

HUST Lathe CNC Controller

Manual

Model: HUST CNC H4CL-T

Version: October, 2006

TABLE OF CONTENTS

1	Main Features of CNC Lathe Controller	1-1
2	Operation	2-1
2.1	Basic Operation	2-1
	Startup Screen	2-1
	Graph Mode Screen	2-2
	MPG – TEST Screen	2-3
	Auto Mode Screen	2-4
	MDI Mode Screen	2-5
	Home Mode Screen	2-5
	Jog Mode Screen	2-6
	Edit Mode display	2-7
	Program Selection Screen	2-8
	I/O Mode Screen	2-10
	Tool Compensation Screen	2-12
	Alarm Description Screen	2-15
	Software Version Screen	2-15
	System Parameter Screen	2-16
	Work Origin Setting Screen	2-17
2.2	Program Editing	2-19
2.2.1	Programming Overview	2-19
2.2.1.1	Part Programs	2-19
2.2.1.2	Programming Methods	2-19
2.2.1.3	Program Composition	2-21
2.2.1.4	Coordinate System	2-23
	Coordinate Axis	2-23
	Coordinate Positioning Control	2-24
	Work Origin	2-26
	Machine Origin	2-27
2.2.1.5	Numerical Control Range	2-29
2.2.2	Program Editing	2-30
2.2.2.1	Creating a New Program	2-30
2.2.2.2	Editing a Program	2-32

2.2.2.3	Entering Decimal Fractions	2-36
2.2.2.4	Editing Notes	2-36

3	G/M Codes	3-1
----------	------------------	------------

3.1	Command codes	3-1
3.2	Rapid Positioning, G00	3-4
3.3	Linear Cutting, G01	3-5
3.4	Arc Cutting, G02, G03	3-7
3.5	Dwell Command, G04	3-11
3.6	Return to the First Reference Point, G28	3-11
3.7	Return to Previous Position from Reference Point, G29	3-12
3.8	Tool Moves to the 2nd Reference Point, G30	3-12
3.9	Skip Function, G31	3-13
3.10	Thread Cutting, G32	3-14
3.11	Tap Cutting Canned Cycle, G33	3-18
3.12	Canned Cycle Functions (For implication of programming)	3-19
3.12.1	Single Cutting Canned Cycle, G90, G92, G94	3-19
3.12.2	Compound Canned Cycle Functions, G70~G76	3-24
3.13	Max. Spindle Speed Setting, G50	3-40
3.14	Constant Surface Cutting Speed Setting, G96	3-40
3.15	Constant Rotation Speed Setting (Constant Surface Cutting Speed Cancellation), G97	3-41
3.16	Feed-rate Setting, G98, G99	3-41
3.17	Inch/Metric Measurement Mode, G20, G21	3-41
3.18	Auxiliary Functions, M-code, S-code	3-41
3.19	Subprograms	3-42
3.20	G40, G41, G42 Tool Radius Compensation	3-44
3.20.1	Total Offset Compensation Setting and Cancellation	3-44
3.20.2	Tool-tip Radius and the Direction of Fictitious Tool-tip	3-46
3.20.3	Tool-tip Radius Compensation, G41 ,G42 ,G40	3-47
3.20.4	Notes on Tool Radius Compensation	3-50

4	MCM Parameters	4-1
4.1	MCM Parameters	4-1
4.1.1	Basic Parameters	4-1
4.1.2	MCM Parameters	4-2
4.2	Description of Parameters	4-5
5	Connections	5-1
5.1	System Connection Descriptions	5-1
5.2	System Installation	5-2
5.2.1	Operating Environment	5-2
5.2.2	Notes on the Control Unit Case Design	5-2
5.2.3	Thermal Case Design	5-3
5.2.4	H4CL-T External Dimensions	5-4
	H4CL-T Panel	5-4
	H4CL-T CPU Main Board Connectors (Rear View)	5-5
	H4CL-T Series Case Dimensions (Rear view)	5-5
	H4CL-T Series Case Dimensions (Top View)	5-6
	H4CL-T Series MDI Panel Dimensions	5-6
	H4CL-T Series Cutout Dimensions	5-7
	H4CL-T Series MDI Panel Cutout Dimensions	5-7
5.3	Input/Output Interface Connection (I/O)	5-8
5.3.1	Input Board / Output Board (Terminal Block Type)	5-8
	NPN Type Standard Input Board (I bit)-24 IN	5-8
	NPN Type Standard Output Board (O bit)-16 OUT	5-8
5.3.2	Input Board/Output Board- CE Standard	5-9
	Input Board	5-9
	Output Board	5-10
5.3.3	I/O Connector Pin Assignment	5-11
5.3.4	Input Signals	5-12
	Input Signal Specification	5-12
	Input Signal Connection Diagram (direct input to the control unit)	5-12
	Input Connecting Diagram (I bit Input Controller)	5-13
5.3.5	Output Signals	5-13
	Output Signal Specification	5-13
	Output Connecting Diagram (Direct Output to Machine)	5-13

	Output Connection Diagram (Output Board to Machine)	5-14
5.4	Connection Diagram	5-15
5.4.1	Connector Type	5-15
5.4.2	Connector Designation	5-15
5.4.3	H4CL-T Connection (Y-shaped terminal)	5-16
	H4CL-T Main Connection Diagram	5-16
	Emergency-Stop Line-1	5-17
	Emergency-Stop Line-2	5-18
	H4CL-T Spindle Connection	5-19
5.4.4	H4CL-T Connection (CE Standard)	5-20
	H4CL-T Main Connection Diagram	5-20
	Emergency-Stop Line-1	5-21
	Emergency-Stop Line-2	5-22
	Servo Motor Connection Diagram	
	(The MITSUBISHI J2S motor is used as an example.)	5-23
5.4.5	H4CL-T Auxiliary Panel + SIO Wiring and Explanation	5-24
	Expansion Output Board size	5-25
	Expansion Input Board size	5-25
5.4.6	Connection Method for Servo Driver & Pulse Generator	5-26
5.4.7	System AC Power Connection	5-27
5.4.8	MPG Connection	5-27
5.4.9	RS232 Connector Pin Assignment and Connection	5-28
5.4.10	D/A Explanation and Wiring (For Spindle)	5-29
	D/A Explanation	5-29
	D/A Wiring	5-29

6	Error Messages	6-1
----------	-----------------------	------------

7	Attachment	7-1
----------	-------------------	------------

	Input Planning	7-1
	Output Planning	7-3
	Output Planning (Servo spindle mode)	7-3
	M-code and I/O	7-4
	G71~G76 Compound Canned Cycle Parameters	7-5
	PLC Parameters	7-6

8 Attachment B-ZDNC Operating Instructions 8-1

Getting Started	8-1
Open the Option Setting Screen	8-1
Display Settings	8-2
PC TO CNC	8-3
CNC TO PC	8-4
Attention	8-4

1 Main-Features of HUST Lathe CNC Controller

Controlled Axes: X, Z and Spindle Encoder Feedback

Program Designed by CAD/CAM on PC. Program input from PC through RS232C interface.

Memory Capacity for CNC main board - 512k.

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.

Tool feed rate can be a millimeter per minute or a millimeter each turn.

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.

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

2 Operation

2.1 Basic Operation

Screen Description

* Startup Screen

After powering the controller, the following startup screen displays:



Fig. 2-1

After 3 seconds, the next screen displays according to the “Mode Selection” setting.

When turning the “PRON” knob from left to right, the following modes are displayed in order:

“Graph” → “MPG - Test” → “Auto” → “MDI” → “Edit” →

“Program Selection” → “Computer Transmission” → “Teaching” → “Manual x 1”
→ “Manual x 10” → “Manual x 100”

“Computer Transmission” and “Teaching” modes are not used. Perform transmission using a PC.

*** Graph Mode Screen**

The following screen displays when the “Mode Selection” knob is set to “Graph”:

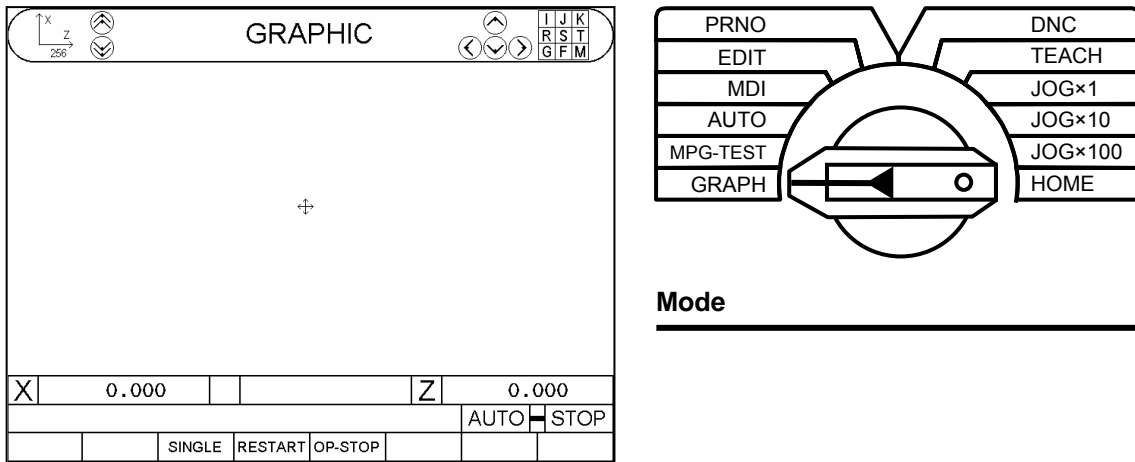


Fig. 2-2

The “+” in the center of the screen indicates the zero position. It can be moved with the Cursor key, or via the characters on the top right corner of the screen.

The number 256 at the top left corner of the screen represents the current horizontal ratio of the graph.

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

There are two graph methods: “Graphing While Moving”, “Quick Graphing”

Switching method: Press the “Quick” in the graph mode before the program starts.

(Press ON and OFF)

HotKey Indicator ON → “Quick Graphing”

HotKey Indicator OFF → “Graphing While Moving”

“Graphing While Moving”: The positioning command of the servo axes and M, T, S codes are performed (the same as auto mode).

“Quick Graphing”: The servo axes are locked but the M, T, and S codes are performed.

The description of the soft keys “Block Execution”, “Restart”, and “Optional Stop” is the same as that in auto mode.

* MPG – TEST Screen

The following screen displays when the “Mode Selection” knob is set to “MPG – TEST”:

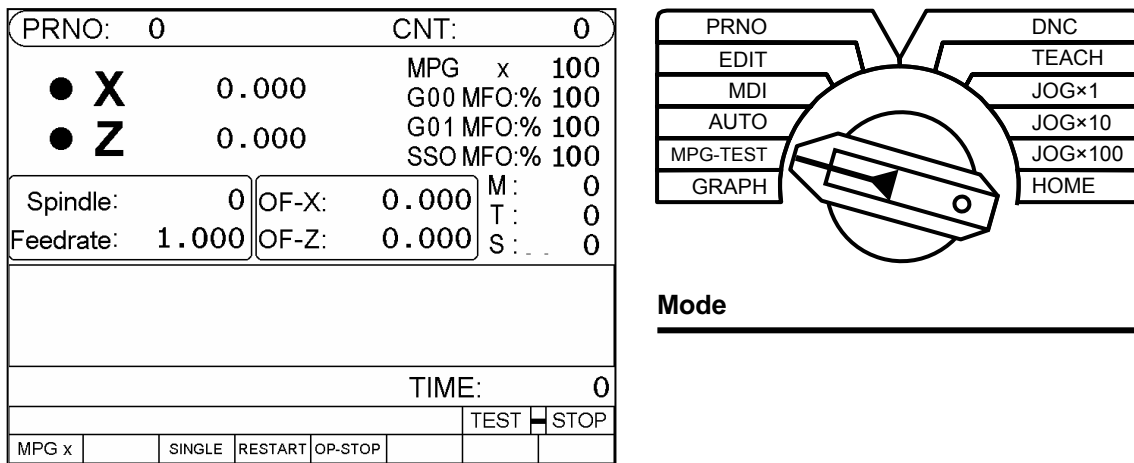


Fig. 2-3

After this mode is selected, the movement of all axes in the program is controlled by the MPG when the program is running. The axis will stop moving when the MPG has no input.

In this mode, the program process will proceed in order when the program is running, regardless of the direction of the hand-wheel. The program process will not return when the hand-wheel is rotated anti-clockwise.

The description of the soft keys “Block Execution”, “Restart”, and “Optional Stop” is the same as in auto mode.

The “HW Multiplication Factor” soft key is for speed adjustment of axial movement after entering the hand-wheel signal.

* **Auto Mode Screen**

The following screen displays when the “Mode Selection” knob is set to “Auto”:

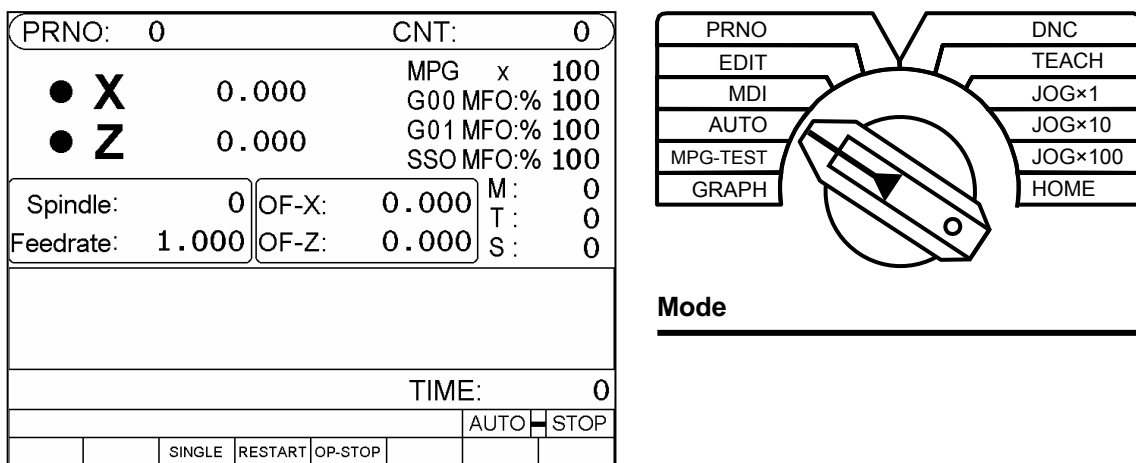


Fig. 2-4

The “soft keys” in auto mode are:

1. **Single Step Execution:** This function can be selected at any time regardless of whether the program is running or not. Whenever the "Start" key is pressed with this function selected, the next command line is executed instead of the entire program.
2. **Restart:** This function needs to be selected before the program runs. When the restart program is selected, operation continues from the previously stopped block. You can search for the stopped block or reset on the edit screen to restart the block.
3. **Optional Stop:** This function needs to be selected before the program runs. When the optional stop is selected, the M01 command in the program is considered a stop command. M01 has no function if this option is not selected.

Part Numbers: The part number increments by 1 when the program runs to M15 and resets to zero when the program runs to M16. Press the "0" key twice to reset.

O13 outputs when the part number reaches the machining count limit (a system parameter).

Part Time: The current running time (sec.) is displayed. The timer is automatically reset when the controller is restarted after the program runs to completion.

*** MDI Mode Screen**

The following screen displays when the “Mode Selection” knob is in “MDI”:

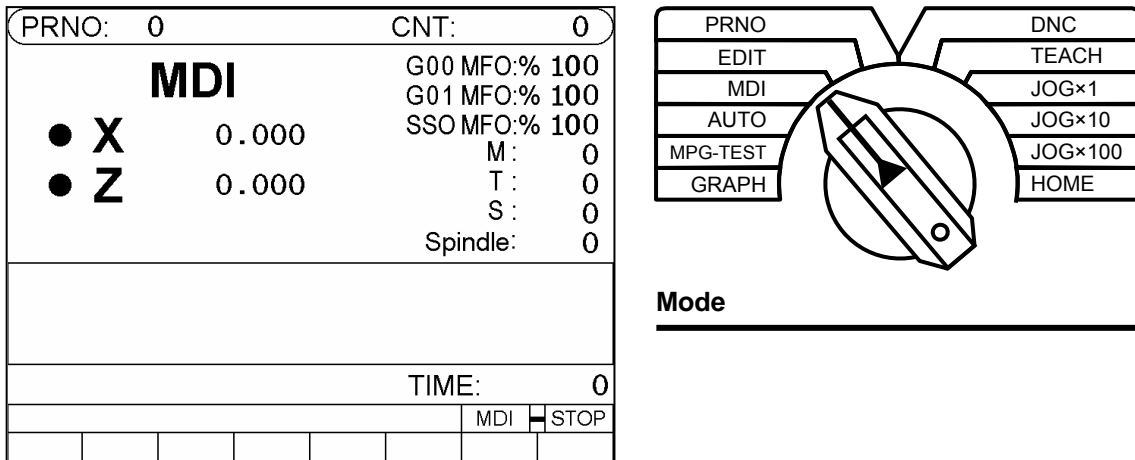


Fig. 2-5

Enter the block command and press the “Start” key in this mode to execute the block command.

*** HOME Mode Screen**

The following screen displays when the “Mode Selection” knob is set to “Home”:

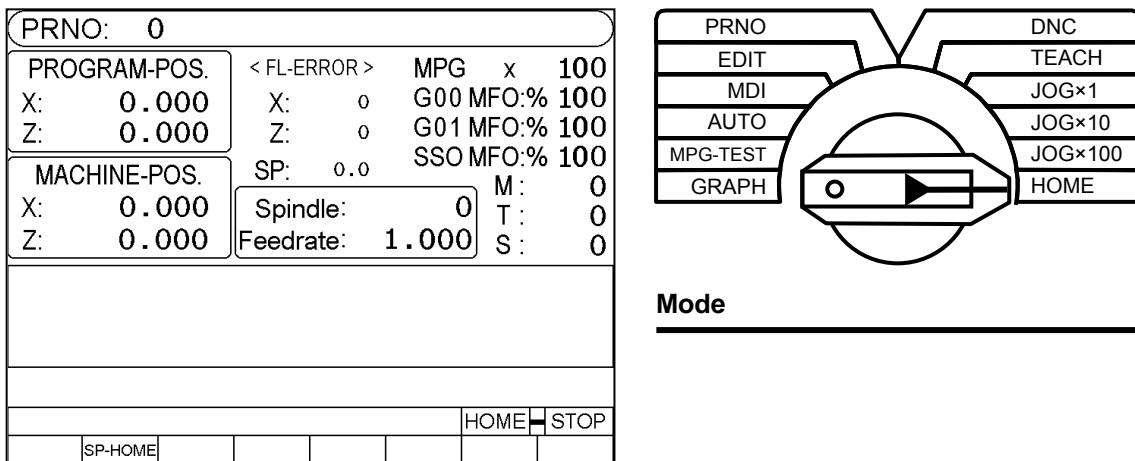


Fig. 2-6

Methods for returning to the origin:

1. The letter X or Z on the screen is highlighted when an axis is selected using the “Axis Selection” knob.
2. Press 「 JOG+ 」 , 「 JOG- 」 , or the start key to execute homing on respective axes.

*** Jog Mode Screen**

The following screen displays when the “Mode Selection” knob is set to “Manual x 1”, “Manual x 10”, or “Manual x 100”:

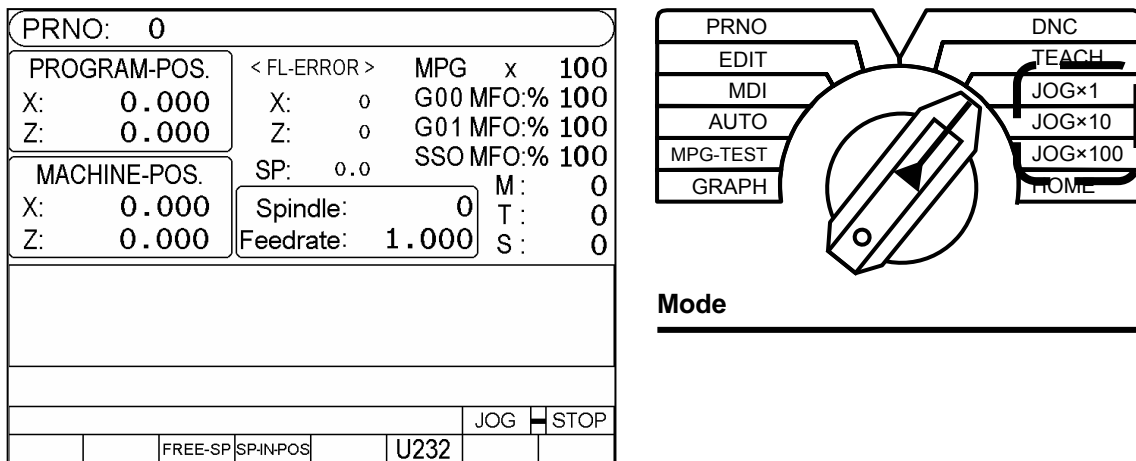


Fig. 2-7

The jog mode provides the following functions:

1. Axis positioning:(Three methods)
 - a. Hand wheel:
Use the “Axis Selection” knob to select an axis (the selected axis is highlighted.) Rotate the hand wheel.
 - b. Continuous movement:
After an axis is selected using the “Axis Selection” knob, continuously press the “JOG +” or “JOG -” to move the corresponding axis in a positive or negative direction. The axis moves at the highest speed when the “High Speed” key is pressed.
(The movement speed of the axis is set from the Parameter page when the “High Speed” key is not pressed.)
2. Manual Switch:
 - a. Spindle: CW, CCW, stop.
 - b. Coolant: Press the key to turn on and press it again to turn off.
 - c. Lubricant: Press the key to provide lubricant after a 1 second delay.
An LED indicates operation.

