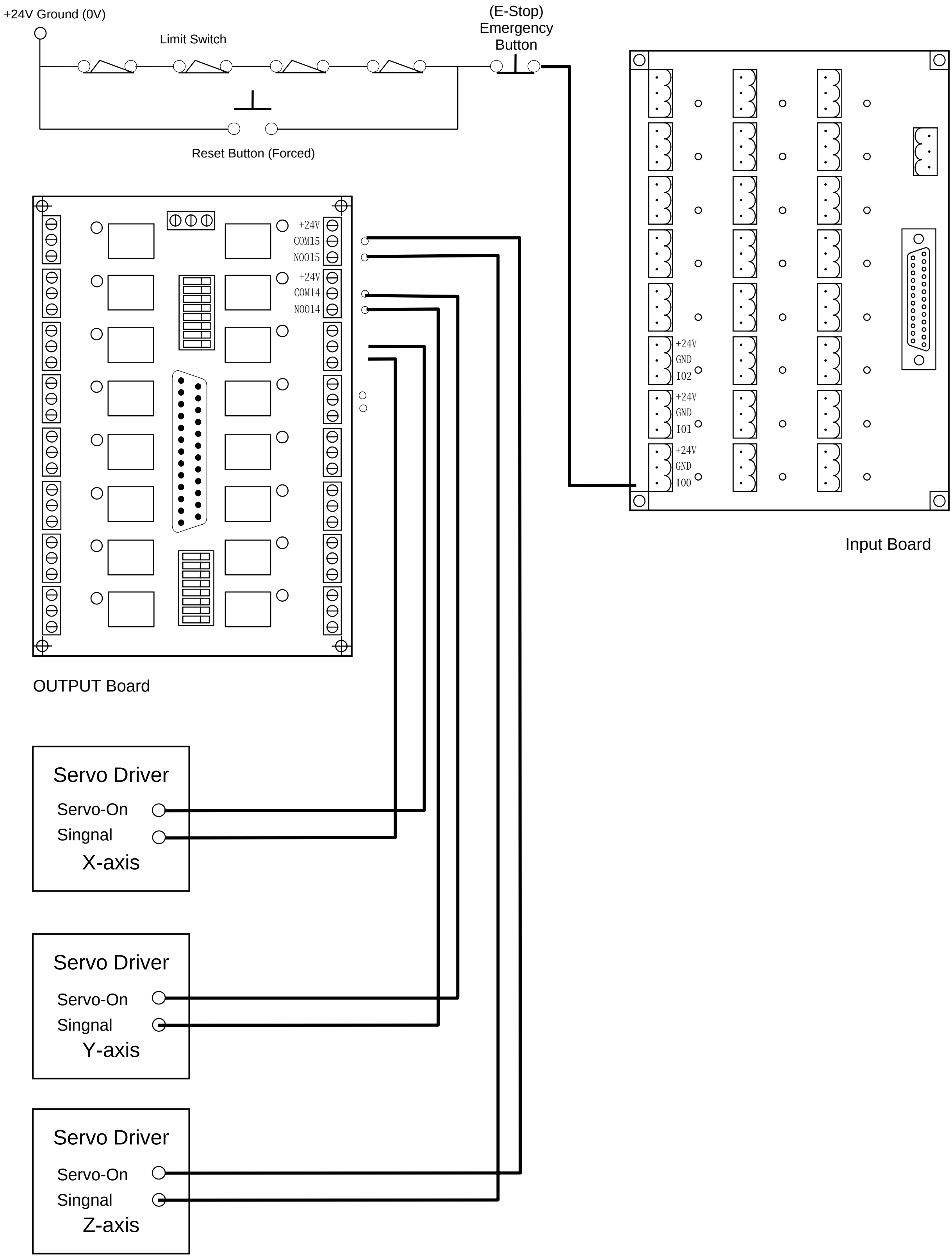


Fig. 5-23

* Servo Motor Connection Diagram (The MITSUBISHI J2S motor is used as an example.)

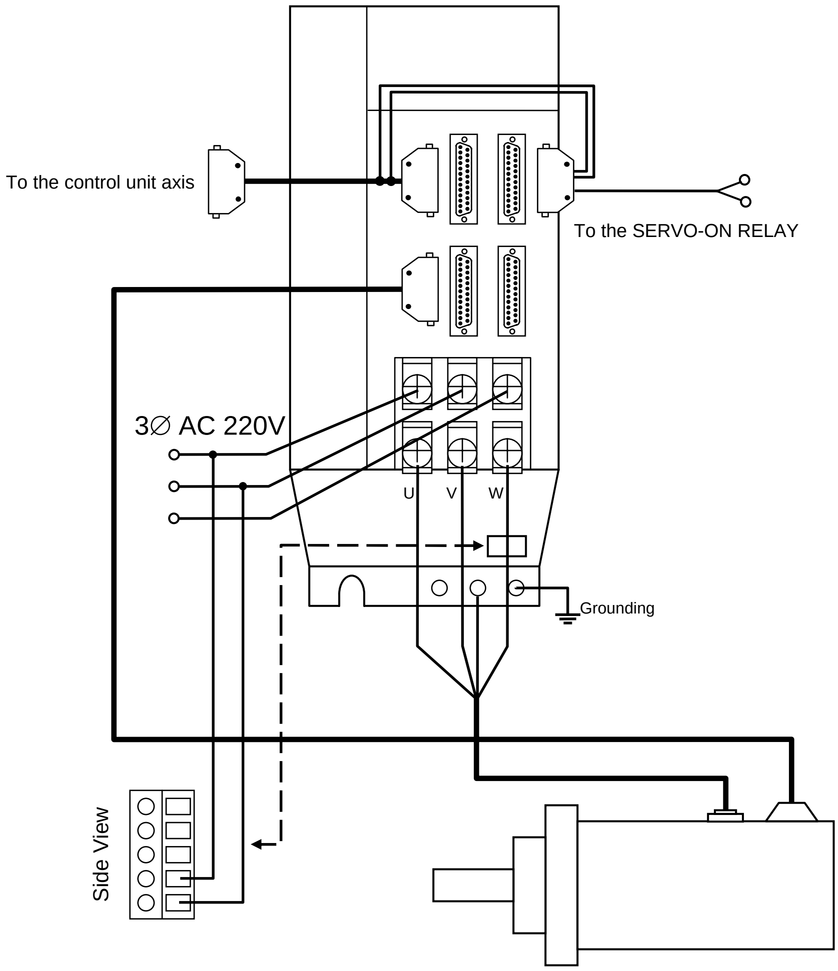
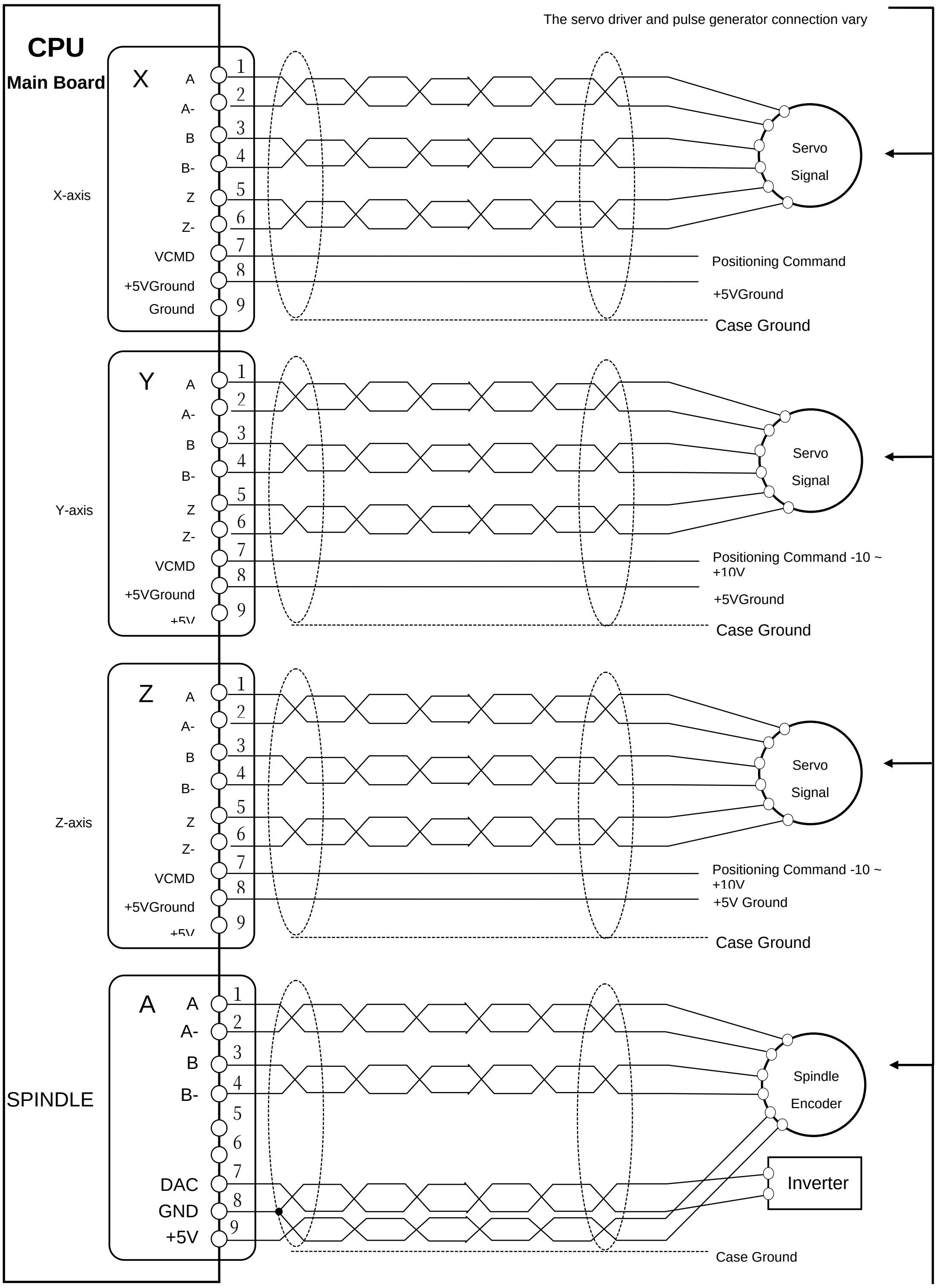


Fig. 5-24

5.4.5 Connection Method for Servo Driver & Pulse Generator

The servo driver is connected to the connectors of the X-, Y-, and Z-axis, and the spindle encoder and inverter to D/A. As shown in Fig. 5-25, the pulse generator is connected to the MPG. The servo driver and pulse generator connection vary depending on their brands. Refer to their operation manuals for details.