* Edit Mode display

The following screen displays when the “Mode Selection” knob is in “Edit”:

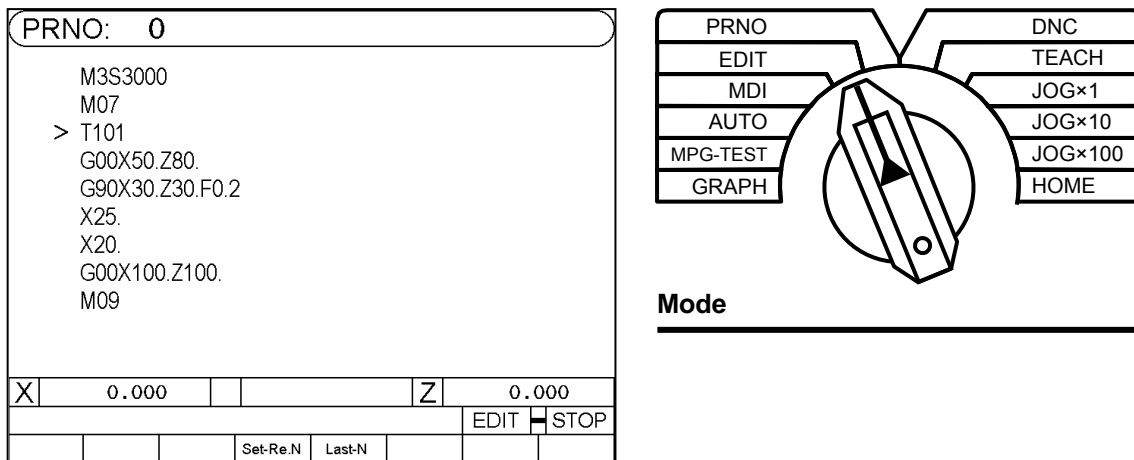




Fig. 2-8

The program can be edited in this mode.

- a. Set to Restart: Use the key   to move to the block to be restarted and press “Set to Restart” once. Press “Restart” during auto mode and “Start” while in edit mode. The pre-configured program block executes.
 - b. Block Interruption: If the program is interrupted during machining, press the “Block Interruption” soft key in edit mode to search for the stopped block.
- * Refer to the **Program Editing** section for more information about editing programs.

*** Program Selection Screen**

The following screen displays when the “Mode Selection” knob is set to “PRON”:

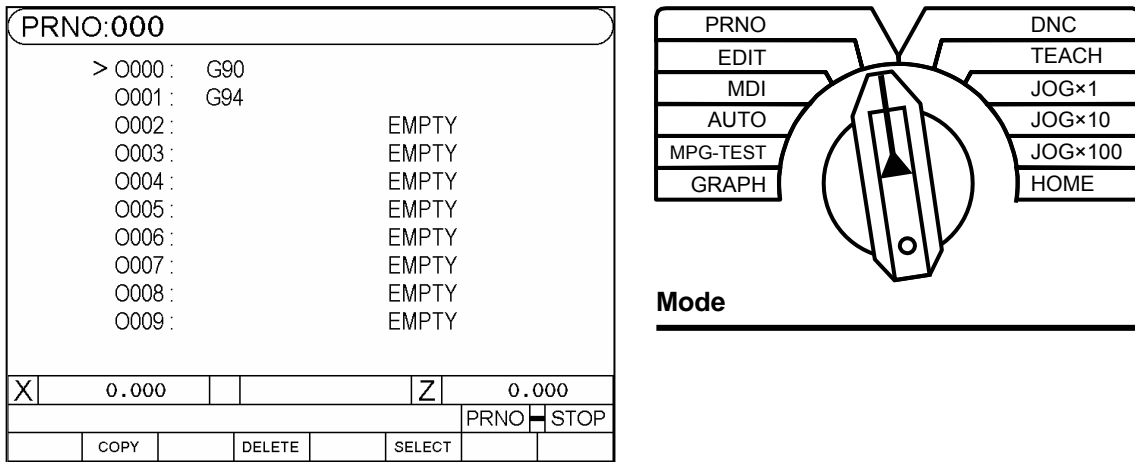


Fig. 2-9

Programmable line numbers range: 0 ~ 699. The controller system uses numbers after 700.

You can enter program comments in this mode up to 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

.

Program selection methods:

1. Select a program:





a. Use the “Cursor ” key or “Page ” key to move the arrow to the desired program number.

b. Press the “Select” or “Enter” key.

2. Program comments:









a. Use the “Cursor ” key or “Page ” key to move the arrow to the program number for which program comments are entered.

b. Enter the desired comment using letters or numbers.

- c. Press the enter key.
3. Delete a program:
- Use the “Cursor  ” key or “Page  ” key to move the arrow to the program number to be deleted.
 - When you press the “Delete” key, a dialogue prompts for confirmation. Press the “YES” or “Y” soft key to delete the program. Press the “NO” or “N” soft key to cancel the operation.
4. Copy a program:
- Press the “Copy” key to display the following screen:

PRNO: 0			
> O000 :	G90		
O001 :	G94		
O002 :		EMPTY	
O003 :		EMPTY	
O004 :		EMPTY	
O005 :		EMPTY	
O006 :		EMPTY	
O007 :		EMPTY	
O008 :		EMPTY	
O009 :		EMPTY	
SOURCE 0		TARGET 0	
X	0.000	Z	0.000
		PRNO STOP	
SOURCE	TARGET	EXE.	

Fig. 2-10

- Use the “Cursor  ” key or “Page  ” key to move the arrow to the **source** program number.
- Press the “Source” key
- Use the “Cursor  ” key or “Page  ” key to move the arrow to the **object** (target) program number.
- Press the “Object” key
- When the source and target program numbers are confirmed, press the “Execution” key to begin copying.

*** I/O (Input/Output) Mode Screen**

Press "I/O/MCM" once to enter I/O mode. The following screen displays:

PRNO: 0			
I00 EM-STOP	I12 Tool 1 Position signal		
I01 X Home LIMIT	I13 Tool 2 Position signal		
I02 Z Home LIMIT	I14 Tool 3 Position signal		
I03 Foot Switch	I15 Tool 4 Position signal		
I04 AUTO / SEMI-AUTO	I16 Tool 5 Position signal		
I05 OPTION SKIP	I17 Tool 6 Position signal		
I06 EDIT LOCK	I18 Tool 7 Position signal		
I07 OPTION STOP	I19 Tool 8 Position signal		
I08	I20 Turret Clamp		
I09	I21 BAR FEEDER OK		
I10 RESET	I22		
I11	I23		
000			
X	0.000	Z	0.000
			AUTO <input type="checkbox"/> STOP
		PANEL	OUTPUT

Fig. 2-11 "Input" Status of the Controller

Press the "Output" soft key to display the following output status screen:

PRNO: 0			
O00 SPINDLE CW	O08 TOOL CW		
O01 SPINDLE CCW	O09 TOOL CCW		
O02 COOLANT	O10		
O03 ALARM LIGHT	O11 BAR FEED		
O04 SPINDLE UNCLAMP	O12		
O05 LUBRICATION	O13 WORKPIECE NO. ON		
O06 UNCLAMP LIGHT	O14 SERVO ON X		
O07 CNC READY	O15 SERVO ON Z		
000			
X	0.000	Z	0.000
			AUTO <input type="checkbox"/> STOP
INPUT		PANEL	

Fig. 2-12 Controller "Output" Status

Press the "Input" soft key to display the input status screen:

Press the “MDI Panel” soft key to display the following MDI panel status screen :

PRNO:000																
I120	I121	I122	I123	I124	I125	I128	I129	I130	I131	I132	I133	I136	I137	I138	I139	I113
MODE		AXIS		GO1%		G00%		SSO%		G						
O111 I111 FEHD		O102 I102 T-CHANGE		O103 I103 LUB		O104 I104 COOLANT		O107 I107 REF-POINT		O108 I108 S-RELEASE						
O112 I112 START		O099 I099 SP-CW		O101 I101 SP-STOP		O100 I100 SP-CCW		O105 I105 T-RADIUS		O106 I106 T-OFFSET						
		O096 I096 JOG+		O098 I098 RAPID		O097 I097 JOG-		O109 I109 DI		O110 I110 D2						
0000																
X	0.000						Z	0.000								
										MPG	HOLD					
INPUT								OUTPUT								

Fig. 2-13 “Input” and “Output” Status of the Controller

PRNO: 0															
				O134 I134		I135		I136							
						EDIT LOCK		AUTO/SEMI-AUTO							
		O096 I096 S-RELEASE		O097 I097 COLLET		O098 I098 T-CHANGE		O099 I099 T-RADIUS		O100 I100 T-OFFSET		O101 I101 GRAPHIC			
O132 I132 FEHD		O133 I133 START		O102 I102 DI		O103 I103 D2		O104 I104 D3		O105 I105 D4		O106 I106 COOLANT		O107 I107 LUB	
		O108 I108 ←C		O109 I109 X↑		O110 I110		O111 I111 G01%		O112 I112 G00%		O113 I113 SSO%			
I140 I141 MODE		I142 I143		O114 I114 ←Z		O115 I115 RAPID		O116 I116 Z→		O117 I117 G01%		O118 I118 G00%		O119 I119 SSO%	
I139 G		O120 I120		O121 I121 X↓		O122 I122 C↓		O123 I123 SP-CW		O124 I124 SP-STOP		O125 I125 SP-CCW			
0000															
X	0.000						Z	0.000							
										TEST	HOLD				
INPUT								OUTPUT							

Fig. 2-13-1 Press the "Output" key on the auxiliary panel to check the output status (key type).

The corresponding screen does not display when this page displays and the work mode knob is turned. This function is used to check the work mode knob is correct.

Press the “Input” soft key to display the input status screen.

Press the “Output” soft key to display the output status screen.

*** Tool Compensation Screen**

Click the “T.Radius / T.Offset” once to enter tool compensation mode. The following screen displays:

Tool Wear Compensation			
NO.	X-AXIS	Z-AXIS	Tool-Radius
01	0.000	0.000	0.000 0
02	0.000	0.000	0.000 0
03	0.000	0.000	0.000 0
04	0.000	0.000	0.000 0
05	0.000	0.000	0.000 0
06	0.000	0.000	0.000 0
07	0.000	0.000	0.000 0
08	0.000	0.000	0.000 0
X	0.000	Z	0.000
			⤴ PAGE ⤵
	WEAR	OFFSET	

Fig. 2-14

In this mode, it is possible to switch between three screens. Press the “soft key” to cycle between tool-wear compensation, tool-length compensation, and parameter screens.

The tool length compensation screen is shown as follows:

Tool Wear Compensation			
NO.	X-AXIS	Z-AXIS	Tool-Radius
01	0.000	0.000	0.000 0
02	0.000	0.000	0.000 0
03	0.000	0.000	0.000 0
04	0.000	0.000	0.000 0
05	0.000	0.000	0.000 0
06	0.000	0.000	0.000 0
07	0.000	0.000	0.000 0
08	0.000	0.000	0.000 0
X	0.000	Z	0.000
			⤴ PAGE ⤵
	WEAR	OFFSET	



Fig. 2-15

There are 40 groups respectively for tool-wear compensation and tool-length compensation.

Note: Switching between pages is possible when the “Page” cursor icon is flashing.

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

and tool-length compensation:



- a. Use the “Cursor  ” key to move the cursor to the parameter to be changed.
- b.
 1. Enter numbers. (Absolute coordinates setting)
 2. The parameter that is set corresponds to the X-axis and has a preamble U . (Incremental coordinates setting)

The parameter set corresponds to the Z-axis and has a preamble W .
- c. Press the Enter key.

Revise values for tool compensation when the program is running:

- a. Only incremental coordinates can be set.
- b. The system parameter Maximum U/W Value during Execution should be set. (Default=0.000.)

2. Parameter Screen:

The parameter screen displays as Fig. 2-16. Press   to switch between pages.


PARAMETERS	X-AXIS	Z-AXIS
JOG Feed	1000	1000
CNT. Limit		0
Collet Type		0
OP-SKIP Funtion 0:OFF 1:ON		0
Feeder Delay Time (10ms)		50
Collet Close Delay Time (10ms)		50
Program Block Start Number		0
Inc. for Program Block Number		0
		PAGE 
WEAR	OFFSET	MCM

Fig. 2-16


PARAMETERS	X-AXIS	Z-AXIS
G71,G72 Cutting depth	0.000 inch	0.000 inch
G71,G72 Retracion	0.000	
G73 Total cutting depth	0.000 inch	0.000 inch
G73 Cutting cycle	0	
G74,G75 Retracion	0.000	
G76 Finish cutting cycle	0	
Chamfer	0	
Tool-tip angle	0	
Min. cutting depth	0.000	inch
Depth for finish cut	0.000	inch
		 PAGE
SYS-MCM	VERSION	ERR-MSG

Fig. 2-17

*** System Parameter Screen**

Press the “System Parameter” soft key on the second page of the parameter screen to display the system parameter page.

- The advanced parameters are protected by a password → the default is 0 (password protection is optional)
- If the advanced parameters password is not “0” then Fig. 2-20 displays.
- If the advanced parameters password is corrected then Fig. 2-21 displays.
- After pressing the “Password Change” key on the system parameter page. Enter the current password and the replacement password as shown in Fig. 2-22.

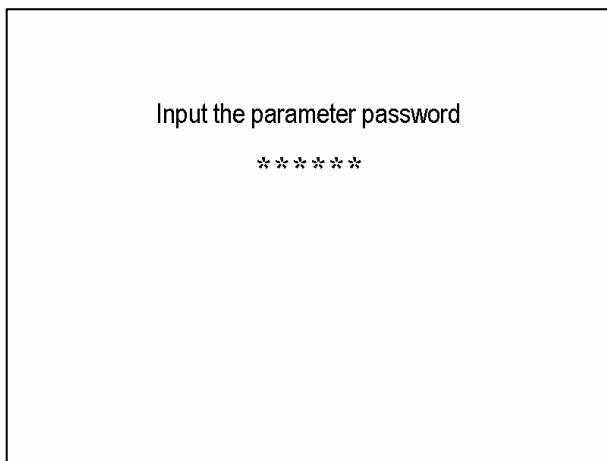


Fig. 2-20

PARAMETERS	X-AXIS	Z-AXIS
Resolution-Den.	100	100
Resolution-Num.	100	100
Encoder-factor (1,2,4)	0	0
Traverse Speed	100000	100000
Rotate direction	0	0
Home Speed-1	2500	2500
Home Speed-2	40	40
Home direction	0	0
Software OT (+)	9999.999	9999.999
Software OT (-)	-9999.999	-9999.999
Backlash Comp	0	0
Pitch Error Comp. (0/1/-1)	0	0
Segment Length for Pitch Err.	20.000	20.000

				PAGE	⌵
Change the password			Pitch Err.		

Fig. 2-21

The password revising	
Input the old password :	*****
Input the new password :	*****
Input the new password again :	*****

Fig. 2-22

* Work Origin Setting Screen

Press the “Work Origin Setting” key to enter the work origin setting screen.

Prerequisite: The work origin setting screen only displays after **homing**.

The following work origin setting screen (1) displays:

PRNO:000			
SAVE-POS.	< FL-ERROR >	MPG x	100
X: 0.000	X: 0	G00 MFO:%	100
Z: 0.000	Z: 0	G01 MFO:%	100
		SSO MFO:%	100
MACHINE-POS.	Spindle: 0	M:	0
X: 0.000	Feedrate: 1.000	T:	0
Z: 0.000		S:	0
Setting Step:			
1. Cut the work-piece press "SAVE-X,Z"			
2. To the next page and press "Write to" twice			
			MPG <input type="checkbox"/> STOP
SAVE-X SAVE-Z		NEXT-P.	

Fig. 2-23

Work origin setting procedure: “Work-piece Cutting and Saving” and “Work-piece Measurements and Dimensions Input”

1. Work-piece Cutting and Saving

- a. Clamp the work-piece for testing (using the foot switch or key switch on the external chuck)
- b. Save the X-axis position
Select an axis for outer diameter cutting with the hand-wheel. Press the “Save the X-axis” key before the X-axis of the tool moves out of the cutting coordinate. The X-axis position of the machine coordinates is

saved.

c. Save the Z-axis position

Select an axis for outer diameter cutting with the hand-wheel. Press the “Save the Z-axis” key before the Z-axis of the tool moves out of the cutting coordinate. The Z-axis position of the machine coordinates is saved.

Press the “Next Page” soft key to enter the work origin setting (2) screen

2. Work-piece Measurement and Dimension Input

The following work origin setting screen (2) displays:

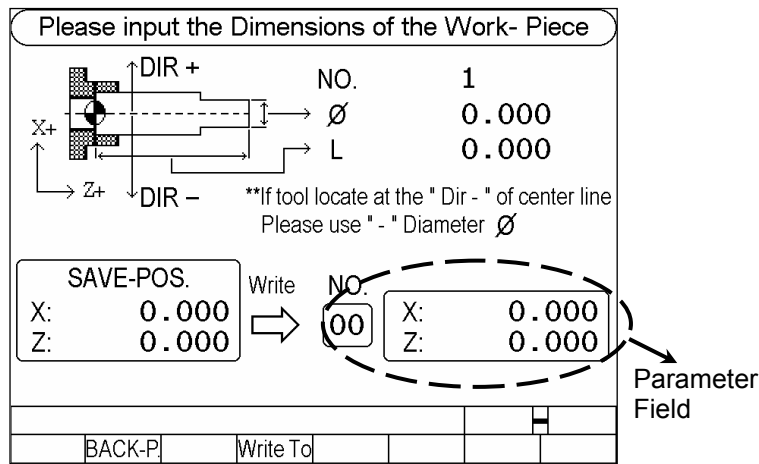


Fig. 2-24

a. Work-piece Measurement

Remove the tool and the work-piece to measure its dimensions.

b. Enter the group and dimensions

Use “Cursor ↑↓” to move to group, X-axis diameter, and Z-axis length to be set. After entering the desired value, press the “Enter” key.

c. Write the Parameter

Press the “Write” key have the controller write the given **parameter field for tool length compensation** and display immediately. See Fig. 2-24.

2.2 Program Editing

2.2.1 Programming Overview

2.2.1.1. Part Programs

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 machine.
2. Determine the work-piece loading method and select the 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 a 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, punched paper tape, or keyboard.

2.2.1.2. Programming Methods

A numerical control unit executes actions exactly in accordance with the commands of the part program. So, programming is very important for numerical control machining. There are two ways to design a CNC part program and they are briefly described in the following:

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 and the controller are performed manually.

The coordinates and movements of the tool used in machining operations should be first calculated 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. Both geometric and trigonometric operations are required for accurate curves. After acquiring the coordinates 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 drawing the part diagram to transmitting the part program are performed with a PC.

For complex part shapes, it is both time-consuming and error-prone to calculate the coordinate values manually, resulting in nonconforming-machined products. To make use of the high-speed capabilities of the computer, the programmer designs a simple part program 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 a PC, which is in turn transmitted to the CNC controller via the RS232C interface. This is called the CAD/CAM system and is used by many units using CNC machines to create a program especially machining a 3-D work-piece.