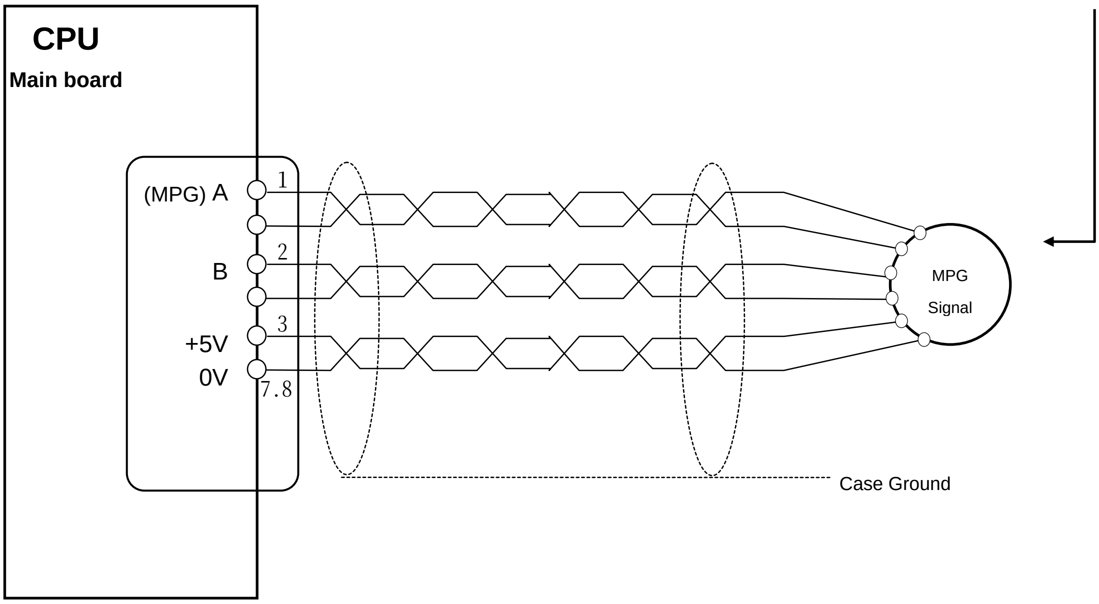


Fig. 5-25 Axes, Spindle, and MPG connection

5.4.6 AC Power System Connection

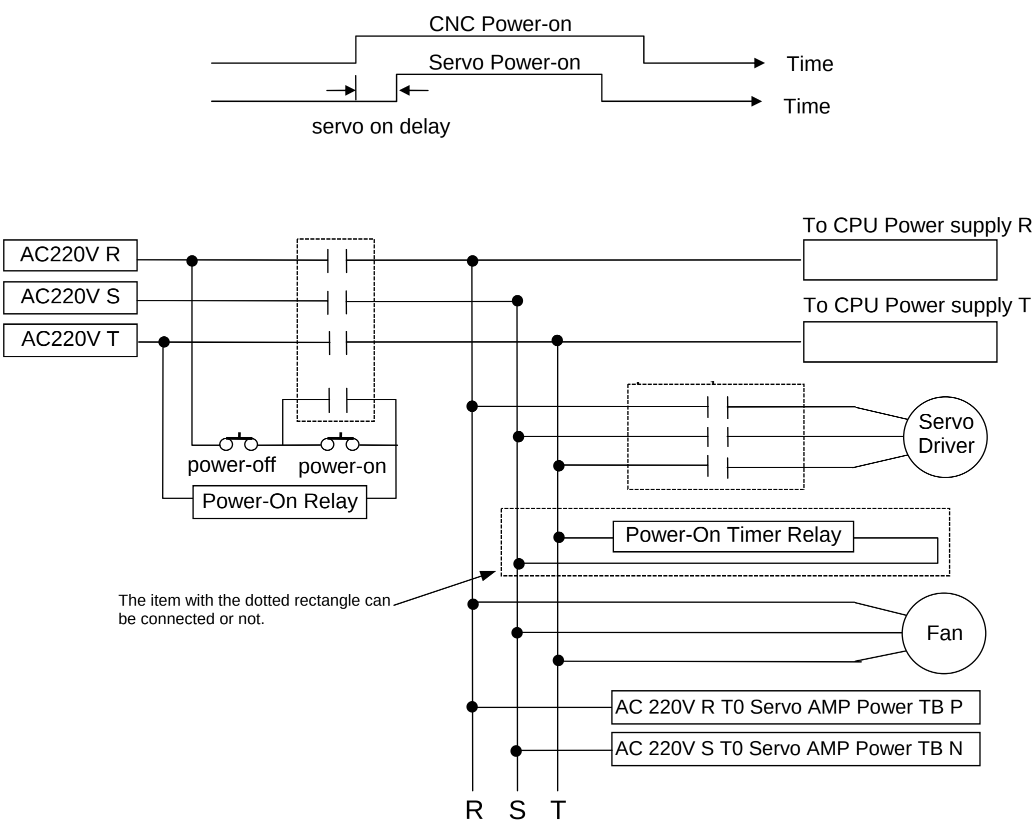


Fig. 5-26 System AC Power Connection

5.4.7 MPG Connection

If the tool moves in the opposite direction marked by the MPG, please exchange the signal line A and B in the MPG.

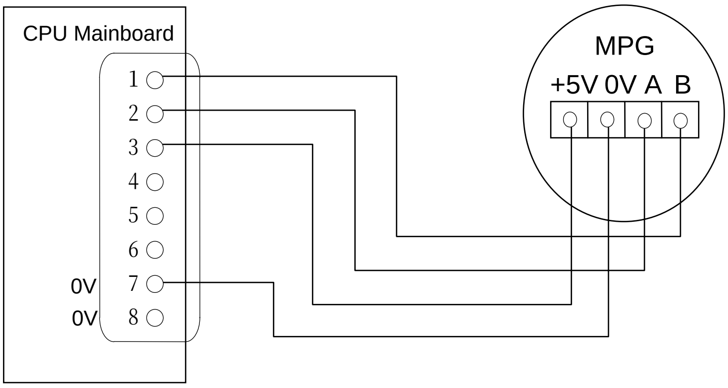


Fig. 5-27 MPG Connection

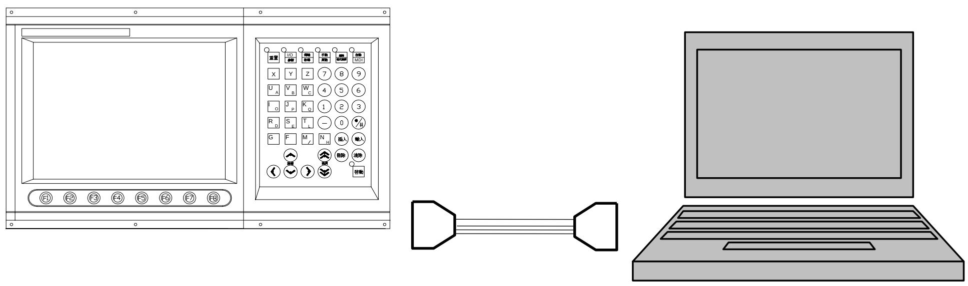
5.4.8 RS232 Connector Pin Assignment and Connection

Fig. 28 shows the connection method for the H4CL-M control unit and PC. Observe the following when connecting:

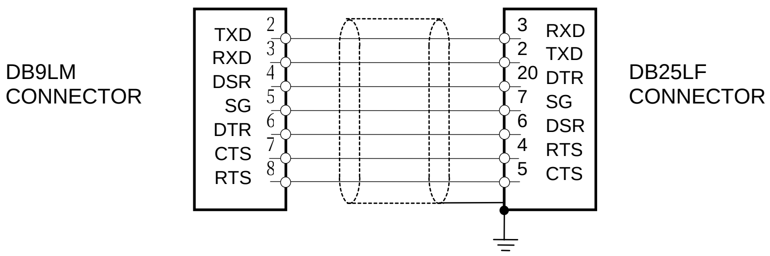
1. The connection between the RS232 port and PC should not be more than 15 meters.

2. If the connection is established in an environment where there is a source of noise, such as the electric discharge machining or welding machine, establish the connection with **<u>twisted cables</u>** or avoid installing in such an environment. **<u>The control unit and PC should not share the same power outlet with the electric discharge machining or welding machine.</u>**

3. The interface voltage of the PC should range from 10 to 15V.



HUST Control Unit H4CL-M → RS232 → PC COM2



HUST Control Unit H4CL-M → RS232 → PC COM1

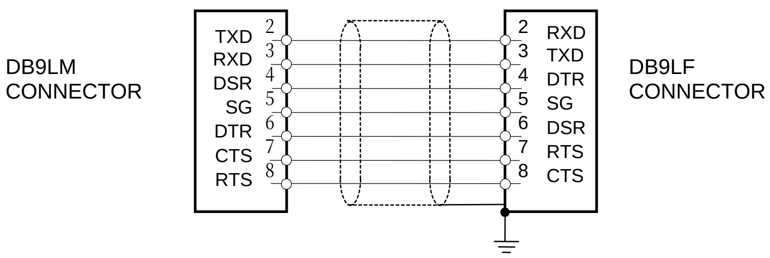


Fig. 5-28 RS232 Connection

6 Error Message Explanations

When an error occurs during the execution of the program, the error message is displayed on the LCD of the H4CL-M Series controller. (Fig 6-1). Possible error messages of the H4CL-M Series controller and their solutions are described in the following:

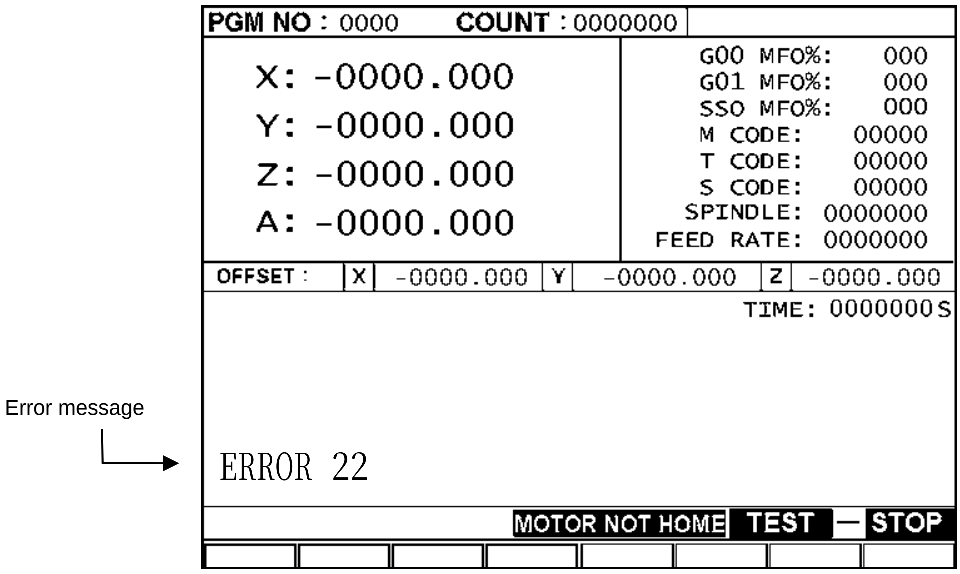


Fig. 6-1

* ERROR-01 MCM DATA ERROR OR BATTERY FAIL

Message:

MCM parameter settings are incorrect or the memory battery has low power.