2.2.1.3. Program Composition

A complete program contains a group of blocks. A block has a serial number and several commands. Each command is composed of a command code (letters A~Z) and numeric values (+, -, 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 G0 X40.000 Z10.000
N20 G00 X30.000 Z5.000
N30 M3 S3000
N40 G1 X10.000 F200
N50 W-5.000
N60 X15.000 Z-10.000
N70 X30.000 W-10.000
N80 G0 X40.000 Z10.000
N90 M5
N100 M2
```

Block is the basic unit of a program. A block contains one or more commands. No blank should be inserted between commands when transmitting a program. A block has the following basic format:

N-___G___X(U)___Z(W)___F___S___T___M___

N : Block Sequence Number

G : Function Command

X, Z : Coordinate positioning command (absolute movement command).

U, W : Coordinate positioning command (incremental movement command).

F : Feed rate.

S : Spindle speed.

T : Tool command.

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, arc cutting, or thread cutting.
2. **Positioning Command:** X, Z, U, W commands, for example, instructs the G-code of the machine to stop 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, T, 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.

Basic Command Format (e.g. the positioning command):

```
X-10.000
X          : Command code
"-        : +/- sign (+ can be omitted)
10.000    : Destination of a tool positioning action.
```

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-T Series and their definitions are described below.

- F: Feed-per-rotation command. mm/rev or mm/min
- G: Function code
- I: The X-axis component of the arc radius.
- K: The Z-axis component of the arc radius
- L: Repetition counters
- M: Machine control code
- N: Program serial number
- P: Call subprogram code; parameter in canned cycles; tool number for

tool compensation.

- Q: Parameter in canned cycles
- R: Arc radius or parameter in canned cycles
- S: Spindle speed.
- T: Tool command
- U: Incremental positioning command on X-axis
- W: Incremental coordinates on Z-axis
- X: Absolute positioning command on X-axis.
- Z: Absolute positioning command on Z-axis.

Each block has a specified format and this format must be used during programming. Incorrect formatting can result in code rejection or major errors.

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” followed by some digits. The number can be generated automatically or manually typed from the keyboard when editing the program. (Refer to Chapter IV). The line number order is not followed, but line numbers must be unique. 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)

2.2.1.4. Coordinate System

Fabrication of a work-piece with a lathe is accomplished by the rotation of the spindle and cutting motion of the tool mounted on the machine. The tool can move in an arc or straight line. A coordinate system is used to describe the geometrical position 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).

*** Coordinate Axis**

The H4CL-T Series uses the well-known 2-D Cartesian coordinate system. The two axes used in the lathe series are defined as X-axis and Z-axis. The Z-axis is

the centerline of the lathe spindle. The intersection of the two axes is the zero point, i.e. X=0 and Z=0.

Fig. 2-25 shows the relationship among each axis, tool motions, and rotation direction of a work-piece. This manual uses the rear tool post as an example.

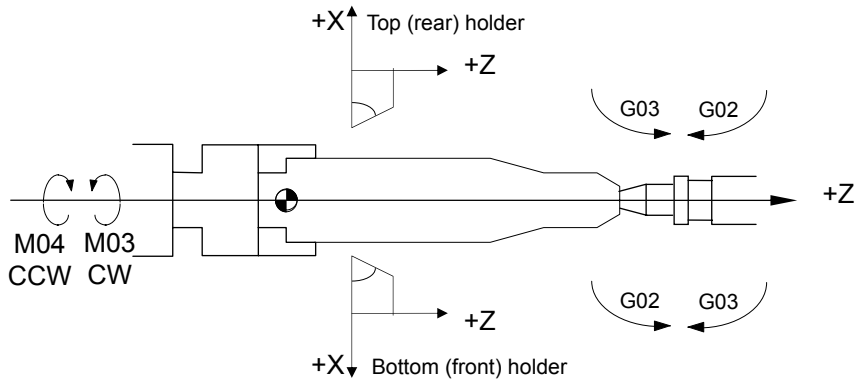


Fig. 2-25 Coordinate System of the CNC Lathe

When the spindle is rotating, your thumb points to the positive direction of the Z-axis and four fingers point to the direction of normal rotation.

* **Coordinate Positioning Control**

The coordinate of the H4CL-T Series is either absolute or incremental, depending on the command code of the coordinate axis, i.e.:

X, Z: Absolute coordinate commands.

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

Please note the diameter is generally used to represent the X-axis coordinate for a lathe, regardless of incremental or absolute coordinates.

Absolute Coordinate Commands

Tool-positioning coordinates are acquired with reference to the origin (work origin or program origin) of the work coordinate system. 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 incremental coordinate is either positive (+) or negative (-). The negative sign represents decrement. Facing toward the direction of movement, if the tool is heading towards the positive (+)

direction, U, W represents an increment. If it is heading to the negative (-) direction, U, W represents decrement.

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

Absolute Commands: (Fig. 2-26)

P0 to P1	G01 X10.000 F0.200
P0 to P2	X24.000 Z30.000
P2 to P3	X32.000 Z10.000
P3 to P4	Z0.000

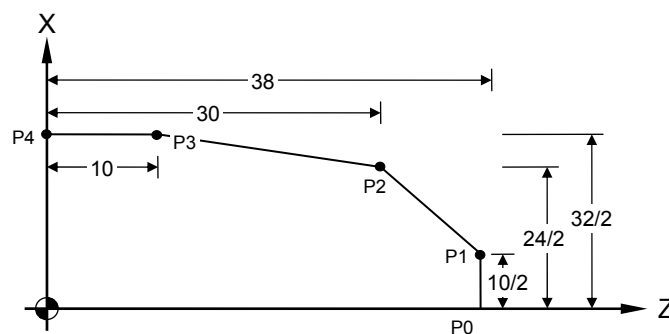


Fig. 2-26 Absolute Commands

Incremental Commands: (Fig. 2-27)

P0 to P1	G01 U10.000 F0.200
P1 to P2	U14.000 W-8.000
P2 to P3	U8.000 W-20.000
P3 to P4	W-10.000

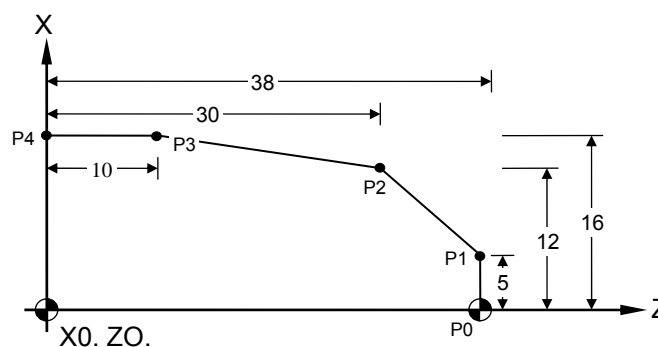


Fig. 2-27 Incremental Commands

Coordinate Interchange:

P0 to P1	G01 X10.000 F0.200
P1 to P2	X24.000 W-8.000
P2 to P3	U8.000 Z10.000
P3 to P4	W-10.000

or

P0 to P1	G01 X10.000 F0.200
P1 to P2	U14.000 Z30.000
P2 to P3	X32.000 W-20.000
P3 to P4	Z0.000

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 coordinates 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 the incremental coordinate system is used.

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

For diagonal (simultaneous positioning on the X and Z-axis) or arc movements, the coordinate value of each axis acquired from trigonometric operations 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 there are, the greater the risk of error. Basically, whether an absolute or incremental coordinate is used depends on the programming requirements and the specifications of the machining diagram.

*** Work Origin**

The specifications of the machining diagram are converted to the coordinate system at the CNC lathe 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 centerline of the spindle. However, it is recommended to select an origin that makes reading of the work-piece coordinate easier. The X-axis of the work origin should be on the centerline of the lathe spindle. One of the following three points can be selected as the work origin of the Z-axis: (Fig 2-28)

1. Left end of the work-piece.
2. Right end of the work-piece.
3. Front of the claw or chuck.

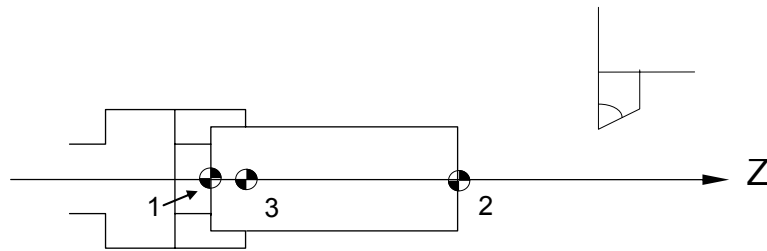


Fig. 2-28 Work Origin Selection (1, 2, or 3)

The work origin is also called work zero point or program origin, program zero point. In this manual, this zero point is always referred to as the work origin. The coordinate system based on the origin is called work coordinate system. The work coordinate origin is the work origin. Referring to section 3.12 for the G10 and G50 work origin setting.

The work-piece after being cut with a CNC lathe is symmetrical. Perform machining of half the side of the work-piece. Therefore, only half of the work-piece should be drawn on the work-piece diagram when creating a program, as shown in Fig 2-29.

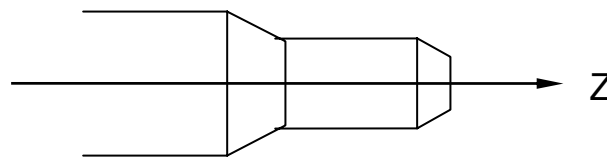


Fig. 2-29 Work-Piece Symmetrical Diagram

* 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 coordinate (or work origin) and calibration of the tool length compensation. This reference point is called the machine origin.

For the H4CL-T Series controller, the machine origin is the stop position of the tool when the homing for each axis is complete. As Fig. 2-30 shows, the machine origin corresponding to the coordinate used to indicate the work origin varies depending on the position of the work origin. 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. In this case, use the positioning measurement function of the controller to obtain the relationship between this position and the coordinate used for the work origin.

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.

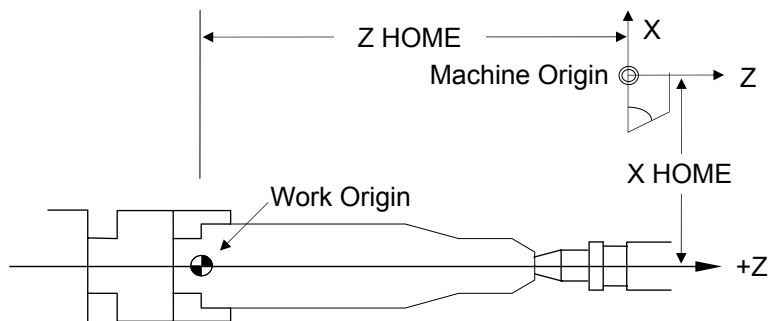


Fig. 2-30 Lathe Origin Diagram

2.2.1.5. Numerical Control Range

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

Min. setting unit	0.001 mm
Max. setting unit	9,999.999 mm
Min. moving unit	0.001 mm
Max. moving unit	9,999.999 mm
Max. stroke	9,999.999 mm

G- code	G00~G99 (G01=G1)
M-code	M000~M999 (M01=M1)
S-code	S1~S9999 rpm
F-code	0.001~0~9999.999 mm/spin
X, Z, U, W, I, K	0.001~+/-9,999.999 mm
R (Radius)	0.001~+/-9,999.999 mm
G04	0 ~ 9999.999 seconds
Program number	0 ~ 999
T-code	<ol style="list-style-type: none"> 1. There are two digits after T when no turret is mounted; select a tool compensation number. 2. There are four digits after a turret is mounted; the first two digits is the tool selection and the last two digits is compensation number.
Memory capacity	512 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.

2.2.2 Program Editing

The program editing operation includes:

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

2.2.2.1. New Program Editing

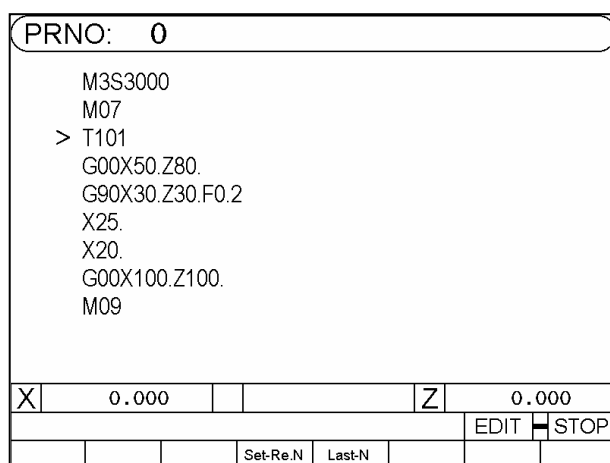


Fig. 2-31

The following keys are used to edit programs:

1. Command keys.
2. Numerical keys 0 ~ 9
3. Cursors – Use \uparrow or \downarrow to move the cursor to the block to edit.
4. Used the \uparrow or \downarrow key to switch to the pervious or next page of the program.
5. Use the Insert key to create or insert a new block.


Enter a new block in a new program or insert a new block in an existing program. Press the Insert key after entering a new block.

6. Use the Enter key to apply/save the new changes.
Use the Enter key after adding a command or changing a command value in an existing block.
7. Use the Delete key to delete a program block.




Creating a Program Example:


Program 1
 N1 G0 X0.Z0.
 N2 G4 X1.
 N3 G0 U480.W-480.
 N4 G4 X1.
 N5 M99

Action and Description:

1. Make sure the controller is in the program-editing mode. Press the  key or turn the knob to begin editing.

2. Enter data:

First block data:   

Press the  key to create a new block, as shown in Fig. 2-32:

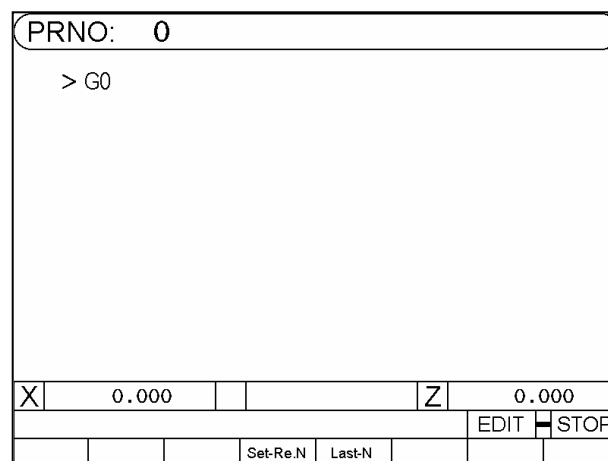





Fig. 2-32





Then enter:

The above-mentioned procedure is used to edit the first block data. Enter the following data for 2nd ~ 5th blocks:

Second block:   

Third block:

G	0	Insert				
U	4	8	0	•	Enter	
W	-	4	8	0	•	Enter

(Note that the sign

-

 can be entered before the

Enter

 key is pressed.)

Fourth block:

G	4	Enter	
X	1	•	Enter

Fifth block:

M	9	9	Insert
---	---	---	--------

If the size of a program exceeds one page, use ↑ or ↓ to check the program on each page for correctness.

2.2.2.2. Editing a Program

We have created PROGRAM 1 in the previous section. The existing program change is described in this section. Changing a program includes the following procedure:

Add (or Change) a Command

Ex: The third block program N3 U480. W-480.
 Changed to N3 U480. W-480. F0.2

Procedure:

1. Make sure the system is in “EDIT” mode.
2. Use ↑/↓ to move to the cursor to block N3.
3. Enter a command code and value to be added (changed), e.g. F0.2.

F	0	•	2	Enter
---	---	---	---	-------

The screen shows as Fig. 2-33.

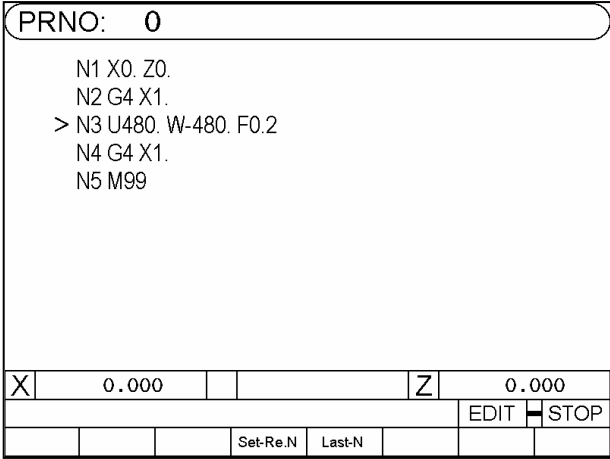


Fig. 2-33

1. Change U480. by entering U360;

U	3	6	0	.	Enter
---	---	---	---	---	-------

 To change an incorrect command, enter the correct command and press

Enter

.

Delete a Command

Ex: The third block program N30 U480. W-480. F0.2
 Changed to N30 U480.W-480.

Procedure:

1. Make sure the system is in “EDIT” mode.
2. Use the

⬆

 /

⬇

 key to move the cursor to block N3.
3. Enter a command to be deleted without values, e.g;

F

Enter

(No value is entered behind F). The screen shows as Fig. 2-34:

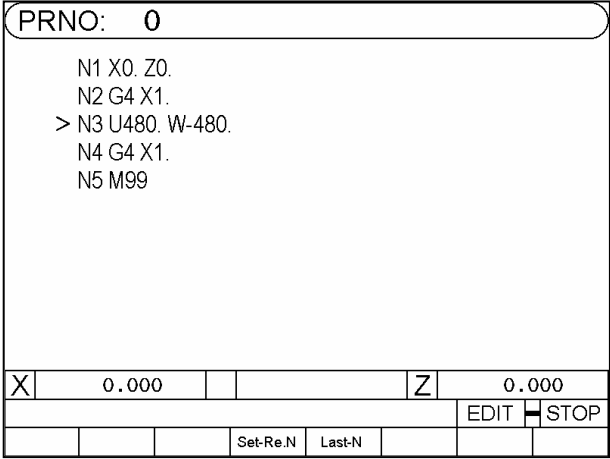




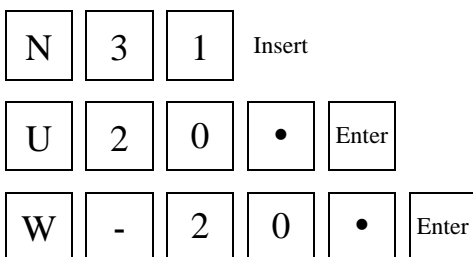
Fig. 2-34

Insert a Block

Ex: Insert the block N31 U20. W-20 between the third block N3 G0 U480. W-480 and between N4 G4 X1

Procedure:

1. Make sure the system is in "EDIT" mode.
2. Use the  /  to move the cursor to block N3.
3. Enter



The screen shows as Fig. 2-35.

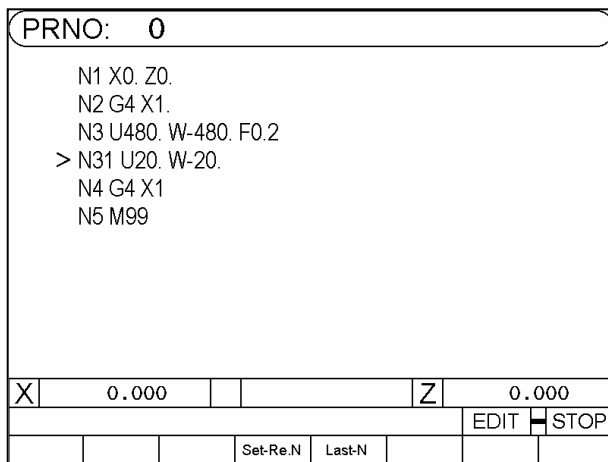





Fig. 2-35

Delete a Block

Ex: Delete the block N31 U20. W-20.

Procedure:

1. Make sure the system is in "EDIT" mode.
2. Use  /  to move to the cursor to block N31.
3. Press the  key .The screen shows as Fig. 2-36.

Move the cursor to block N4 after the block N31 is deleted.

PRNO: 0						
N1 X0. Z0.						
N2 G4 X1.						
N3 U480. W-480.						
> N4 G4 X1.						
N5 M99						
X	0.000			Z	0.000	
					EDIT	STOP
		Set-Re.N	Last-N			

Fig. 2-36

Delete a Program

In the "PRNO" mode, move the cursor to the program to be deleted and press the key. The following message displays:

PRNO:000						
O000 : G90						
O001 : G94						
> O002 : EMPTY						
O003 : EMPTY						
O004 : EMPTY						
O005 : EMPTY						
O006 : EMPTY						
O007 : EMPTY						
O008 : EMPTY						
O009 : EMPTY						
DELETE O002 (y/n) ?						
X	0.000			Z	0.000	
					PRNO	STOP
		Yes	No			

Fig. 2-37

At this time, press the key to delete the program O02. When you press the key, no action is performed.

To delete all programs 0~999, follow the procedures below: Procedure:

Switch to the MDI mode and instruct the command **G10 P2001**. All contents in the program are deleted.

Note: After the procedure is complete, all program data in memory is erased. Therefore, never perform this action unless it is absolutely necessary.

2.2.2.3. Entering Fractions

A command value is entered in either integer or real-number format with a maximum of 7 digits. You cannot enter a fraction for a parameter that requires an integer. You can insert a decimal point at the specified position for a command that requires real-number input. The input is corrected after being processed by the control unit. An error may occur if an integer is entered for a command that requires a real-number. This is further explained in the following paragraphs.

When an integer is entered for a command (such as X, Y, I, J) that requires a real-number, the control unit automatically puts a decimal point at the position specified in the given format. The table below shows various values after validation.

Input	4/3 Format
X2	X0.002 mm
Z35	Z0.035mm
U2500	U2.500 mm
W125.	125.000 mm
F300	F0.3 mm/spin

For commands that require real-number input, the integer is changed. Although the screen still shows the entered data. The user should pay attention to this. To avoid errors, it is recommended to enter data with a decimal point. The "0" after the decimal point can be omitted. Integer codes, such as G, M, N, S, are not affected.

G, M, N, S codes: Variables	Integer input.
X, Y, Z, U, V, W, I, J code	Real input.
F-code	Integer input.

Suggestion: To avoid confusion, except for the G, M, N, S, all other commands require real-number input. The "0" after the decimal point can be omitted.

2.2.2.4. Editing Notes

Block (Program) Serial number

1. The letter N of the block serial number can be omitted if necessary.
2. The number after N is only a symbol. The blocks are sorted in line order rather than the assigned line value.

For instance, if N35 is inserted behind N30, the order is:

```

Program 1
N10 G0 X0. Y0.      ..... First block
N20 G4 X1.          ..... Second block
N30 U480. V-480.    ..... Third block
N35 U20. V-20.      ..... Fourth block
N40 G4 X1           ..... Fifth block
N50 M99             ..... Sixth block
  
```

If the block line number N35 is changed to N350 the program executes in the same order.

3. The line number of a block is edited in the form of a "string". That is to say, N10, N010, N0010 represents different blocks and a complete string must be entered to search a block serial number.

Block Notes

1. Do not use two G-codes in the same block.
2. Do not repeat any coordinate code of a command, such as X, Y, Z, U, V, I, W, J and R, in the same block.
3. If you specify absolute coordinates and incremental coordinates for the same axis in a block, only the incremental coordinates will be executed.
Example:
G1 X100. U50.----- Only U50 will be executed.
4. A maximum of 80 characters can be entered in a block, or the Err-08 message displays.

3 G/M Codes

3.1 Command codes

The previous chapters have introduced the format of part programs. This chapter will describe the command codes of the H4CL-T series and provide simple examples for each command to explain its applications.

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

1. One-shot G-codes

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

Ex: N10 G0 X30.000 Z40.000

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

N30 G1 X20.000 Z50.000 . . . G04 no longer valid in this block.

2. Modal G-codes

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

Wherein G00, G01, G02, G03 Same group.

G40, G41, G42 Same group.

G96, G97 Same group.

G98, G99 Same group.

Ex: N10 G0 X30.000 Z5.000 . . . G0 is defined.

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

N30 G1 X30.000 F0.2 . . . G1 replaces G0 and becomes valid.

The G-codes of H4T controller are listed in Table 3-1.

Table 3-1 G-Code Definitions

G-code List			
G- code	Function	G- code	Function
* 00	Fast positioning (fast feeding)	70	Canned cycle, fine cut
* 01 #	Linear cutting (cutting feeding)	71	Compound canned cycle, lateral rough cut
* 02	Arc cutting, CW (cutter at rear)	72	Compound canned cycle, traverse rough cut
* 03	Arc cutting, CCW (cutter at rear)	73	Compound canned cycle, contour rough cut
04	Dwell command	74	Lateral Grooving Canned Cycle
		75	Traverse grooving canned cycle
10	Data input	76	Compound thread cutting canned cycle
20	System measurement in INCH mode	80 #	Drilling canned cycle cancellation
21	System measurement in METRIC mode	* 81	Drilling canned cycle setting
28	Tool moves to the 1st reference point	* 82	Drilling canned cycle (dwell at bottom)
29	Return to the previous position from the ref. point	* 83	Deep hole drilling canned cycle
30	Tool moves to the 2nd reference point	* 85	Boring canned cycle
31	Skip function	* 86	Boring canned cycle (spindle stop at hole bottom)
32	Thread cutting	* 89	Boring canned cycle with dwell at hole bottom)

33	Tap Cutting Canned Cycle		
* 34	Tapered thread cutting	90	Single lateral cutting canned cycle
* 40 #	Tool radius compensation cancellation	92	Single thread cutting canned cycle
* 41	Tool radius compensation setting (left)	94	Single traverse cutting canned cycle
* 42	Tool radius compensation setting (right)	* 96	Constant surface cutting speed setting
50	Coordinate system & max. spindle speed setting	* 97 #	Surface cutting at constant speed cancellation
		* 98	Feed-rate setting (mm/min)
		* 99 #	Feed-rate setting (mm/rev.)
<ul style="list-style-type: none"> · # -- Default controller settings · * -- Modal G-codes 			

3.2 Fast Positioning, G00

Format:

G00 X(U)____ Z(W)____

X, Z : End point in absolute coordinates.

U, W : End point in incremental coordinates relative to the block starting point.

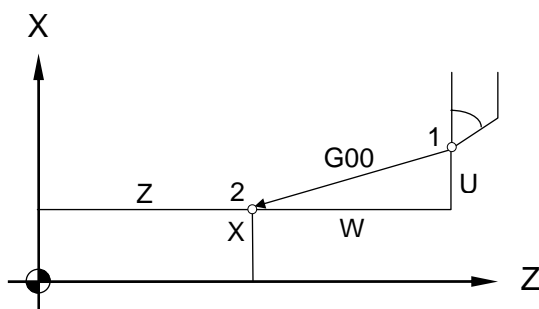


Fig. 3-1 Fast positioning

G00 (or G0) is used to instruct the tool to move to the defined end point of a program block at the maximum speed of MCM #33. The start point is the position at which the tool is located before it moves. This command can control the movement of 1~2 axes simultaneously. The axis that is not set by the command does not execute any movement.

Ex: Fig 3-2, A point moves to B point rapidly.

G0 X4.00 Z5.60 . . . X and Z-axes are set with absolute commands

G0 U-6.00 W-3.05 . . . X and Z-axes are set with incremental commands

G0 X4.00 W-3.05 . . . X and Z-axes are set with absolute or incremental commands

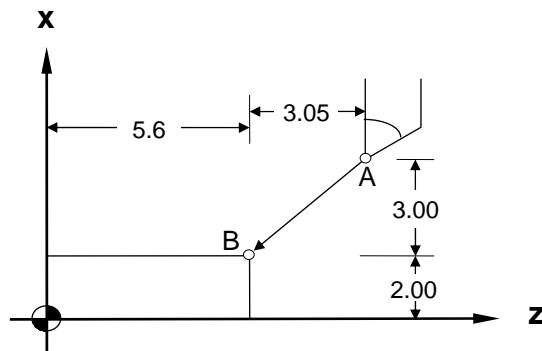


Fig. 3-2 G00 Programming Example

Tool moves to X4.00, Z5.60 rapidly. Since both X and Z axes are repositioning, the tool moves according to the lower feed-rate set in the parameter “Highest Feed-rate”.**Ex:** Fig. 3-2 assuming that the “Highest Feed-rate” is:

$$X = 5000.00 \text{ mm/min}, Z = 3000.00 \text{ mm/min},$$

Then $F_z = 3000.00$. . . Z-axis feed-rate

$$F_x = 3000.00 * (3.00/3.05)$$

$$= 2950.82 \text{ (less than } 5000.0, \text{ X-axis set value)} \quad . . . \text{ X-axis feed-rate}$$

The feed rate of both axes is within the MCM parameter settings. Therefore, the tool will feed at the calculated rate on both axes.

When only a single axis (X or Z) executes fast positioning, it moves at the respective speed set in the “Highest Feed-rate” parameter.

3.3 Linear Cutting, G01

Format:

G01 X(U)____ Z(W)____ F____

X, Z : End point in absolute coordinates

U, 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.

G01 (or G1) is used for linear cutting work. It can control the X, 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 defined after an F-code (Modal code) remains valid until it is replaced by a new feed-rate.

The formula to calculate X, Z cutting feed-rate:
(U and W are actual incremental values.)

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

$$\text{Z feed-rate, } F_z = \frac{W}{\sqrt{U^2 + W^2}} \times F \quad (2)$$

Example: Start point is X=2.0 (diameter), Z=4.60.

G01 X4.00 Z2.01 F0.300 . . . Absolute command

G01 U2.00 W-2.59 F0.300 . . . Incremental command

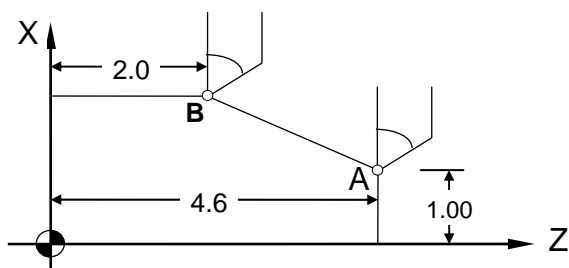


Fig. 3-3 G01 Programming Example

3.4 G02, G03 Arc Cutting

The arc-cutting program contains four command groups, as showed in the list below. The combination of these commands determine the arc path of the tool in a single block.

			Command	Description
1	Arc feed direction		G02 G03	Clockwise Counter clockwise
2	End point	Absolute command Incremental command	X, Z U, W	End point in absolute coordinates Increment from arc start point to end point
3	Difference from arc start point to center Arc radius		I, K R	I=X-axis, K=Z-axis Radius range -9999.~9999.mm
4	Arc feed-rate		F	Minimum setting 0.01 mm/rot.

The end point can be defined either by absolute or incremental coordinates. The size of the arc can be defined either by the coordinate difference or radius. The arc cutting direction (CW or CCW) is relative to the center of the arc. Note that the CW or CCW direction is determined when the tool is at the top (rear) holder. The direction is reversed when the tool is at the bottom (front) holder.

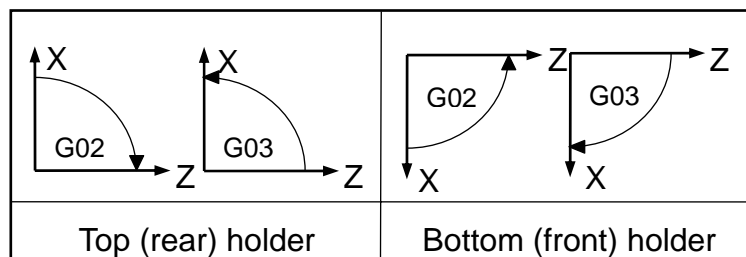


Fig. 3-4 G02, G03 Direction

Arc cutting command:

	Top (rear) holder	Bottom (front) holder
G02	Clockwise	Counter clockwise
G03	Counter clockwise	Clockwise

An arc comprises three elements, a start point, and end point and a center (See Fig. 3-5). The start point (S) is the tool coordinates when the G02 and G03 execute.

The end point (E) is the coordinates of X (U) and Z (W) in the program format.

The center (C) is defined by I and K values. They are the coordinate difference between the arc start point and center. This value can be either positive or negative. Definition of the I and k values are same as the increments (U, W).

The arc feed-rate is defined by F-value.

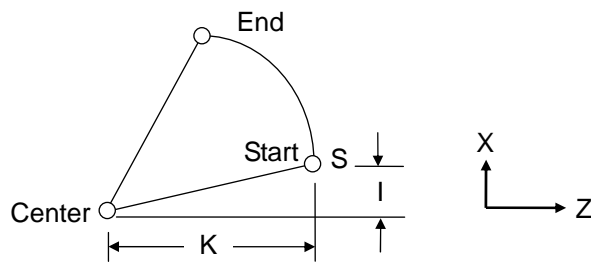


Fig. 3-5 Arc cutting

The arc center can be defined by the radius instead of I and K.

But if the arc angle is between -1° and 1° or 179° and 181° , only I and K can be used for setting.

Format:

G02 X(U)___ Z(W)___ I___ K___ F___

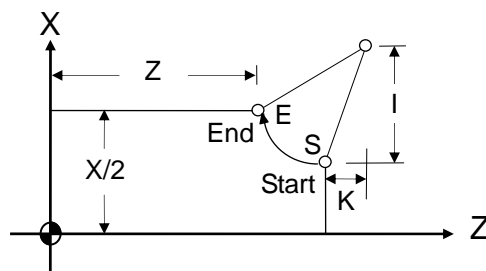


Fig 3-6 G02 Arc cutting

G03 X(U)___ Z(W)___ I___ K___ F___

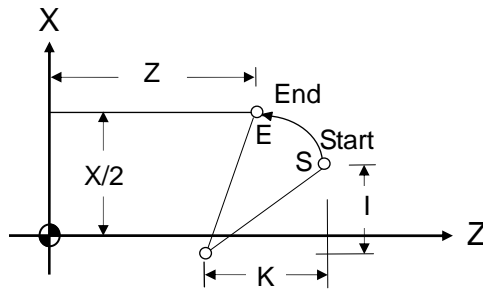


Fig 3-7 G03 Arc Cutting

G02 X(U)___ Z(W)___ R___ F___

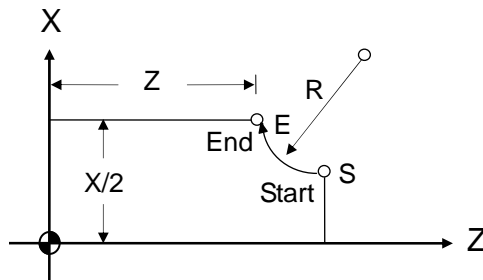


Fig. 3-8 Defined by Radius "R"

Example: The following four commands are different in settings but execute the same arc cutting work.

1. G02 X5.000 Z3.000 I2.500 F0.3
2. G02 U2.000 W-2.000 I2.500 F0.3
3. G02 X5.000 Z3.000 R2.500 F0.3
4. G02 U2.000 W-2.000 R2.500 F0.3

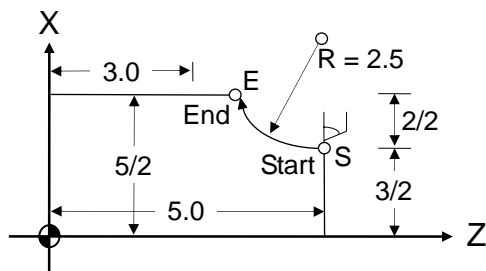


Fig. 3-9 G02 Programming Example

There are two different arc types available for arc cutting (Fig. 3-10):

1. Use "+R" if arc angle $< 180^\circ$.
2. Use "-R" if arc angle $> 180^\circ$.
R is within the range from -4000.mm to +4000.mm.

Ex: In Fig. 3-10, an arc is cut with an angle $< 180^\circ$ (+R):

G02 Z60.000 X20.000 R50.000 F0.300

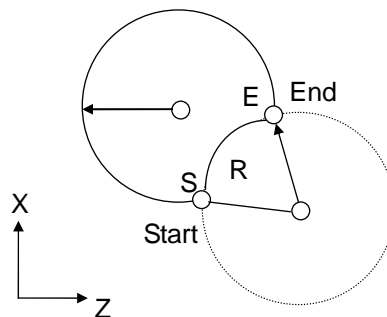


Fig. 3-10 Arc cutting

Please note the following 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 the H4CL-T system uses a fixed $1 \mu\text{m}$ 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:

$$F_c = 85 \times \sqrt{R \times 1000} \text{ mm/min}$$

Where R= Arc radius in mm.

3.5 Dwell Command, G04

Format:

G04 X_____

X: Dwell time in sec (the X here indicates time rather than position).

To meet machining requirements, the axial movement may need to be held during the execution of a program block, which completes before the command for the next block is executed. This command can be used for this purpose. The G04 function is used for this purpose.

The minimum dwell time is 0.001 sec, the maximum is 8000.0 seconds.

Ex: N1 G1 X10.000 Z10.000 F0.1

N2 G4 X2.000 hold for 2 seconds

N3 G00 X0.000 Z0.000

3.6 Return to the First Reference Point, G28

Format:

G28

or G28 X_____ Z_____

or G28 X_____

or G28 Z_____

The first reference point coordinates are set based on the X, Z, and settings in MCM parameter G28. The X, Z values in this format are not used. They only indicate which axis is to return to the reference point. Therefore, regardless of whether G28 is an independent block or contains X, Z commands simultaneously, the tools return to the reference point based on the X, Z settings of the MCM parameter.

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

Ex: T100 . . . Tool compensation is canceled (it cannot co-exist with G28 in the same block).

G28 X10. . . . Tool returns to the 1st reference point on the X-axis.

3.7 Return to Previous Position from Reference Point, G29

Format:

G29 X____ Z____

The X, Z, values in this format are not used. 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: G01 X60. Z30. . . . Tool moves to the position X60., Z30.
 N2 G28 . . . Tool returns from X60, Z30 to the first reference point.
 N3 G29 . . . Tool returns from the reference point to X60., Z30.

As the example above, the N3 block may have the following combinations:

N3 G29 . . . Return to X60.00, Z30.00
 N3 G29 X ____ Z ____ . . . Return to X60.00, Z30.00
 N3 G29 X ____ . . . Tool returns to X60.00
 N3 G29 Z ____ . . . Tool returns to Z30.00

3.8 Tool Moves to the 2nd Reference Point, G30

Format:

G30 X____ Z____

Execution of this command is the same as G28, but the reference point is set in MCM parameter G30.

3.9 Skip Function, G31

Format:

G31 X (U)____ Z(W)____

X, Z: Predicted end point in absolute coordinates

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

P: Defined skip point.

To ensure a 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, Z coordinates. Once an I/O signal is detected during cutting, the G31 skip function is established and the block G31 is skipped.

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 defined block.

Ex:
 N50 G31 W100.000
 N60 G01 U50.000
 N70 Z90. X60./2

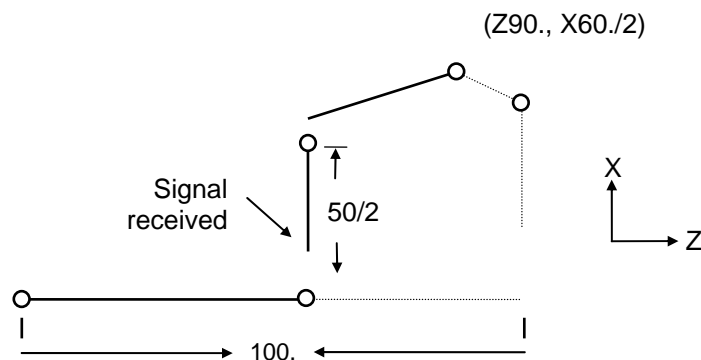


Fig. 3-11 G31 Skip Function

In Fig. 3-11, the dashed 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 is given.
- The skip function is invalid during the program dry run, feed-rate adjustment and auto acceleration/deceleration.

3.10 Thread Cutting, G32

The G32 command is applicable to tapered and non-tapered thread cutting work.

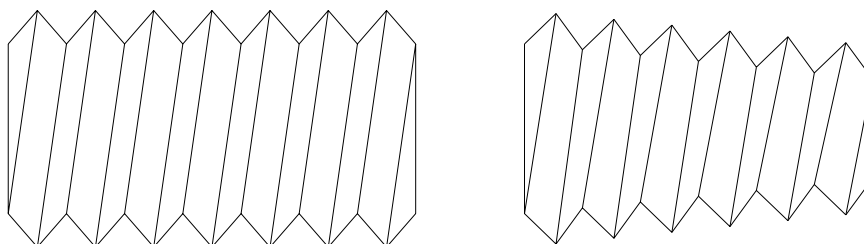


Fig. 3-12 G32 Thread cutting

Format:

G32 X(U)___ Z(W)___ F___

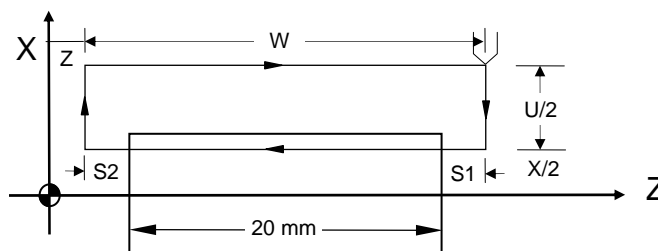


Fig. 3-13 Thread Cutting

(The U/2 Setting Should Not be Less Than Retraction Amount).

- X, Z : End point of thread cutting in absolute coordinates
- U, W : End point of thread cutting in incremental coordinates relative to the start point.
- F : Thread pitch

Both fine cut and rough cut of the thread cutting proceed along the same path. The cutting action on the Z-axis does not start until the Grid signal is received from the spindle. All repeated cutting actions start at the same point.

Due to delay of the server system, imperfections could result at both ends of the thread (S1 and S2). To avoid this problem, the thread length specified in the program should be slightly longer than the actual length of the processed thread. S1 and S2 are leads. The length of S1 and S2 is estimated using the formula below.

$$S1 = (S * F/1800) * (-1 - \ln A)$$

$$S2 = (S * F/1800)$$

S1, S2 : Imperfect thread length, mm

S : Spindle speed, rpm

F : Thread pitch, mm

A : Acceptable thread error

Relationship between A and (-1 - Ln A):

A	-1 - Ln A
0.005	4.298
0.010	3.605
0.015	3.200
0.020	2.912
0.025	2.689

Programming example 1: Non-tapered thread cutting

Specifications: Thread pitch F= 2 mm,

cutting lead starts S1 = 3 mm,

cutting lead ends S2 = 3 mm,

Thread depth = 1.4 mm (in diameter) by 2 cuts

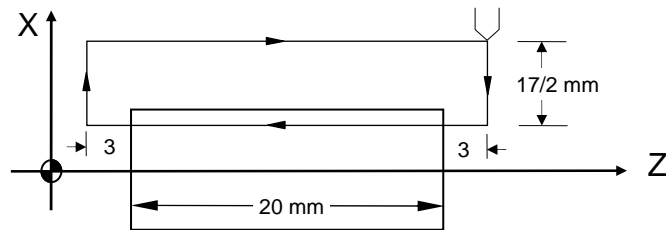


Fig. 3-14 Non-tapered Thread Cutting

```

N10 G0 X30.0 Z50.0
N20 M03 S2000
N30 G0 U-17.000 (first cut = 1.0/2mm)
N40 G32 W-26.000 F2.00
N50 G0 U17.000
N60 W26.000
N70 G0 U-17.400 (second cut = 0.4/2mm)
N80 G32 W-26.000 F2.00
N90 G0 U17.400
N100 W26.000
N110 M05
N120 M02
    
```

Programming example 2: Tapered thread cutting

```
G32 X(U) _____ Z(W) _____ F _____ R_____
```

- X, Z : End point of thread cutting in absolute coordinates.
- U, W : End point of thread cutting in incremental coordinates relative to the start point.
- F : Thread pitch.
- R : Half of the difference (diameter) between the greater and smaller ends of the tapered thread.

Specifications:

Thread pitch $F=2$ mm
 Cutting lead starts $S1 = 2$ mm,
 Cutting lead ends $S2 = 2$ mm,
 Thread depth = 1.4 mm (diameter) formed by two cutting
 actions.

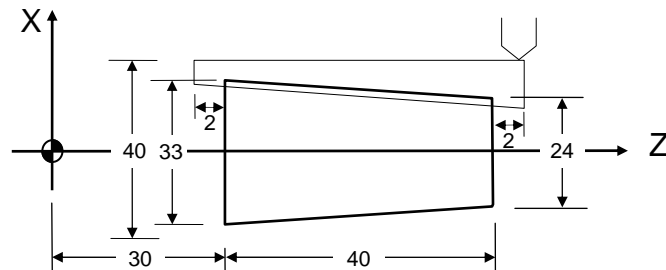


Fig 3-15 Tapered Thread Cutting

If the angle between the tapered thread and Z-axis is smaller than 45° , the thread pitch is set based on Z-axis. Otherwise, X-axis applies.