Recommended remedy:

1. Check that the MCM parameters are correct, or double-click AUTO/MDI to enter MDI mode. Execute G10 P1000 command to clear incorrect parameters and reset the program.

2. If the controller has not been switched on for more than one year, the data in the memory will be lost. Where the controller generates a (BT1) low power message, turn on the contoller for 4 hours to recharge the battery. If the problem persists after recharging, contact your dealer to replace the battery.)

* ERROR-02 SERVO ALARM, PLEASE HOME AGAIN

Message:

An error occured in the servo loop system. The possible error status includes:

1. The control unit sends commands too quickly, and the servomotor cannot respond in time.
2. The control unit does not receive feedback.

Recommended remedy:

1. Check the F value is set appropriately.
2. Check the resolution and maximum feed-rate are correct. Also check the MCM parameter settings.
3. Make sure that the machine and motor function appropriately and the wiring is correct.

*** <u>ERROR-03</u> COUNTER LIMIT**

Message:

The counter of M02, M30, M99 exceed the settings of MCM #170

Trobleshooting:

1. Clear the MCM #169 counter value to 0 and press Reset to remove the error.
2. Alternatively, execute <u>G10 P201</u> command in the AUTO or MDI mode to clear the MCM #169 counter value to 0 and press Reset to remove the error.

*** <u>ERROR-04</u> G60 MISSING REPEAT NO. (LA)**

Message:

No repeat command(L) is contained in G60 rotating axis command. G60 is not used in the G54 coordinate system.

Recommended remedy:

Check the program and change the L value of G60 block.

*** <u>ERROR-08</u> EXCEED 64 CHARACTERS FOR ONE BLOCK**

Message:

A block in the program contains more than 64 characters.

Recommended remedy:

Check the program and make sure that no block contains more than 64 characters.

*** ERROR-10 RS232 ERROR**

Message:

An error in RS232C communication signal of the controller.

Recommended remedy

1. Check and make sure that the baud rate of the controller (i.e. MCM #168) is same as the baud rate of the computer or MMI.

2. Check the cable connection between the controller and the computer or MMI.

*** ERROR-11 PROGRAM MEMORY ERROR**

Message:

An error in program memory. The battery may have no power or the data exceeds the memory capacity.

Recommended remedy:

Enter the “MDI” mode and execute “G10 P2001” command. Clear all program data and check the battery. If the controller generates (BT1) low power message, turn on the controller to recharge the battery (if the controller has not been switched on for more than one year, the data in the memory will be lost.)

*** ERROR-13 ERROR G CODE COMMAND**

Message:

An incorrect G-code exists in the program data of the H4CL-M Series controller and cannot be accepted.

Recommended remedy:

Check the program and make sure the G-code is correct.

*** ERROR-14 X-AXIS OVER TRAVEL**

Message:

The X-axis tool moves beyond the pre-set hardware over-travel limit.

Recommended remedy:

Press the "Forced Homing" key and the lamp on the left-top corner of the key illuminates. Use the JOG (single step) function to return the axial movement from the limit to within normal range.

***   <u>ERROR-14</u> Y-AXIS OVER TRAVEL**

Message:

The Y-axis tool moves beyond the pre-set hardware over-travel limit.

Recommended remedy:

Press the "Forced Homing" key and the lamp on the left-top corner of the key illuminates. Use the JOG (single step) function to return the axial movement from the limit to normal range.

*   **<u>ERROR-14</u> Z-AXIS OVER TRAVEL**

Message:

The Z-axis tool moves beyond the pre-set hardware over-travel limit.

Recommended remedy:

Press the "Forced Homing" key and the lamp on the left-top corner of the key lights up". Use the JOG (single step) function to return the axial movement from the limit to within normal range.

***   <u>ERROR-18</u> END OF FILE NOT FOUND**

Message:

Program end error (M02, M30 missing).

Recommended remedy:

Check the end of the program and add an M02 or M30 block.

***   <u>ERROR-20</u> SOFTWARE OVER-TRAVEL**

Message:

The program runs beyond the software over-travel limit.

Recommended remedy:

Check the program or reset the MCM #171~182 software over-travel limit.

***   <u>ERROR-22</u> EM-STOP, HOME AGAIN**

Message:

The controller is now in the emergency stop state.

Recommended remedy:

After the cause of the emergency is removed, restore the emergency stop button and press “reset”.

***　　ERROR-24 M98 EXCEED 8 LEVEL**

Message:

Subprogram calls exceeds 8 levels.

Recommended remedy:

Modify the part program and make sure subprogram calls does not exceed 8 levels.

***　　ERROR-25 WRONG CIRCLE FORMAT OR DATA ERROR**

Message:

The arc command or data format in the program is incorrect.

Recommended remedy:

Check the part program and recalculate the intersection of the arc. Make sure the coordinates of the intersection point is correct.