```

N10 GO X60.0 Z100.0
N20 M03 S2000
N30 G0 X23.0 Z72.0 (First cut = 1.0/2mm)
N40 G32 X32.000 Z28.000 F2.00 R-4.5
N50 G0 X40.000
N60 Z72.000
N70 G0 X22.6 (Second cut = 0.4/2mm)
N80 G32 X31.6 Z28.0 F2.00 R-4.5
N90 G0 X40.000
N100 Z72.000
N110 M05
N120 M02
  
```

3.11 Tap Cutting Canned Cycle G33

Format:

G33 Z (W)_____ F_____

Z (W) : End point coordinate of taping length

F : Thread pitch

G33 Execution process of Z-axis tap cutting canned cycle

1. Z-axis starts tap cutting
2. Spindle off
3. Wait for complete stop of spindle
4. Spindle reverses (opposite direction to the previous round)
5. Z-axis retraction
6. Spindle stops

Programming example: A single thread with a pitch of 1mm

```
N10 M3 S800
```

```
N20 G33 Z100. F1.0
```

```
N30...
```

Note 1: The rotating direction of the spindle should be determined based on the rotating direction of the tap before taping starts. The spindle stops when the cutting is complete. It needs to be restarted when further processing is required.

Note 2: Since this command is applicable to rigid taping, a deceleration time is reserved for the spindle after the stop signal of the spindle becomes valid and the Z-axis keeps moving along with the rotation of the spindle until it completely stops. Therefore, the actual depth of the pitch is deeper than required.

Note 3: Refer to G32 thread cutting for other notes.

3.12 Canned Cycle Functions (For implication of programming)

The canned cycle function is a special G-code of command groups. It comprises canned cycle cutting actions commonly used in machining processes. The command groups of H4CL-T Series are classified into single canned cycle and compound canned cycle command groups. Both are handy and effective in programming and applications.

3.12.1 Single Cutting Canned Cycle, G90, G92, G94

This command group executes repeated cutting with a block. It should end with G01 after use; otherwise, the cutting cycle will repeat.

1. Outer/Inner Diameter Linear Lateral Canned Cycle, G90

Format:

G90 X(U)_____ Z(W)_____ F_____

X, Z : End point C in absolute coordinates (Fig. 3-20)

U, W : End point C in incremental coordinates relative to the start point A

F : B~C~D feed-rate

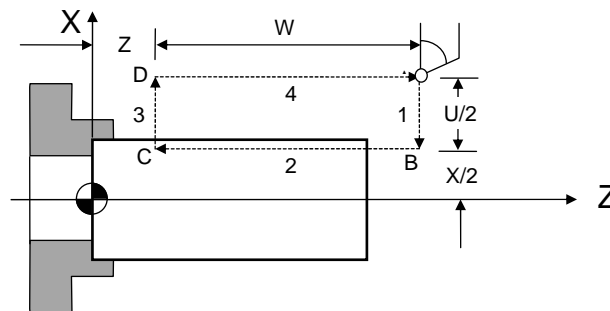


Fig. 3-20 G90 Linear Cutting Path

In Fig. 3-20, the cutting paths 1 and 4 are fast positioned by G00. The cutting along the paths 2 and 3 is executed at the feed-rate F. Whenever the start button (CYCST) is pressed in a block, the tool moves along the paths 1~2~3~4 to execute a cutting cycle.

2. Outer/Inner Diameter Tapered Lateral Canned Cycle, G90

Format:

G90 X(U)_____ Z(W)_____ R_____ F_____

R : The difference between point B and C in radius.

X, Z, U, W and F are identical to those in lateral linear canned cycle.

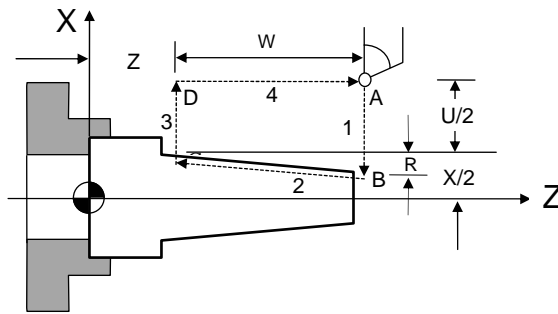


Fig. 3-21 G90 Tapered Cutting Path

When using incremental coordinates, the signs (+/-) of U and W are determined by the tool's direction of movement. If the direction is positive, the increment of U and W is (+), and vice versa. R value is as Fig. 3-22.

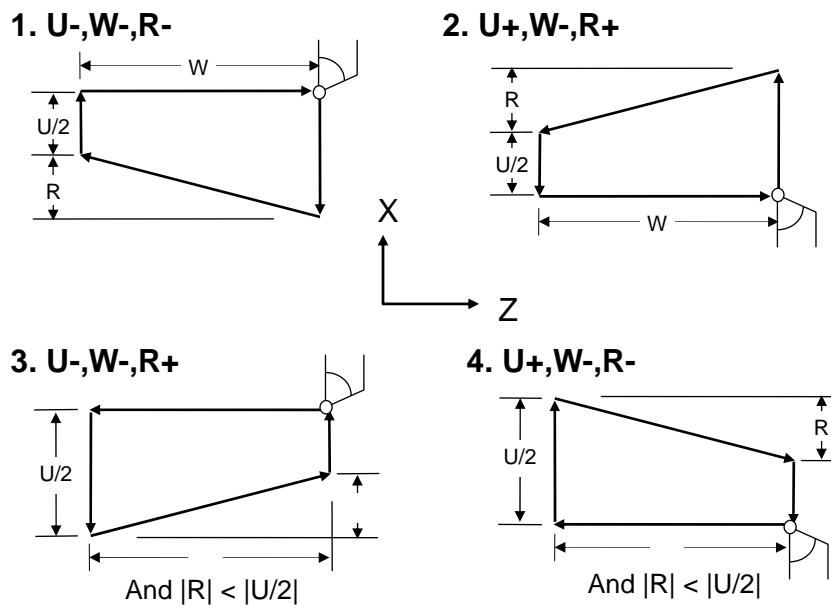


Fig. 3-22 G90 Cutting Path and Direction

3. Linear Thread Cutting Canned Cycle, G92

The advantage of the G92 block is that it functions as four G32 blocks.

Format:

G92 X(U) ___ Z(W) ___ I ___ K ___ L ___ Q ___ F ___

X, Z : End point C in absolute coordinates

U, W : End point C in incremental coordinates

X, Z, U, W, L, Q, F are identical to those of the linear thread cutting canned cycle.

Description of the tapered thread cutting is identical to linear thread cutting.

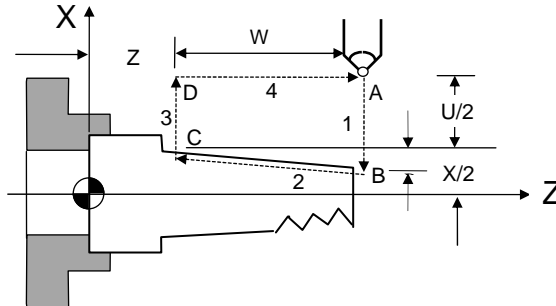


Fig. 3-24 G92 Tapered Thread Cutting Canned Cycle

5. Linear Traversed Canned Cycle, G94

Format:

G94 X(U)____ Z(W)____ F____

X, Z : End point C in absolute coordinates

U, W : End point C in incremental coordinates relative to the start point A

F : B~C~D feed-rate

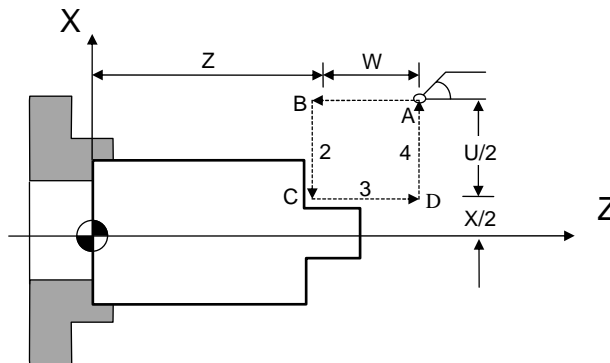


Fig. 3-25 G94 Linear Traversed Cutting Path

In Fig. 3-25, the cutting paths 1 and 4 are fast positioned by G00. The cutting along paths 2 and 3 is executed at the feed-rate F. Whenever the start button (CYCST) is pressed in a block, the tool moves along the paths 1~2~3~4 to execute a cutting cycle.

6. Tapered Traversed Canned Cycle, G94

Format:

G94 X(U) Z(W) R F

R : The difference between point B and C in radius.

X, Z, U, W and F are identical to those of the linear traversed canned cycle.

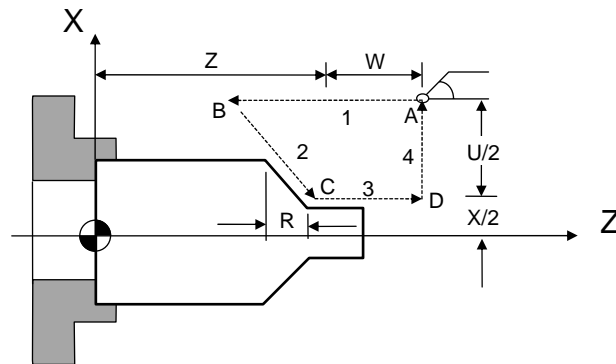


Fig. 3-26 G94 Tapered Traversed Cutting Path

When using incremental coordinates, the signs (+/-) of U and W are determined by the tool's moving directions. If the moving direction is positive, the increment of U and W is (+), and vice versa. R value is as Fig. 3-27.

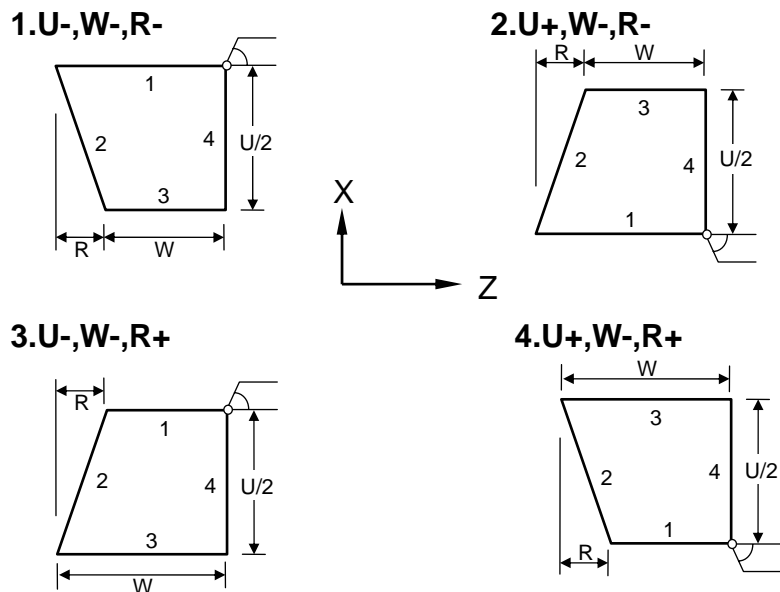


Fig. 3-27 G94 cutting Path and Direction

Note that G90, G94, G92 are modal codes and all the values for X(U), Z(W) and R remain valid unless they are redefined or another G-command is given.

As shown in Fig. 3-28, if the length of movement on Z-axis is fixed, the canned cycle is repeated merely by executing the X-axis positioning command.

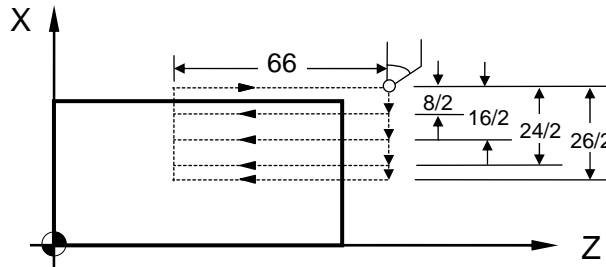


Fig 3-28 G90 Programming Example

```

N10 G0 X80.0 Z100.0
N20 M3 S2000
N30 G90 U-8.0 W-66.0 F2.00
N40 U-16.0
N50 U-24.0
N60 G0 U-26.0
N70 G1 W-66.0 F1.00 . . . Finishing cut with G01
N80 U2.0
N90 G0 X80.0 Z100.0
N100 M5
N110 M2
    
```

3.12.2 Compound Canned Cycle Functions, G70~G76

Compound canned cycles simplifies the operation of CNC commands, once the data of a work-piece is set for fine cut, the CNC automatically determines the tool path for the rough cut. Compound canned cycles are also used for thread cutting. This function is particularly suited for column cutting.

1. Compound Canned Cycle, Lateral Rough Cut, G71

Format:

G71 U(d) R(e)

G71 P(ns) Q(nf) U(u) W(w) F(f) S(s) T(t)

T_____ X_____ Z_____ S_____

N(ns)

.

.

N(nf)

G70 P(ns) Q(nf) --- Fine cut (See the above description)

When changing the tool for fine cutting, the tool number T**** should be inserted in the line before N(ns). Repeated insertion of the number is allowed but the inserted T-codes (the tool numbers) must be the same. The X and Z of the first block with the T-code indicate the safe position to which the tool retracts for changing. The other blocks that contain T-codes indicate the safe path to return to (ns)(ns). No blocks with T-codes will be executed for the rough cut.

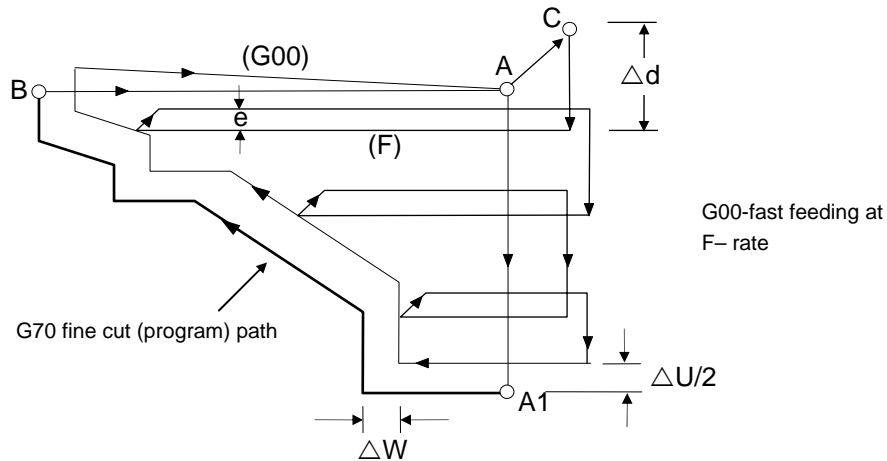


Fig. 3-29 Tool Path of G71 Rough Cut Canned Cycle

In Fig. 3-29, the fine cut path is A~A1~B. A~C is the distance reserved for fine cut tool retraction. The cutting depth is U(d). The amount of the material to be removed for fine cutting is (u/2) and (w). The amount of retraction after each cut is R(e). The path of the final rough cut is parallel to the path of the fine cut. The definition of command groups in the program format is described below:

U(d) : Cutting depth (radius programming ,+).
If not specified, the **parameter "G71, G72 Feeding Amount"** is used.

R(e) : Amount of retraction after each rough cut (radius programming).
If not specified, the **parameter "G71, G72 Retraction Amount"** is used.

P(ns) : The number of the first block for a fine cut cycle.

Q(nf) : The number of the last block for a fine cut cycle.

U(u) : Amount of material to be removed for fine cut, X-axis.

W(w) : Amount of material to be removed for final cut, Z-axis.

F(f),S(s),T(t): F = feed-rate. S = spindle speed, T = tool selection.

The F, S, and T functions of G71 and previous blocks are applicable to G71, but all F, S, and T functions from N(ns) to N(nf) are not applicable to G71. They are only applicable to the fine cut command G70.

- . N(ns)~N(nf) specify the machining path of A1~B.
- . A maximum of 50 blocks can be inserted from N(ns) to N(nf).
- . No subprogram is available from N(ns) to N(nf).
- . No assignment of positioning commands on Z-axis is allowed from A to A1.
- . The feed-rate from A to A1 is either G00 or G01.
- . The X and Z tool path from A1 to B must be incremental or decremental.
- . The cutting depth U(d) and retraction amount of rough cut R(e) are modal codes. They remain valid until another value is specified.

G71 is applicable to the following four cutting types. They are all parallel to the Z-axis. Whether U and W are positive or negative (Fig. 3-30) is determined by the direction of tool path.

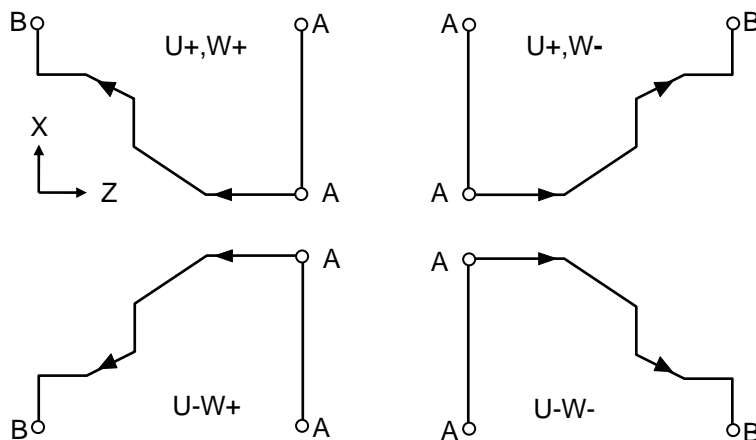


Fig. 3-30 G71 Rough Cut Canned Cycle

N160 M05 S0

N170 M02

2. Compound Canned Cycle, Traversed Rough Cut, G72

Format:

G72 W(d) R(e)

G72 P(ns) Q(nf) U(u) W(w) F(f) S(s) T(t)

T____ X____ Z_____ S_____

N(ns)

N(nf)

G70 P(ns) Q(nf) --- Fine cut (See the description above.)

When changing the tool for fine cut, the tool number T**** should be inserted in the line before N (ns). Repeated insertion of the number is allowed but the inserted T-codes (the tool numbers) must be same. The X and Z of the first block with the T-code indicate the safe position to which the tool retracts for changing. The other blocks that contain T-codes indicate the safe path to return to (ns). No blocks with T-codes will be executed for rough-cut.

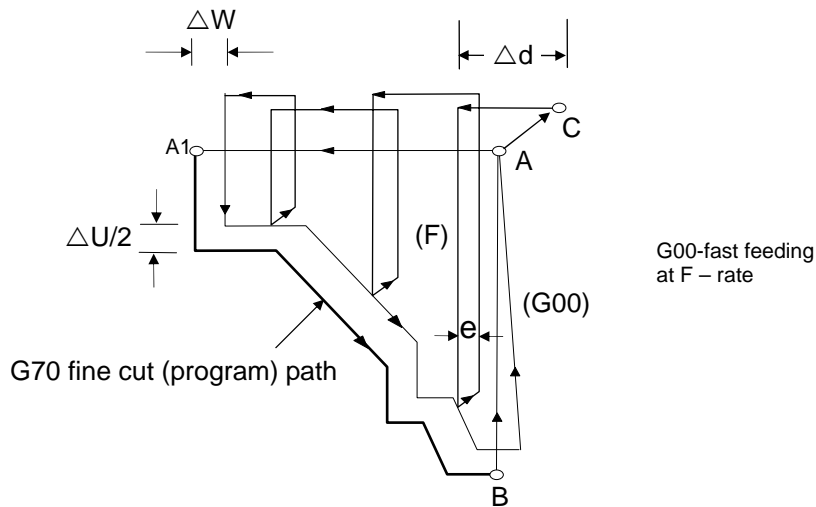


Fig. 3-32 Cutting Path of G72 Compound Canned Cycle

As shown in Fig. 3-32, all functions of G72 are same as G71, except that the cycle path is parallel to the X-axis.

- . N(ns)~N(nf) define the machining path of A1~B.
- . No assignment of positioning commands on X-axis is allowed from A to A1.
- . The feed-rate from A to A1 is either G00 or G01.

- The X and Z tool path from A1 to B must be incremental or decremental.
- No subprogram is available from N(ns) to N(nf).

G72 is applicable to the following four cutting types. They are all parallel to X-axis. Whether U and W are positive or negative (Fig. 3-33) is determined by the direction of tool path.

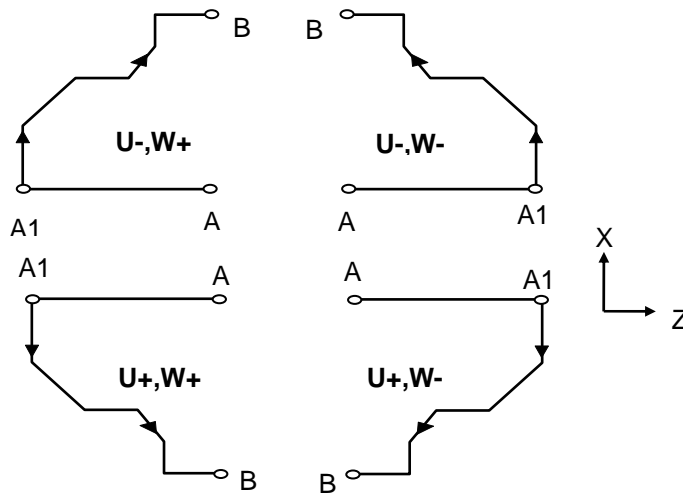


Fig. 3-33 Cutting Path of G72 Compound Canned cycle

Programming example of G72, G70 compound canned cycles:

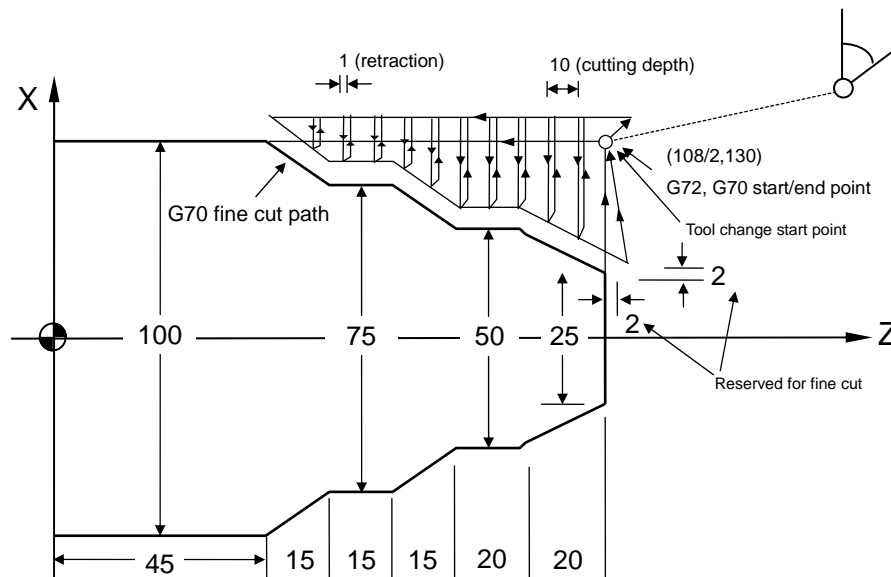


Fig. 3-34 Programming Example of G72, G70 Compound Canned Cycles

```
N10 G00 X108.000 Z130.000
N20 M03 S2000
N30 G72 W10.000 R1.000
N40 G72 P80 Q130 U4.0 W2.0 F3.00
```

```
N50 T101 X108.000 Z130.000
N60 T101 X____ Z____
N70 T101 X____
```



This box contains fine cut tool change commands. The blocks containing T-code are not executed for the rough-cut. However, they will be executed to change tools for fine cut. N60 and N70 that contain T-codes can be added to N80 [P(ns)] (fine cut start point) repeatedly. They are used for programming a save path from the safe position for tool change to the fine cut start point to remove obstacles, if any.

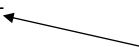
```
N80 G00 Z45.000
N90 G01 X75.000 W15.000 F1.50
N100 W15.000
N110 X50.000 W15.000
N120 W20.000
N130 X25.000 W20.000
N140 G70 P80 Q130
N150 M05 S0
N160 M02
```

3. Compound Contour Canned Cycle, Rough cut, G73

To save machining time, G73 is used to cut a work-piece that has been machined in a rough cut, forging or casting process and formed with a shape similar to the finished-product.

Format:

```
G73 U( i) W( k) R(d)
G73 P(ns) Q(nf) U( u) W( w) F(f) S(s) T(t)
T____ X____ Z____ S____
N(ns) . . . . .
      . . . . .
      . . . . .
```



When changing the tool for fine cut, the tool number T**** should be inserted in the line before N(ns). Repeated insertion of the number is allowed but the inserted T-codes (the tool numbers) must be same. The X and Z of the first block with the T-code indicate the safe position to which the tool retracts for changing. The other blocks that contain T-codes indicate the safe path to return to (ns). No blocks with T-codes are executed for the rough cut.

N(nf)

G70 P(ns) Q(nf) --- Fine cut (See the description above.)

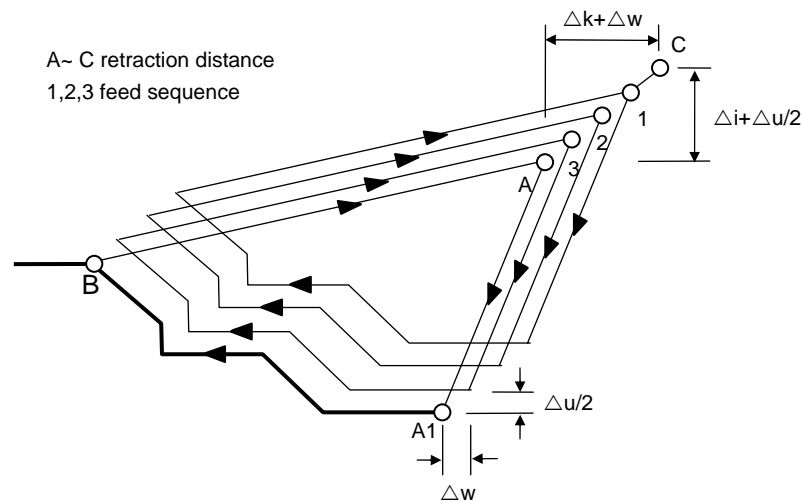


Fig. 3-35 Cutting Path of G73 Compound Canned Cycle

U(i) : Cutting amount on X-axis. (radius programming)

If not defined, the **parameter "G73 Total Cutting Amount "** is used.

W(k) : Cutting amount on Z-axis.

If not defined, the **parameter "G73 Total Cutting Amount"** is used.

R(d) : Rough Cutting Cycles

I.e. times of cuts required to reach the defined cutting depth on X and Z-axes. If not defined, the **parameter "G73 Cutting Cycles"** is used.

P(ns) : The first block number of a fine cut cycle.

Q(nf) : The last block number of a fine cut cycle.

U(u) : Amount of material to be removed for fine cut, X-axis.

W(w) : Amount of material to be removed for final cut, Z-axis.

F(f),S(s),T(t): F = feed-rate, S = spindle speed, T = tool selection.

The F, S, and T functions of G73 and previous blocks are applicable to G73, but all F, S, and T functions from N(ns) to N(nf) are not applicable to G73. They are only applicable to the fine cut command G70.

- N(ns)~N(nf) define the machining path of A A1 B.
- A maximum of 50 blocks can be inserted from N(ns) to N(nf).
- No subprogram is available from N(ns) to N(nf).
- The tool returns to A when the cycle finishes.
- The cutting amount U(i), W(k) and the cutting cycles R(d) are modal codes. They remain valid until another value is defined.

Programming example of G70, G73 compound canned cycles:

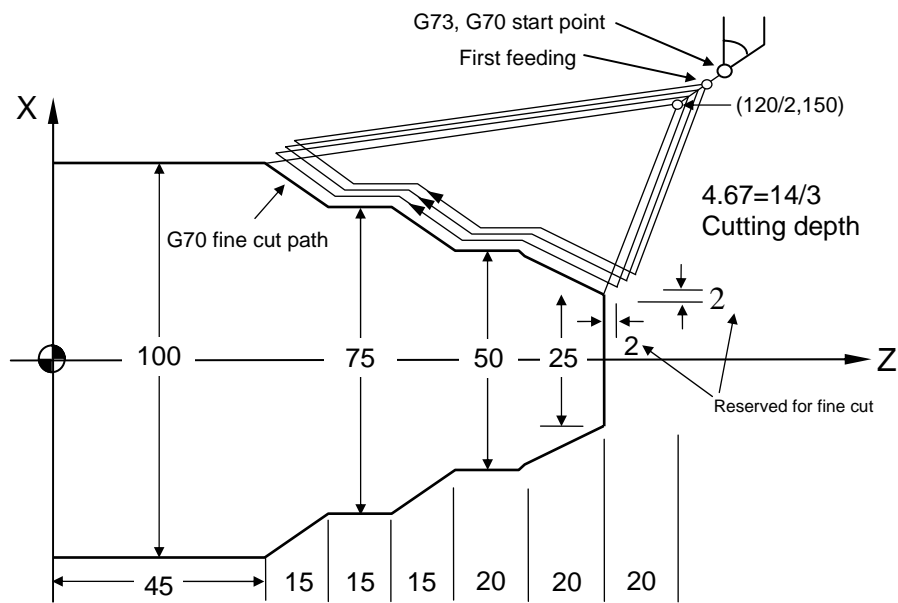


Fig. 3-36 Programming example of G70, G73 Compound Canned Cycles

```

N20 G00 X120.000 Z150.000
N30 G73 U14.000 W14.000 R3
N40 G73 P80 Q100 U4.000 W2.000 F3.00
N50 T101 X120.000 X150.000
N60 X___ Z___
N70 X___
N80 G00 X25.000 W-20.000
N90 G01 X50.000 W-20.000 F1.5
    
```

This box contains fine cut tool change commands. The blocks containing T-code are not executed for the rough-cut. However, they will be executed to change tools for fine cutting. N60 and N70 that contain T-codes can be added to N80 [P (ns)] (fine cut starting point) repeatedly. They are used to program a save path from the safe position for tool changes to the fine cut starting point to remove any obstacles.

N100 W-20.000
 N110 X75.000 W-15.000
 N120 W-15.000
 N130 G01 X100.000 W-15.000
 N140 G70 P80 Q100

4. Canned Cycle, Fine Cut, G70

After a work-piece undergoes rough cut with G71, G72 or G73, G70 is used for fine cut of the work-piece to ensure its precision.

Format:

G70 P(ns) Q(nf)

P(ns) : The number of the first block for a fine cut cycle.

Q(nf) : The number of the last block for a fine cut cycle.

The F, S, and T functions of G71, G72, G73 and previous blocks are applicable to G70. wherer F, S, or T is changed in the blocked from P(ns) to Q(nf), the changed values praveil. When G70 is executed, the tool returns to the start point and reads the next block.

5. Lateral Grooving Canned Cycle, G74

Format:

G74 Z(W)___ K(k) R(e) F ___

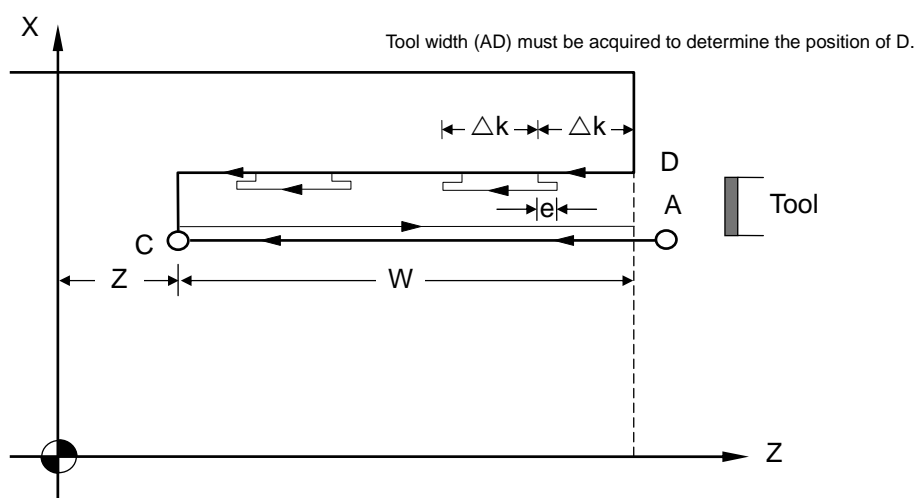


Fig 3-37 Cutting Path of G74 Lateral Grooving Canned Cycle

R(e) : The retraction amount after each cut of k on Z-axis.
If not defined, **parameter "G74, G75 Retraction Amount"** is used.

Z : Absolute coordinates of point C on Z-axis.

W : Incremental coordinates of points A~C on Z-axis.

K(k) : Cutting amount on Z-axis. (Integer μm only)

F : Feed-rate.

- . The retraction amount R(e) is a modal code. It remains valid until another value is defined.
- . To initialize axial drilling, X axis must be located in the center ($X=0$).

Example:

G0 X0. Z80. \Rightarrow Move tool quickly to the position X0. Z80. is relative to the work origin.

M03 S2000 \Rightarrow Positive rotation of spindle, speed 2000(rpm).

G74 Z30.K10.R3.F0.2 \Rightarrow Drilling canned cycle: "Z30" indicates that the drilling cycle ends at the absolute coordinate Z30. K10 indicates 10,000 (μm) per drilling. R3 indicates 3000 (μm) per retraction.

M05 S0 \Rightarrow Spindle stops.

M02 \Rightarrow Program ends.

- X : Absolute coordinates of Point C on X-axis.
- U : Incremental coordinates of points A~C on the X-axis
- K(k) : X-axis cutting amount. (Integer μ m with diameter programming)
- F : Feed-rate.

. The retraction amount R(e) is a modal code. It remains valid until another value defined.

Programming example of grooving canned cycle:

```
N10 G0 X80.0 Z50.0
N20 M03 S2000
N30 G75 X60. K5. R1. F0.5
N40 M5 S0
N50 M2
```

7. Compound Thread Cutting Canned Cycle, G76

G76 is easier to operate than G32, G92. Cutting threads with this function can save much time programming and calculating.

Format:

```
G76 P(m)(r)(a) Q( d min) R(d)
G76 X(U) _____ Z(W) _____ R(i) P(k) Q( d) F(l)
```

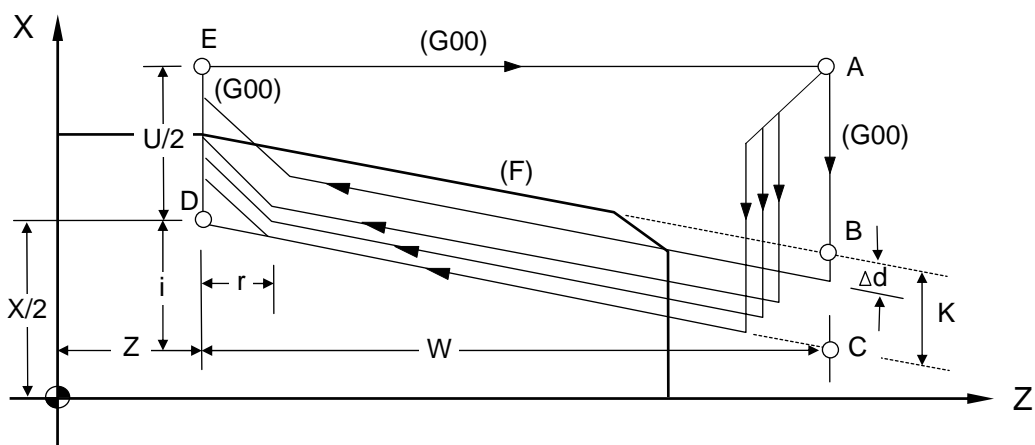


Fig. 3-40 G76 Compound Thread Cutting Canned Cycle

- m : Fine cut times (2-digit, 01~99)
If not defined, **parameter "G76 Fine Cut Times"** is used.
- r : Chamfering settings (2 digits)
Length of chamfering = $0.1 \times \text{chamfering settings (r)} \times \text{thread pitch}$.
If not defined, the **parameter "Chamfering Settings"** is used.
- a : Tool-tip angle (2 digits).
The available angles are $0^\circ, 5^\circ, 10^\circ, 15^\circ, \dots$ to 90° . If not defined, the **parameter "Tool-tip Angle"** is used.

m, r, and a are defined simultaneously by the command code P.

For $m=2, r=12, a=60^\circ$, then the command is G76 P021260.

- Q(d min): Minimum cutting amount (integer μm)
When the cutting amount of the nth cutting ($d\sqrt{n} - d\sqrt{n-1}$)
 $< d \text{ min}$, the cutting will resume with $d \text{ min}$ as the minimum
cutting amount. If no minimum cutting amount is defined, the
parameter "Minimum Cutting Depth" is used.

- R(d) : Amount of material to be removed for the fine cut
If not defined, the **parameter "Reserved Thread Depth"** is
used.

X, Z : Absolute coordinates of cutting end point (D).

U, W : Incremental coordinates of the cutting end point (D).

R(i) : Radius difference of thread part (i=0 indicates normal linear
thread cutting).

P(k) : Thread height (radius programming on X-axis, unit: integer μm)

Q(d) : First cutting depth (radius programming, unit: integer μm)

F(l) : Thread pitch, (same as G32)

- . What must be noted is that length of the path DE (U/2) must be greater than the length of the chamfer.
- . The fine cut times m, chamfering settings r, tool-tip angle a, minimum cutting amount Q (d min) and reserved thread depth R(d) are modal codes. They remain valid until another value is defined.

The workload of tool-tip will be relieved and a better thread cutting result will be obtained by applying G76 to execute single-side cutting

In Fig. 3-39,, the feed-rate between C and D is defined by F and fast feeding is applied to other paths. The (+)(-) values of the increments in Fig. 3-39 are as follows:

U, W : Negative (determined by the directions of AC and CD).

R : Negative (determined by the directions of AC).

Fig. 3-40 provides more information about the depth of each cut.

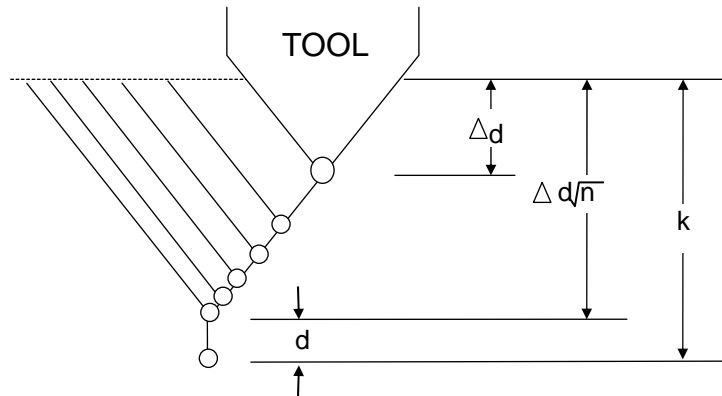


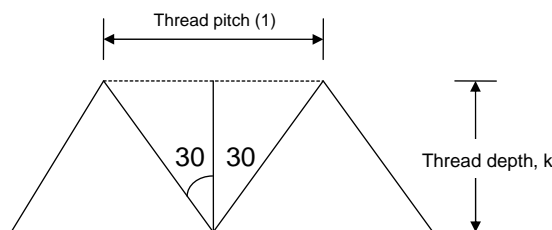
Fig. 3-41 Cutting Description

The thread height K is acquired from the thread pitch and the tool nose angle. The formula is:

$$\text{Thread height } k = (\text{pitch}/2) / \tan (\text{angle}/2)$$

$\tan (\text{angle}/2)$,acquired from the trigonometric table.

Ex: If tool nose angle $a=60^\circ$, Thread pitch $F(l)=2$ mm.



$$\begin{aligned} \text{Thread height } k &= (2/2) / \tan(60/2) \\ &= (1) / \tan 30 = (1)/0.5774 = 1.732 \end{aligned}$$

The first cutting depth (cutting amount) is d ,the n^{th} cutting depth is $d\sqrt{n}$, the cutting amount will decrease progressively every time. Note that n should not exceed 30. Otherwise, an alarm will be generated. In this case, please use the normal thread cutting.

- . When executing G70~G73, the serial numbers defined by P and Q should not be the same.
- . In G70, G71, G72, and G73, chamfering and R angle should not be used to terminate the last positioning command used for fine cut shaping blocks defined by P and Q.

3.13 Max. Spindle Speed Setting, G50

The setting function for the maximum spindle speed (G50) normally goes with setting function of the constant surface cutting (G96).

Format:

G50 S_____

S : Max. spindle speed (rpm or rev/min)

3.14 Constant Surface Cutting Speed Setting, G96

Format:

G96 S_____

S : Surface cutting speed (m/min)

The surface cutting speed refers to the relative velocity between the tool-tip and cutting point (on the surface) of the rotating work-piece. A tool has its advised surface cutting speed range for optimizing the cutting result. G96 is used to control the surface cutting speed. The relationship between the surface cutting speed, work-piece diameter and spindle rotation speed is expressed by:

$$V = DN$$

V : The surface cutting speed is the S value of G96.

D : Diameter of the surface is cut, m.

N : Spindle rotation speed, rev/min.

When the surface cutting speed is constant and the tool cuts the surface inwards, D will become lesser and N will become greater. Hence the max. rotation speed must be limited using G50 S___. Once this limit is reached, the speed will not increase any more.

Ex: N10 G50 S2000 . . . Max. rotation speed of the spindle is 2000 rpm.

N20 G96 S200 . . . The constant surface cutting speed is 200 m/min.

3.15 Constant Rotation Speed Setting (Constant Surface Cutting Speed Cancellation), G97

Format:

G97 S_____

This function maintains the spindle speed defined by S. It cancels the constant surface cutting speed at the same time.

3.16 Feed-rate Setting, G98, G99

G98 : Feed per minute, mm/min

G99 : Feed per revolution, mm/rev

The feed-rate F in H4T turner series is defined by G98 and G99. G99 is the default value. The conversion formula is:

$$F_m = F_r * S$$

F_m : Feed per minute, mm/min.

F_r : Feed per revolution, mm/rev.

S : Spindle speed, rev/min.

3.17 Inch/Metric Measurement Mode , G20, G21

Format: G20 -- System measurement in INCH mode

G21 -- System measurement in METRIC mode

3.18 Auxiliary Functions, M-code, S-code

The auxiliary function M-code is comprised of the letter M and 2 digits attached behind (M-codes for general). The range of the M-codes is 00 ~99 and different codes represents different functions as shown below:

Currently, H4CL-T Series provides the following M-codes:

M-CODE	Function
M00	Program Suspension.
M01	Option Suspension.
M02	Program End.
M30	Program Finished.
M98	Subprogram Call
M99	Subprogram ends or main program repeats.
M**	Customized M code (PLC)
M03	Spindle rotates in normal direction
M04	Spindle rotates in reversed direction
M05	Spindle stops
M08	Coolant ON.
M09	Coolant OFF.
M10	Spindle chuck unclamped
M11	Spindle chuck clamped
M15	Work -piece counter +1
M16	Clear work piece counter

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.

3.19 Subprogram

Where there are certain fixed programs or command groups in a main program that demand repeated execution, these commands could be saved in memory as subprograms, so that the main program could be designed with a simplified structure. Subprograms can be called out one after another in auto mode.

1. Structure of the Subprogram

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

PROGRAM 05 Subprogram number
 Content
 Content
 M99 Subprogram ends

If a subprogram is not called by the main program but executed directly by pressing “CYCST”, the program loops.

2. Execution of the Subprogram

Format:

M98 P _____ L _____

P : Subprogram number

L : Execution times of the subprogram. If not defined, the subprogram is to be executed 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 a second sub-prgrams. The H4CL-T Series controller provides a maximum of 5 levels stepwise calls:

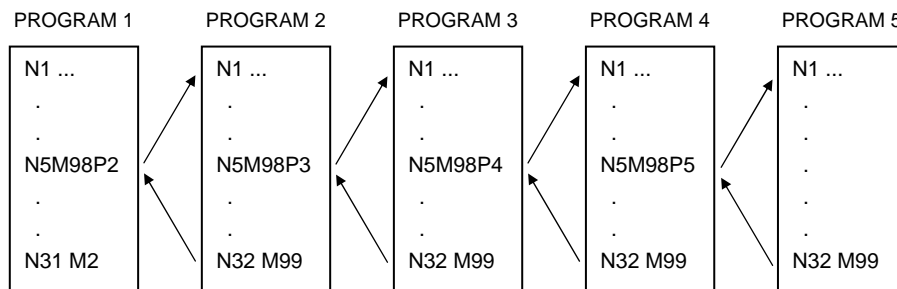


Fig. 3-43 Subprogram Stepwise Call

The M98 and M99 blocks should not contain any positioning commands, such as X..., Z... .

3.20 G40 G41 G42 Tool Radius Compensation

3.20.1 Total Offset Compensation Setting and Cancellation

Total offset compensation = Length compensation + Wear compensation

Format:

	Compensation Set	Compensation Cancel
Without Turret	T	T00
With Turret	T	T 00

: Compensation number, indicating which set of compensation data is to be applied.

: Tool number, indicating which tool is to be selected.

When a compensation number is selected, the control unit will simultaneously select the X-axis and Z-axis compensation values for tool length and wear compensation. These values are summed up for compensation of the tool path.

Any small differences between the cut work-piece and specifications found during the cutting test after the tool is calibrated with G10, can be remedied by wear compensation (referring to the Tool Wear Compensation page). If the difference is very small, positive values should be used. If the difference is large, negative values should be used.

Ex: N10 G01 X50.000 Z100.000 T0202

N20 Z200.000

N30 X100.000 Z250.000 T0200

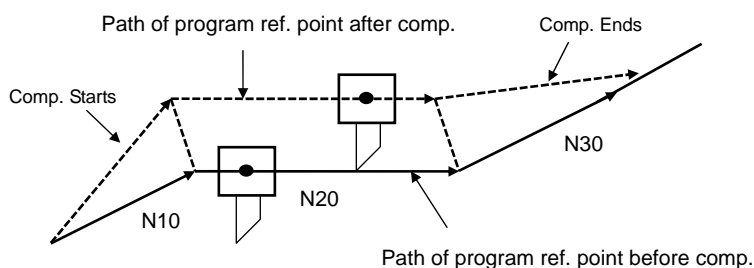


Fig. 3-44 Example of Tool Length Compensation

In this example, T0202(T202) indicates that the second tool and the second set of compensation data are selected. T0200 (or T200) indicates that the tool length compensation is cancelled.

Notes:

1. After powering the CNC, compensation is automatically cancelled and the compensation number is reset to "0" or "00".
2. Compensation must be cancelled to execute "Auto-CYCST".
3. The length compensation command "T-code" can form an independent block in the program without positioning definition. CNC executes an internal computation for compensation, but the tool does not execute any positioning movement.

As previously mentioned, the tool-tip of the disposable tool is arc-shaped, as Fig 3-45. The A point is usually the contact point between the tool-tip and work-piece when executing outer/inner diameter cutting along the Z-axis. B is the contact point when executing linear cutting along the X-axis. P point is the tangential intersection point of A and B, usually called a fictitious tool-tip for it doesn't actually exist. It is also called a program reference point for it moves along the programmed path when there is no radius compensation. In fact, this point is also a basis for programming.

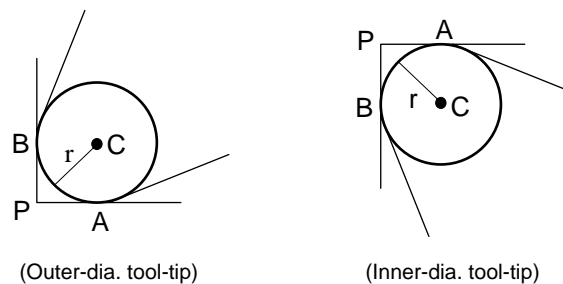


Fig. 3-45 An Arc Tool-tip

When executing outer or inner diameter cutting along the Z-axis, A and P are on the same path. When executing linear cutting along the X-axis, B and P are on the same path. Therefore, though the actual cutting points are A and B (the oblique line is not reachable), the cutting sizes are both correct, as shown in Fig. 3-46 (left). However, it would be very different when cutting an oblique line with 2 axes moving simultaneously. As Fig. 3-46 (right), P1~P2 is the fictitious tool-tip path (it is also the programmed path) and the actual cutting path is the tool-tip path CD along the arc. Obviously some part of the material is missed. This could

be remedied by radius compensation to position the cutting path from C~D to P1~P2. The amount of positioning is calculated by CNC internally.

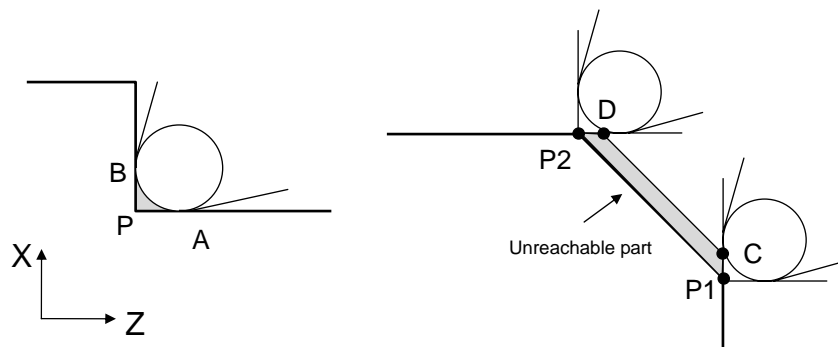


Fig. 3-46 Cutting Error without Radius Compensation (Taper)

Similarly, an error occurs during arc-cutting, as shown in Fig 3-47. It could also be remedied by radius compensation, with the amount of compensation calculated by the CNC.

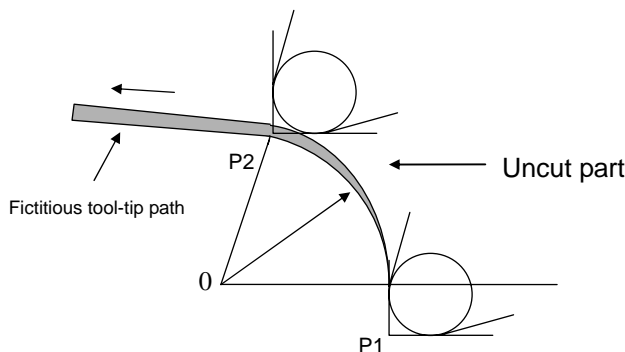


Fig. 3-47 Cutting Error without Radius Compensation (Arc)

3.20.2 Tool-tip Radius and Direction of Fictitious Tool-tip

When executing tool-tip radius compensation, the radius and arc must be accurate; otherwise, the cutting result will not be precise. Manufacturers of disposable tools always provide accurate tool-tip radius data. The radius data are to be entered in the "R" field on the Tool Length Compensation page (unit: mm).

Besides the tool-tip radius, the direction of fictitious tool-tips must be acquired (refer to the position of P in Fig. 3-45). The direction is defined by integers 0~9 (Fig. 3-48). In the system with tools on the top (rear) holder, the direction 3 is for outer diameter

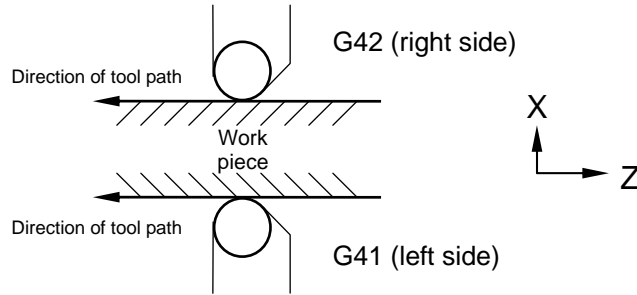


Fig. 3-49 G41 and G42 Applications

Initial Setting for Tool Radius Compensation:

When G41/G42 is executed, the tool moves in a linear motion to the X, Y coordinate defined in the G41/G42 block at G01 speed. When reaching the defined X, Y coordinates, the tool-tip deviates at a distance equivalent to the tool radius. The initial setting of the G41/G42 command is only available in the G00 or G01 linear cutting model. The system sends an error message if it is executed in the G02 or G03 arc-cutting mode. A simple description of the tool initial setting function is given below (assuming that the direction of the tool-tip is 0,9):

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 X_____ Z_____ F_____ T_____
N2 G41(G42) X_____ Z_____
N3 Z_____
```