***　　ERROR-30.1 BATT. LOW**

Message:

The controller battery (BT1) has failed.

Recommended remedy:

Turn on the controller to recharge the battery (BT1) or replace the failed battery (BT1).

***　　ERROR-31 NONE PLC**

Message:

The PLC program does not exist.

Recommended remedy:

Check that the PLC simulation program exists in the FLASHROM and is properly inserted (EVN, ODD location).

*** <u>ERROR-35</u> RS232C PROGRAM NO. ERROR**

Message:

The part program number transmitted from RS232C is incorrect.

Recommended remedy:

Check DNC10 or HCON part program number for correctness.

*** <u>ERROR-36</u> EXECUTION MODE ERROR**

Message:

An error occurred in the selection of execution modes.

Recommended remedy:

Check the selected execution mode for correctness.

*** <u>ERROR-37</u> NC ALARM**

Message:

An error occured in the external control device.

Recommended remedy:

Check the external control device. Remove the error and RESET the function.

*** <u>ERROR-50~99</u>**

Message:

The user defined G65 error signal is incorrectly set.

Recommended remedy

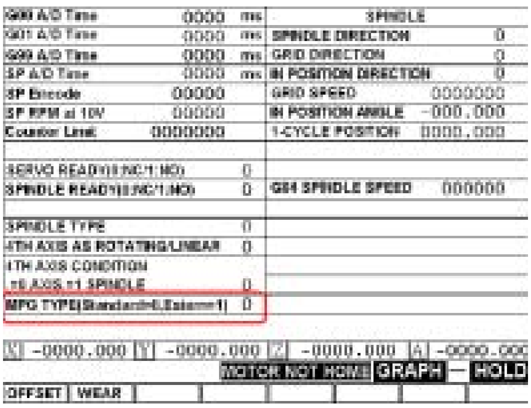
Check that the user defined G65 error signal is correctly set.

7 Attachment A

＊ Input Planning

Input	Description	Remarks
I00	EM-STOP	NC
I01	X-axis HOME LIMIT	
I02	Y-axis HOME LIMIT	
I03	Z-axis HOME LIMIT	
I04	Program start	
I05	Editing mode lock	
I06	A-axis HOME	The fourth axis is axial
I07	A SERVO READY	The fourth axis is axial
I08	MPG hand-wheel A-axis	Hand-wheel type=1
I09	MPG hand-wheel X-axis	Hand-wheel type=1
I10	MPG hand-wheel Y-axis	Hand-wheel type=1
I11	MPG hand-wheel Z-axis	Hand-wheel type=1
I12	MPG X 10	Hand-wheel type=1
I13	MPG x 100	Hand-wheel type=1
I14	X SERVO READY	NC
I15	Y SERVO READY	NC
I16	Z SERVO READY	NC
I17	SPINDLE READY	NC
I18	X-axis + OT	NC
I19	X-axis – OT	NC
I20	Y-axis + OT	NC
I21	Y-axis – OT	NC
I22	Z-axis + OT	NC
I23	Z-axis – OT	NC

Parameter page settings:
MPG TYPE =0, (the framed field in the right picture) indicates a fixed hand-wheel.
Hand-wheel type=1 indicates a handle-type hand-wheel.
I8~I13 are used for selection of the axial signal for the hand-wheel as planned in the above table.



＊ Output Planning

Output	Description	Remarks
O00	Spindle rotation CW	
O01	Spindle rotation CCW	
O02	Coolant	
O03		
O04		
O05	Lubricant	
O06		
O07		
O08		
O09		
O10		
O11		
O12	X SERVO ON	
O13	Y SERVO ON	
O14	Z SERVO ON	
O15	A SERVO ON	

＊ M-code and I/O

M-code	Description	I/O	Remarks
M03	Spindle rotation CW	O00=1	
M04	Spindle rotation CCW	O01=1	
M05	Spindle stop	O00=0,O01=0	
M08	Coolant on	O02=1	
M09	Coolant off	O02=0	
M10	Lubricant on	O05=1	
M11	Lubricant off	O05=0	

＊ USB Expansion

After connecting the USB to the controller for ten seconds, the yellow LCD of the USB illuminates for one second, indicating a successful connection.

Switch to the transmission mode and press the USB button at the bottom of the screen to enter the USB operation mode.

To read data, move the cursor to the file to be read and press the respective button.

To save data, enter a file name and press the respective button to save the data in the file name.

USB can read files containing the following extensions. A file containing an invalid extension unreadable.

Part Program	➔.NCD
MCM Parameter	➔.MCM
PLC	➔.PLC
Screen	➔.TBL
System	➔.SYS
Variable	➔.VAR

"Read part program"	➔Read the part program of the USB device in the controller.
"Read parameters"	➔Read the part program of USB device on the controller.
"DNC"	➔Read the part program of USB device on the controller and execute simultaneously.
"Save part program"	➔Save the program of the controller to the USB device.
"Save MCM parameters"	➔Save the MCM parameters of the controller to the USB device.
"Save variables"	➔Save the controller variables to the USB device.
"Read all programs"	➔Read the part program of the USB device to the controller.
"Read MCM parameters"	➔Read the parameter of the USB device on the controller.

"Read PLC"	➔Read the PLC file of the USB device from the controller.
"Read screen"	➔Read the screen file of the USB device from the controller.
"Read system"	➔Read the system file of the USB device from the controller.
"Read variables"	➔Read the variable file of the USB device from the controller.