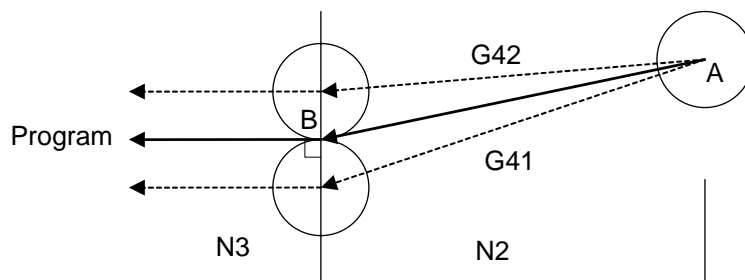


Fig. 3-50 Initial Setting of the Tool Radius Compensation – 1

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

```
N1 G01 X_____ Z_____ F_____ T_____
N2 G41(G42) X_____ Z_____
N3 G02 X_____ Z_____ I_____ K_____
```

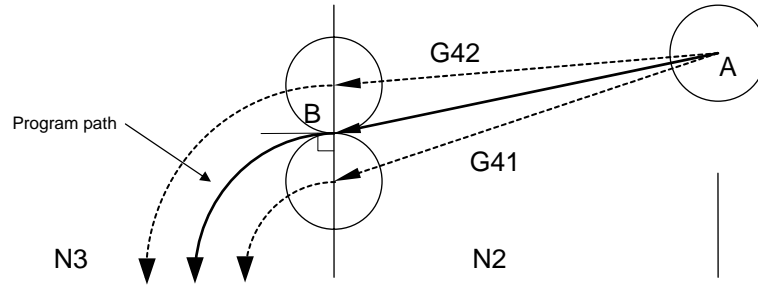


Fig. 3-51 Initial Setting of the Tool Radius Compensation - 2

Tool Radius Compensation Cancellation:

Once G41 or G42 is executed successfully, G40 command must be used to cancel the tool radius compensation. The movement for cancellation of the radius compensation can only be executed in the G00 or G01 mode. G40 is not directly available for blocks G02 and G03. Cancellation can only be effective after successful arc cutting. Below are some examples of the cancellation of tool radius compensation (assume the direction of tool-tip is 0,9):

1. N20 G41(G42)
- . . .
- . . .
- N31 G01 Z_____ F_____
- N32 G40 X_____ Z_____

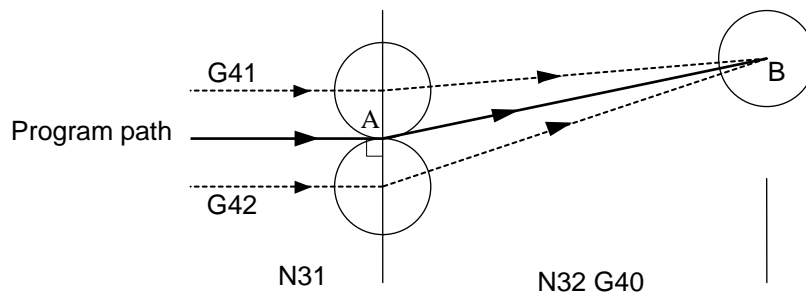


Fig. 3-52 Tool Radius Compensation Cancellation -1

```

2.  N10 G41(G42) .....
      . . .
      . . .
N15 G02 X___ Z___ I___ K___ F___
N20 G01
N25 G40 X___ Z___
    
```

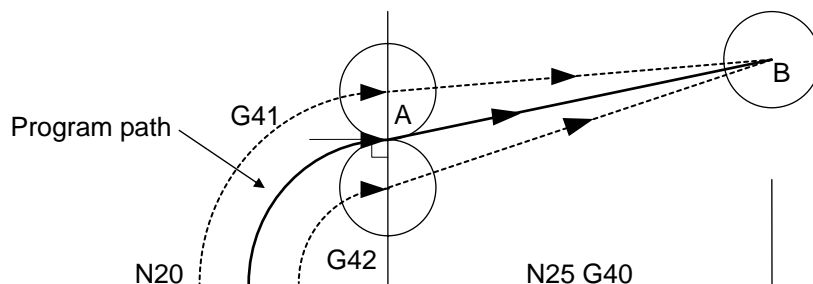


Fig 3-53 Tool Radius Compensation Cancellation - 2

G40, G41, G42 are modal codes. Once G41 (or G42) is set, do not set it again before using G40 to cancel the compensation.

3.20.4 Notes on Tool Radius Compensation

1. When radius compensation is executed, there should be at least one block containing positioning commands between two neighboring blocks. The following commands do not perform tool positioning, though they have mechanical actions. Therefore they are not allowed for continuous blocks .

M05 M-code output

S2100 S-code output

G4 X1.000 Suspension

G1 U0.000 Feed distance=0

G98 G-code only

2. Only G00 and G01 are applicable to blocks with tool-tip radius compensation. Arc commands G02, G03 are not allowed
3. The tool radius compensation function is not available for MDI operation.

4. Tool-tip radius compensation is not allowed for G74, G75, or G76.
5. When cutting an inside corner, the arc radius R of the inside corner must be equal or greater than the tool radius (r). Otherwise an alarm is generated. Cutting and outside corner of an arc is not subject to this regulation.
6. If the corner is shaped like an arrowhead (Fig. 3-54), outside corner cutting will be correct. The correctness of cutting on the inside corner depends on the length of opening C . If the length is less than the tool diameter, no cutting is possible and an over-cutting alarm is generated. If it is greater than the tool diameter, the tool cuts toward the inside corner. However, part of the work-piece cannot be cut in the sharp inside corner. .

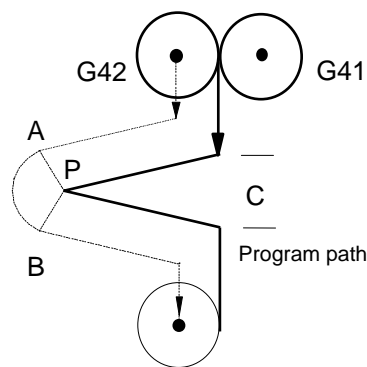


Fig. 3-54

7. When cutting a stepwise work-piece with a step value smaller than the tool radius, an over-cutting alarm is generated as shown in Figure 3-55.

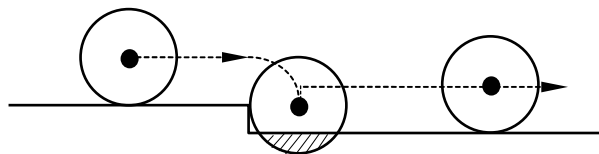


Fig. 3-55 Over-cutting (Shaded Area)

Tool-tip Radius Compensation Example:

Tool number = 02, tool-tip direction = 3, tool-tip radius = 1.5 mm. The X-axis coordinate is defined by the diameter.

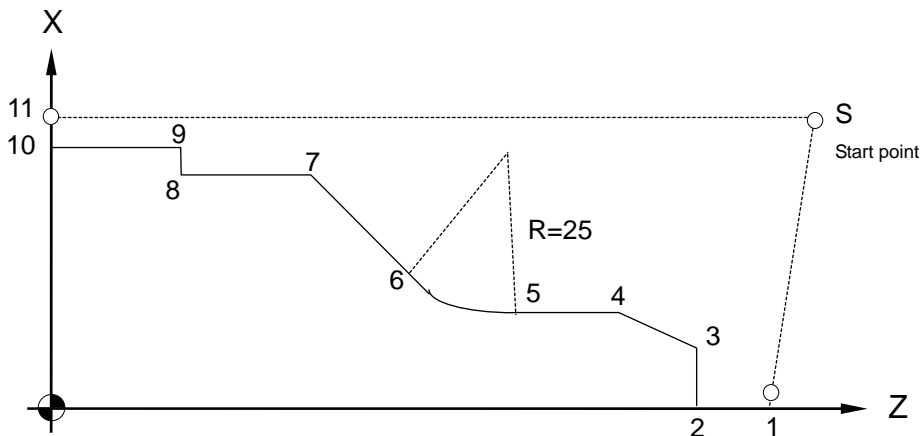


Fig. 3-56

N10 G0 X100. Z120.	. . . Point S
N20 G0 X0. Z110.	. . . Point 1
N30 M3 S2000	
N40 G42 Z100. T02 F3.0 insertion	. . . Point 2, compensation
N50 G1 X20.	. . . Point 3
N60 X30. Z91.34	. . . Point 4
N70 Z75.	. . . Point 5
N80 G02 X44.644 Z57.322 I25. F1.5	. . . Point 6, arc cutting
N90 G01 X76. Z37.644 F3.0	. . . Point 7
N100 Z20.	. . . Point 8
N110 X80.	. . . Point 9
N120 Z0.	. . . Point 10
N130 G40 X90. cancellation	. . . Point 11, compensation
N140 G0 X100. Z120.	. . . Point S
N150 M05	
N160 M02	

An over-cutting alarm is generated if you try to return to Point S directly from Point 10. This is because the angle of 9-10-S is too sharp. The alarm is also generated if the radius compensation is greater than 2.0 mm, which is the distance from 8 to 9.

4 MCM Parameters

4.1 MCM Parameters

The MCM parameter setting function allows the user to define controller system constants according to mechanical specifications and machining conditions.

These parameters are classified into two groups: basic parameters and MCM parameters.

4.1.1 Basic Parameters

Quickly press the

I/O
MCM

 key twice to enter the parameter setting screen, as shown below:

PARAMETERS	X-AXIS	Z-AXIS
JOG Feed	1000	1000
CNT. Limit		0
Collet Type		0
OP-SKIP Funtion 0:OFF 1:ON		0
Feeder Delay Time (10ms)		50
Collet Close Delay Time (10ms)		50
Program Block Start Number		0
Inc. for Program Block Number		0
		PAGE ⌵
WEAR	OFFSET	MCM

Fig 4-1

PARAMETERS	X-AXIS	Z-AXIS
G71,G72 Cutting depth	0.000 inch	0.000 inch
G71,G72 Retracion	0.000	
G73 Total cutting depth	0.000 inch	0.000 inch
G73 Cutting cycle	0	
G74,G75 Retracion	0.000	
G76 Finish cutting cycle	0	
Chamfer	0	
Tool-tip angle	0	
Min. cutting depth	0.000	inch
Depth for finish cut	0.000	inch
		⌶ PAGE
SYS-MCM	VERSION	ERR-MSG

Fig 4-2

4.1.2 MCM Parameters

The correct and proper setting of these parameters is important for 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 successfully set

Press System Parameter on the second page of the basic parameters to enter the system parameter page below:

(A password is required when the password protection function is active)

PARAMETERS	X-AXIS	Z-AXIS
Resolution-Den.	100	100
Resolution-Num.	100	100
Encoder-factor (1,2,4)	0	0
Traverse Speed	100000	100000
Rotate direction	0	0
Home Speed-1	2500	2500
Home Speed-2	40	40
Home direction	0	0
Software OT (+)	9999.999	9999.999
Software OT (-)	-9999.999	-9999.999
Backlash Comp	0	0
Pitch Error Comp. (0/1/-1)	0	0
Segment Length for Pitch Err.	20.000	20.000

					PAGE
<input type="checkbox"/>	Change the password			Pitch Err.	

Fig 4-3

PARAMETERS	X-AXIS	Z-AXIS
Home Limit 0:NC 1:NO	0	0
MPG Den.	100	100
MPG Num.	100	100
U,W Max (RUN)	0.000	0.000
Acc. / Dec. mode	0	
G00 Acc. / Dec. Time	100	
G99 Acc. / Dec. Time	100	
Spindle Acc. / Dec. Time	100	
MPG Acc. / Dec. Time	64	
Spindle Numbers	1	
G83 Clearance	0.000	

					↑ PAGE
<input type="checkbox"/>	Change the password				

Fig 4-4

PARAMETERS	SP. 1	SP.2	SP.3
SP. Control Mode	0	0	0
SP. Encoder Factor	4	4	4
SP. Encoder Pulse	4096	4096	4096
SP. Acc. / Dec. Time	100	100	100
SP. RPM at 10V	3000	3000	3000
SP. Rotate Dir.	0	0	0

WHEN SPINDLE CHANGE BACK TO AXIS			
PARAMETERS	SP. 1	SP.2	SP.3
Rotate Dir.	0	0	0
Acc. / Dec. Time	100	100	100
Max. Speed	1000	1000	1000

					↕ PAGE ↘
Change the password					

Fig 4-5

PARAMETERS	
Graphic Mode , Z-axis Scale	0.000
=0 Radius,=1 Diameter setting	0
Tool Numbers	0
Tool CW Time (10ms)	50
Tool CCW Time (10ms)	50
LUB. Interval Time (10ms)	50
LUB. On Time (10ms)	50
U232 Device ? 0:No 1:Yes	0
Control Mode 0:Panel 1:Key	0
Panel Type 0:SW 1:Key	0

					↕ PAGE ↘
Change the password	BN-MCM	CLEAR ALL-PGM	LD-MCM	CLEAR OFFSET	

Fig 4-6

X-axis, compensation amount each segment							
01	0.000	11	0.000	21	0.000	31	0.000
02	0.000	12	0.000	22	0.000	32	0.000
03	0.000	13	0.000	23	0.000	33	0.000
04	0.000	14	0.000	24	0.000	34	0.000
05	0.000	15	0.000	25	0.000	35	0.000
06	0.000	16	0.000	26	0.000	36	0.000
07	0.000	17	0.000	27	0.000	37	0.000
08	0.000	18	0.000	28	0.000	38	0.000
09	0.000	19	0.000	29	0.000	39	0.000
10	0.000	20	0.000	30	0.000	40	0.000

					↕ PAGE
Z-AXIS					

Fig 4-7 X-axis Ball Screw Error Compensation

Z-axis, compensation amount each segment							
01	0.000	11	0.000	21	0.000	31	0.000
02	0.000	12	0.000	22	0.000	32	0.000
03	0.000	13	0.000	23	0.000	33	0.000
04	0.000	14	0.000	24	0.000	34	0.000
05	0.000	15	0.000	25	0.000	35	0.000
06	0.000	16	0.000	26	0.000	36	0.000
07	0.000	17	0.000	27	0.000	37	0.000
08	0.000	18	0.000	28	0.000	38	0.000
09	0.000	19	0.000	29	0.000	39	0.000
10	0.000	20	0.000	30	0.000	40	0.000

						⬆ PAGE
X-AXIS						

Fig 4-8 Z-axis Ball Screw Error Compensation

SP. 1 PARAMETERS	
SPINDLE DIRECTION	0
FIND GRID ? 0:Yes 1:No	0
GRID DIRECTION	0
IN POSITION DIRECTION	0
GRID SPEED	1
IN POSITION SPEED	1
IN POSITION ANGLE	0.00
SPINDLE HOME SENSOR	0
SPINDLE HOME SPEED	0

						⬆ PAGE ⬇
Change the password						

Fig 4-9 Server Spindle Parameter Setting - (1)

SP. 2 PARAMETERS	
SPINDLE DIRECTION	0
FIND GRID ? 0:Yes 1:No	0
GRID DIRECTION	0
IN POSITION DIRECTION	0
GRID SPEED	1
IN POSITION SPEED	1
IN POSITION ANGLE	0.00

						⬆ PAGE ⬇
Change the password						

Fig 4-10 Server Spindle Parameter Setting - (2)

When setting=1 and input I05 is ON, a program block containing “/1” will be skipped.

6. M12 feeding-delay setting

Format=□□□□(Default= 50) unit:10ms

M12 is the feeding command code. The countdown will be initiated after the PLC receives the M12 command code, and once the countdown is finished, the output 0011 (0011 feed output) will turn to ON.

The command is for feeder. The feeding movement varies among manufacturers; this description is for reference only.