8. Attachment B - zDNC Operating Instructions

1. Getting Started

Click  on the desktop to execute zDNC

zDNC

2. Open the Option Setting Screen

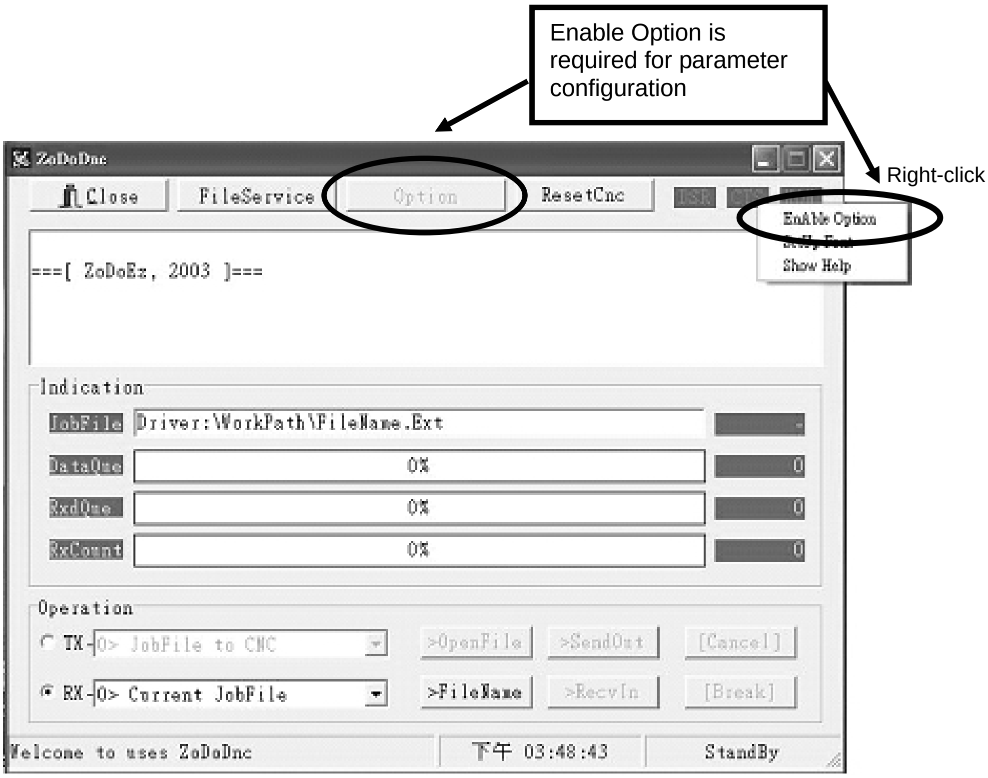
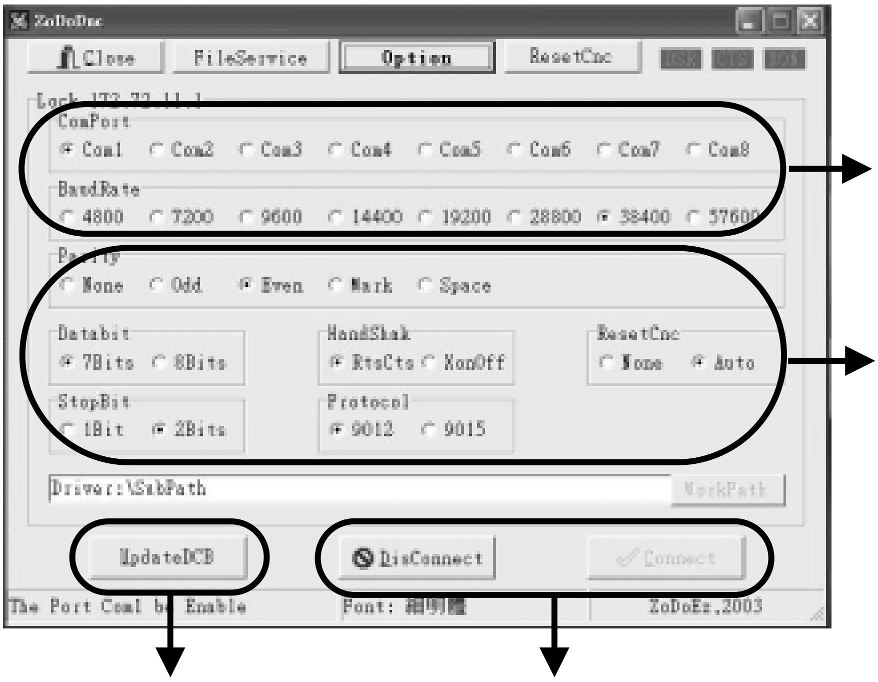


Fig 8-1

3. Display Settings



Corresponding to controller settings

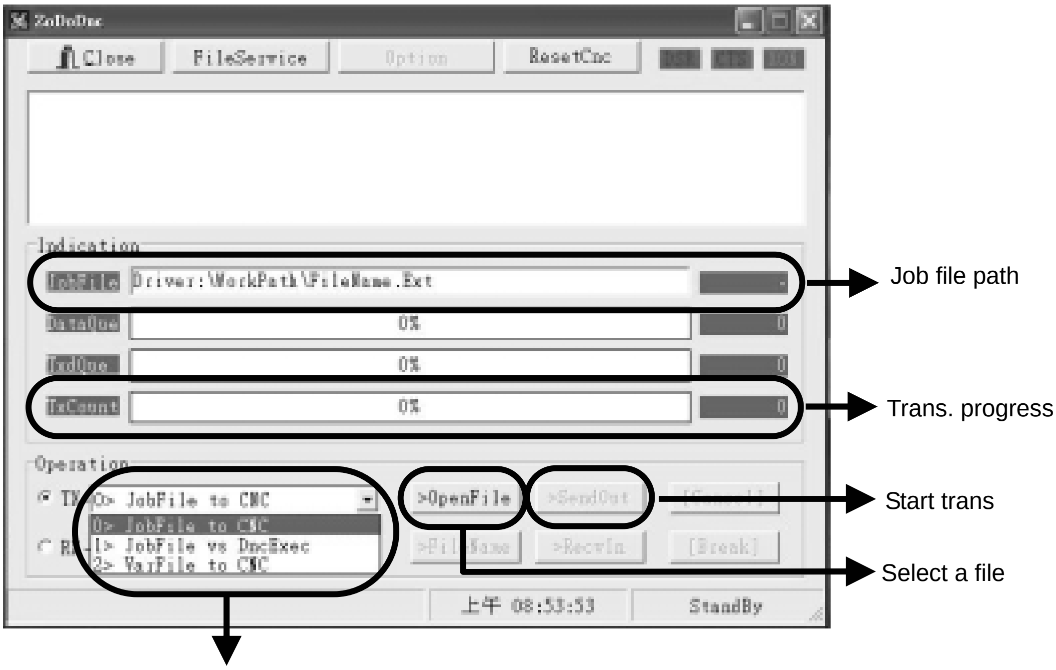
To avoid connection failure, do not check boxes other than those indicated here.

Save the changes

To change the settings, press DisConnect. When the settings are configured, press Connect.

Fig 8-2

4. PC TO CNC



0: Transmit the part program to CNC

1:Transmit the part program to CNC and execute simultaneously (PLC required)

2: Transmit variables to CNC

Fig 8-3

5. CNC TO PC

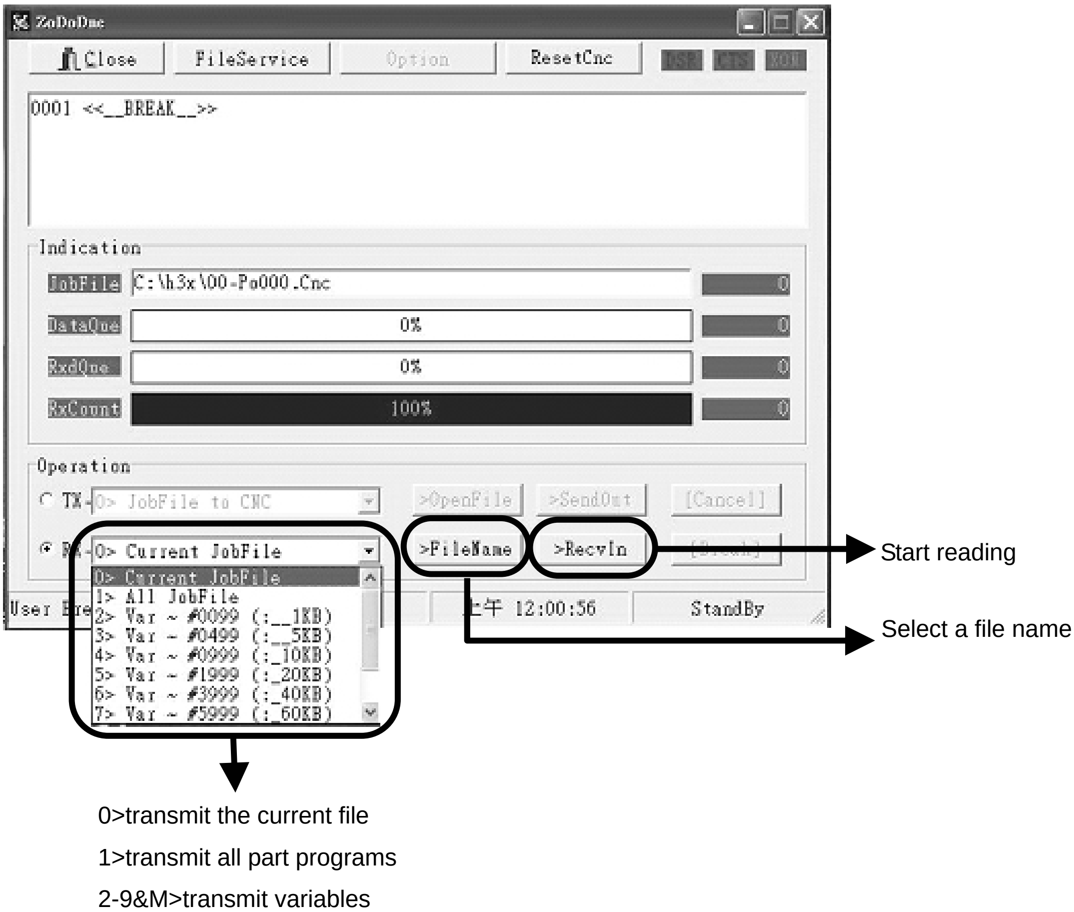


Fig 8-4

6. Attention

※ DNC function is required to transmit huge part programs.

※ PLC should not restrict the availability of R100, R239, C04 when DNC is required, because the system needs to change the value of these three items to enter DNC mode.

※ For DNC operation, settings are only required at the ZDNC (computer) end rather than the controller end, if PLC does not give any restrictions.