7. Chuck release and return delay setting

Format=□□□□(Default=50) unit:10ms

Condition: Auto/Semi-auto mode → the command is valid only in auto mode

The countdown will be initiated when the PLC receives the Chuck release command code M10. After the countdown is finished the command is executed.

8. The row number automatically generates the initial value.

Set the starting number that is automatically generated by the single-section serial program number .

Format=□□□□ (Default= 0)

9. The row number automatically generates the partition value.

Set the increasing number between single-section serial program number .

Format=□□□□ (Default= 0)

10. X-axis resolution denominator setting.

11. X-axis resolution numerator setting.

12. Z-axis resolution denominator setting.

13. Z-axis resolution numerator setting.

The numerator or denominator of the resolution is set based on the specifications of the mechanical axial gearing (such as ball screw) and the encoder pulses of the servomotor. **No unauthorized change of setting is allowed.**

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

Example:

X-axis ball screw pitch = 5.000mm

Motor Encoder = 2500 pulses, multiplication factor = 4

Gear Ratio= 5:1(Servomotor rotates 5 turns = ball screw Rotates 1 turn)

$$\begin{aligned} \text{Resolution} &= \frac{5000}{2500 \times 4} \times \frac{1}{5} \\ &= \frac{1}{10} \end{aligned}$$

X-axis resolution denominator= 10

X-axis resolution numerator= 1

14. X-axis resolution multiplication factor

15. Z-axis resolution multiplication factor

Format= (Default=4)

Setting= 1, encoder pulses multiplied by 1.

Setting= 2, encoder pulses multiplied by 2.

Setting= 4, encoder pulses multiplied by 4.

Only one factor is selected at a time.

Note:

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

Ex: When the encoder pulses at 2000 rpm, and the multiplication factor is set to 2, the pulses are $2000 \times 2 = 4000$.

16. X-axis maximum feed rate setting.

17. Z-axis maximum feed rate setting.

Format= (Default =10000) Unit: mm/min

Note: The setting is in integer format.

Ex: If you set the Z-axis to 5000, it indicates that your highest feed rate is 5000 mm per minute.

The maximum feed rate is calculated as follows:

$$F_{\max} = 0.95 \times \text{RPM (the maximum rpm of the servomotor)} \times \text{Pitch (ball screw pitch)} \div \text{GR}$$

Ex: The highest rotation speed of the servomotor on X-axis is 3000 rpm, the ball screw pitch is 5mm.

GR = 5:1 (Servomotor rotates 5 turns = ball screw Rotates 1 turn)

$$F_{\max} = 0.95 \times 3000 \times 5 \div 5 = 2850$$

Recommended setting: 2850

18. X-axis, direction of motor rotation.

19. Z-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)

Machine design varies among manufacturers, and the position of the servomotor on the ball screw varies too. Parameters can be used to adjust the direction if the machine moves in opposite direction after the servomotor is installed. The parameters affect the direction of the machine coordinates.

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 decoder 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 feedback error. 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-)

20. X-axis homing speed when tool returns to machine origin 1.

21. Z-axis homing speed when tool returns to machine origin 1.

Format=□□□□ (Default=2500), Unit: mm/min

22. X-axis homing speed when tool returns to machine origin 2.

23. Z-axis homing speed when tool returns to machine origin 2.

Format=□□□□ (Default=40), Unit: mm/min

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 the limit switch \geq (FDCOM \times ACC) \div 60000

Note: (FDCOM = Homing velocity 1
 (ACC = G01 acceleration/deceleration time
 (60000 msec (60 secs \times 1000 = 60000 msec)

Ex: FDCOM = Homing velocity 1 = 3000 mm/ min
 ACC acceleration/deceleration time = 100 ms
 Minimal length of limit switch = 3000 \times 100 / 60000 = 5 mm

24. X-axis homing direction when tool returns to machine origin.
 25. Z-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

26. X-axis software OT Limit in (+) direction setting.
 27. Z-axis software OT Limit in (+) direction setting.

Format=□□□□.□□□ (Default=+9999.999), Unit: mm.

The set value is the positive distance between software OT and the machine origin.

28. X-axis software OT Limit in (-) direction setting.
 29. X-axis software OT Limit in (-) direction setting.

Format=□□□□.□□□ (Default= -9999.999), Unit: mm.

The setting value is the negative distance between software OT and the machine origin.

The concept and description of OT limit:

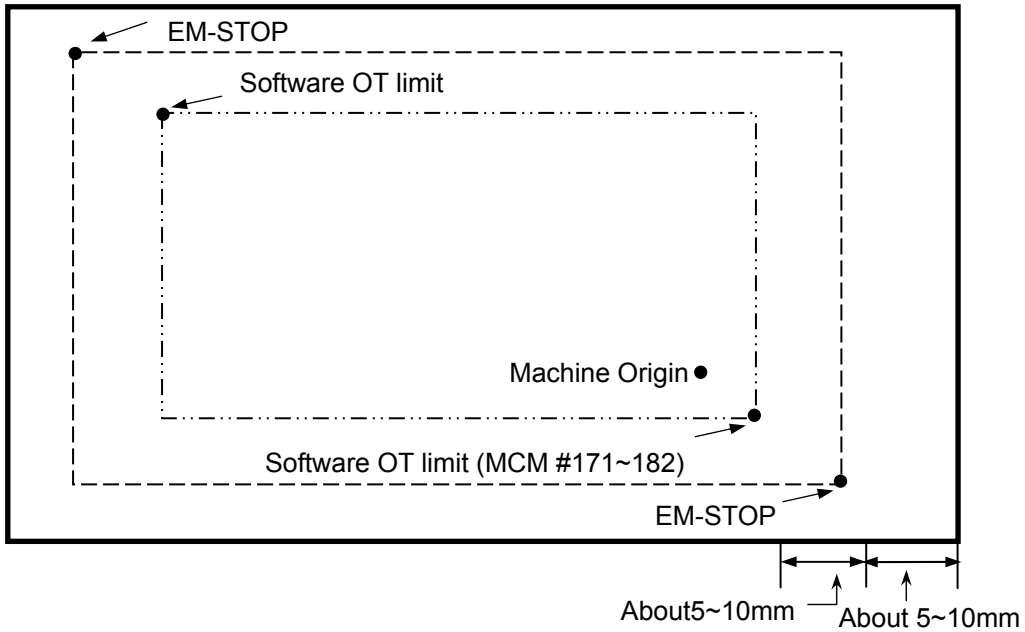


Fig 4-11

30. X-axis backlash compensation setting.

31. Z-axis backlash compensation setting.

Format=□.□□□ (Default 0), Unit: mm

32. X-axis ball screw pitch error compensation setting.

33. Z-axis ball screw pitch error compensation setting.

Format=□□ (Default=0)

Setting = 0, compensation cancel

Setting = -1, negative compensation

Setting = 1, positive compensation

34. X-axis segment length of pitch error compensation.

35. Z-axis segment length of pitch error compensation.

Format=□□□□.□□□ (Default=20.000), Unit: mm

Press the "Ball Screw Compensation" soft key to set the compensation for each segment (max. 40 segments).

36. X-axis origin signal format setting.

37. Z-axis origin signal format setting.

Format= (Default=0)

Setting = 0, Origin signal format: NC

Setting = 1, Origin signal format: NO

38. X-axis MPG hand-wheel resolution denominator setting.

39. X-axis MPG hand-wheel resolution numerator setting.

40. Z-axis MPG hand-wheel resolution denominator setting.

41. Z-axis MPG hand-wheel resolution numerator setting.

Format= (Default=100), See the following examples.:

Example: Resolution denominator=100

Resolution numerator=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 = $100 \times (100 \div 100)$

= 0.1 mm

= 100 μ m

Example: Resolution denominator=200

Resolution numerator=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 = $100 \times (500 \div 200)$

= 0.250mm

= 250 μ m

42. Maximum "U" value during execution.

43. Maximum "W" value during execution.

Format=□□.□□□(Default: 0.000)

During execution, only increments are allowed for change of tool compensation data. These settings define the maximum input value. Setting = 0.000 indicates that no change of tool compensation data is allowed during execution.

44. Accel./decel. type.

Format=□, Default=0

Setting = 0, Linear type. Setting = 1, "S" curve.

When the setting = 1, all linear types are replaced by "S" curve types.

45. G00 accel./decel. time setting.

Format=□□□(Default=100), Unit: msec

Setting range: 4~512 ms.

When the accel./decel. type is set to 0, the G00 accel./decel. type is linear.

When the accel./decel. type is set to 1, the G00 accel./decel. type is "S" curve. In this case, the setting remains same as the linear type, but the actual acceleration/deceleration time is the double of the setting.

46. G99 accel./decel. time setting.

Format=□□□ (Default=100), Unit: msec

Setting range: 4 ~ 1024 msec. 100 msec is recommended for both G00 and G01.

47. Spindle accel./ decel. time setting.

Format=□□□□ (Default=100) ; Unit: msec

48. MPG hand-wheel accel./decel. time setting.

Format=□□□ (Default=64), Unit: msec

Setting range: 4~512 ms.

49. Number of spindles in use setting

Format=(Default=1). Maximum=3.

First spindle → SPINDLE adapter

Second spindle → Y-AXIS adapter

Third spindle → D/A holder

Condition: when the first spindle is the server spindle (angle controlled by first spindle), one spindle at a time.

50. Drilling canned cycle buffer distance setting

Format=. (Default=0.000)

Execute the G83 drilling command to make the respective axis adopt the feeding buffer distance setting of G01 from G00 fast positioning.

51. Spindle control methods

Format=(Default=0)

Setting= 0, Open-loop AC motor control.

Setting= 1, Closed-loop spindle positioning control (working with the spindle motor)

52. Spindle encoder amplification factor setting

Format= (Default=4)

Setting = 1, encoder pulses multiplied by 1.

Setting = 2, encoder pulses multiplied by 2.

Setting = 4, encoder pulses multiplied by 4.

Only one factor is selected at a time.

53. Spindle encoder pulses setting.

Format= (Default = 4096)

Setting = encoder pulses × spindle encoder amplification factor

54. Spindle accel./ decel. time setting.

Format= (Default=100) ; Unit: msec

Setting range: 4~3000 ms

55. Spindle motor rpm at output voltage = 10V

Format= (Default=3000), Unit: RPM

The spindle motor rpm is calculated as follows:

$$\text{Motor RPM} = \frac{F \times 60 \times 2}{P} \quad \begin{array}{l} F : \text{Inverter Frequency} \\ P : \text{Poles (A regular AC motor has 4 poles)} \end{array}$$

There is a 2%~3% slip when the motor operates at the highest rpm. The actual slip depends on design of the motor.

56. Spindle rotating direction settings.

Format= (Default=0)

Setting=0 , CW

Setting=1 , CCW

57. Spindle rotating direction when switching to standard axial control

Format= (Default= 0)

Setting=0, CW

Setting=1, CCW

58. Spindle accel./ decel. time when switching to standard axial control

Format=, (Default=100), Unit: (msec).

Setting range= 4~3000 ms.

59. Spindle maximum rpm when switching to standard axial control

Format=,(Default=1000), Unit: RPM.

Setting range= 1~nominal rpm

60. The ratio of the horizontal axis (Z axis) in graph mode

Format=. ,

61. Radius or diameter programming.

Format= (Default=0)

Setting= 0, X-axis is radius programming.

Setting= 1, X-axis is diameter programming.

62. Set number of blades used.

Format=, Default= 0

This function cooperates with the use of a blade switching component for a maximum of 8 blades.

Since the blade switching component for each company varies, this setting is for reference only.

63. CW rotation time for tool search setting

Format=□□□□, Default 50, Unit:10ms.

Please pause this setting once the tool is found in CW rotation, and then execute CCW rotation.

The design varies among manufacturers; the description is for reference only.

64. CCW rotation time for tool search setting

Format=□□□□, Default 50, Unit:10ms.

Setting for CCW rotation time

The design varies among manufacturers; this description is for reference only.

65. Lubricant interval setting

Format=□□□□□□, Default 18000, Unit:10ms.

66. Lubrication time setting

Format=□□□□□□, Default 1000, Unit:10ms.

67. Use of U232 storage device.

Format=□ (Default = 0) [U232 is **optional**]

Setting=0 , U232 storage device is not used.

Setting=1 , U232 storage device is used.

When setting =1 (U232 storage device is used),U232 will be connected to the controller via RS232. When the startup procedure of U232 is completed, a "U232" function softkey appears on the JOG screen.

68. Mode selection setting 0: Auxiliary panel; 1: key

Select an operation mode for the controller,

Setting=0 , Select the operation mode from the auxiliary panel; the mode key is invalid.

Setting=1 , Select the operation mode with the mode key.

69. Auxiliary panel type setting 0: knob; 1: key

Setting=0, The auxiliary panel is operated with knobs (5 knobs)

Setting=1, The auxiliary panel is operated with keys (only an operation mode key is available)

G71 ~ G76 compound cutting parameters: See Chapter III "G/M Codes"

"Time – 1~4" and "Number of Tools": See Chapter VII "Attachment".

The parameters below are applicable to the close-loop spindle positioning control.

70. Spindle rotating direction settings.

Format= (Default=0)

Setting=0 , CW

Setting=1 , CCW

71. Spindle homing grid settings.

Format=, (Default=0)

Setting=0 , the spindle searches for grid.

Setting=1 , the spindle does not search for grid.

72. Spindle homing grid direction settings.

Format= (Default=0)

Setting=0 , CW

Setting=1 , CCW

73. Rotation direction setting for spindle positioning.

Format= (Default=0)

Setting=0 , CW

Setting=1 , CCW

74. Spindle homing grid rpm setting.

Format= (Default=1)

75. Spindle positioning rpm setting.

Format= (Default=1)

76. Spindle positioning angle in the JOG mode.

Format=□□□.□□ (Default=0.000)

77. Value to be displayed per rotation of the spindle Format=□□□□□

(Default=0).

This parameter is only available at the Recommended value of 360000

78. Spindle homing origin signal setting

Format=□ (Default=0) (First spindle only)

Setting=0, first spindle searches for grid directly

Setting=1, First spindle searches for external input signal (I022 and I023 are on) before searching for grid

79. Spindle homing origin signal rpm setting

Format=□□□□ (Default 0) (First spindle only)

Setting for spindle searching origin signal (I022 and I 023) rpm

5.2 System Installation

5.2.1 Operating Environment

The control unit used for the H4CL-T 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 / 0.1×10⁻⁶ sec.
per 0.01 second.
- * Others

Please consult us before installing the system in an environment exposed to a high degree of dust, coolant, or organic solution.

5.2.2 Notes on the Control Unit Case Design

- * A sealed case with dust proof 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 100 mm apart. In a magnetic field environment, this distance should be increased to at least 300 mm.
- * Refer to the operator's manual for servo driver mounting information.

5.2.3 Thermal Case 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 the case is shown as follows:

- (1) With a cooling fan, the allowable temperature rise is 1 °C /6 W/1 m².
- (2) Without a cooling fan, the allowable temperature rise is 1 °C /4 W/1 m².

This means that a case with a heat transfer area of 1 m² 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.

Ex 1: (With a cooling fan)

Heat Transfer Area = 2 square meter.

Allowable Internal Temperature Rise = 10 °C.

Maximum Allowable Heat Source = 6W×2×10 = 120W.

Other cooling devices, such as the cooling fin, must be incorporated for calculation if the heat source is more than 120W.

Ex 2: (Without a cooling fan)

Heat Transfer Area = 2 square meters.

Allowable Internal Temperature Rise = 10 °C.

Maximum Allowable Heat Source = 4W×2×10 = 80W.

Other cooling devices, such as the cooling fins, must be incorporated for calculation if the heat source is more than 80W.

5.2.4 H4CL-T External Dimensions

* H4CL-T Panel

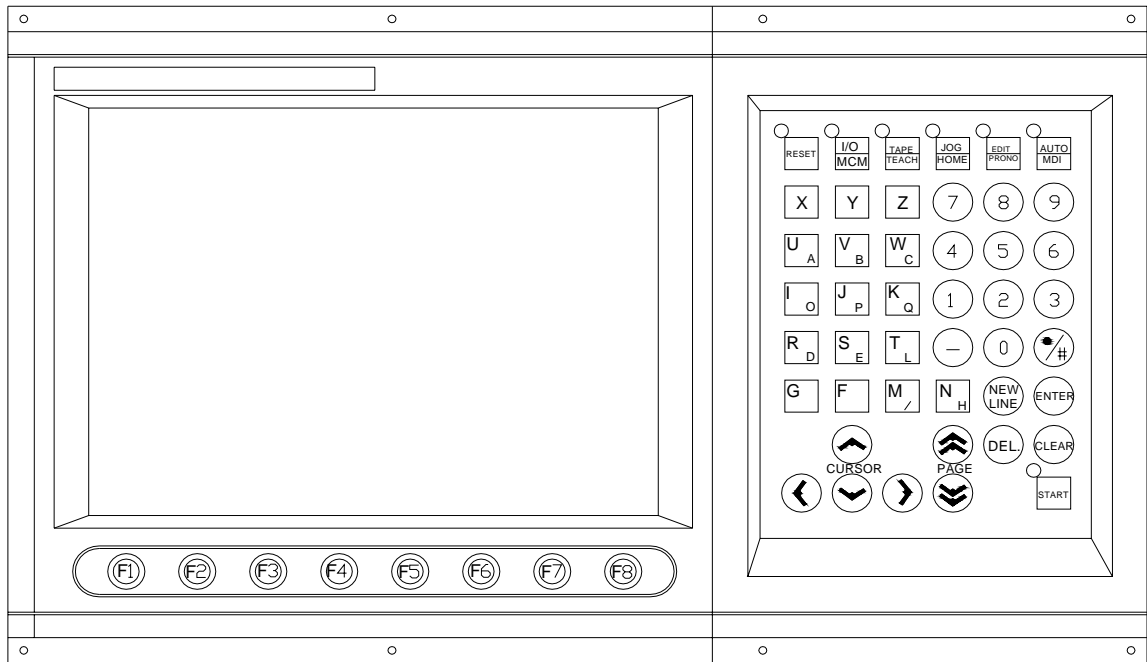


Fig. 5-2 H4CL-T Control Unit Panel

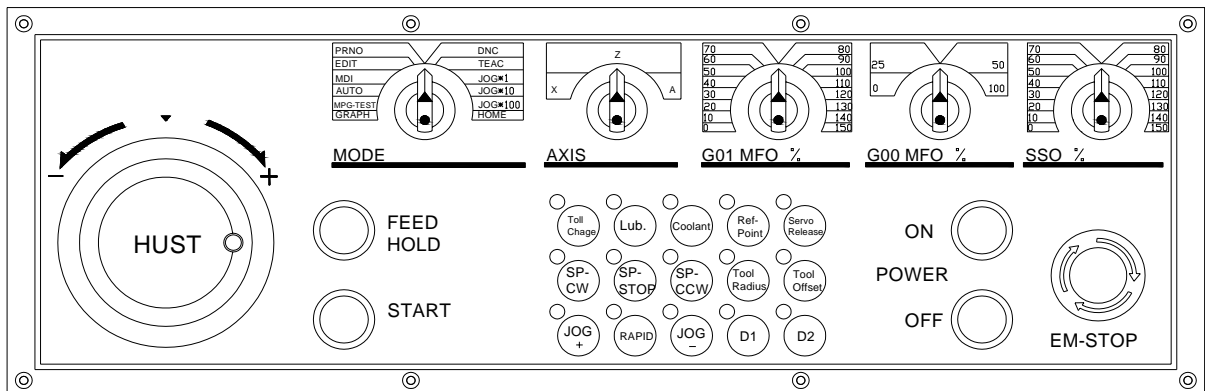


Fig. 5-3 H4CL-T MDI Panel

* H4CL-T CPU Main Board Connectors (Rear View)

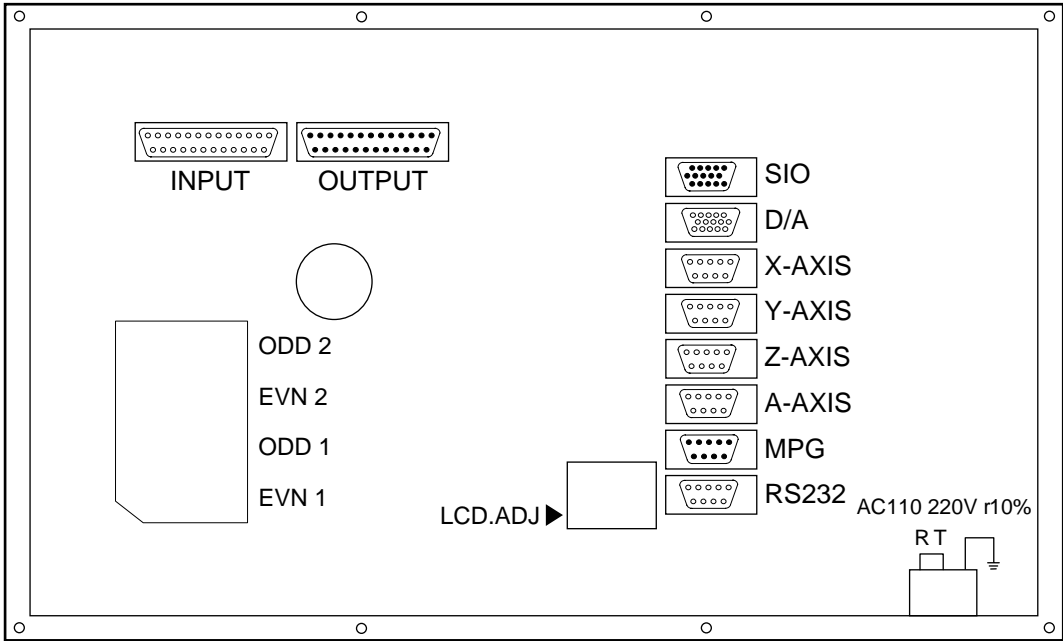


Fig. 5-4 H4CL-T CPU Main Board Connectors

* H4CL-T Series Case Dimensions (Rear view)

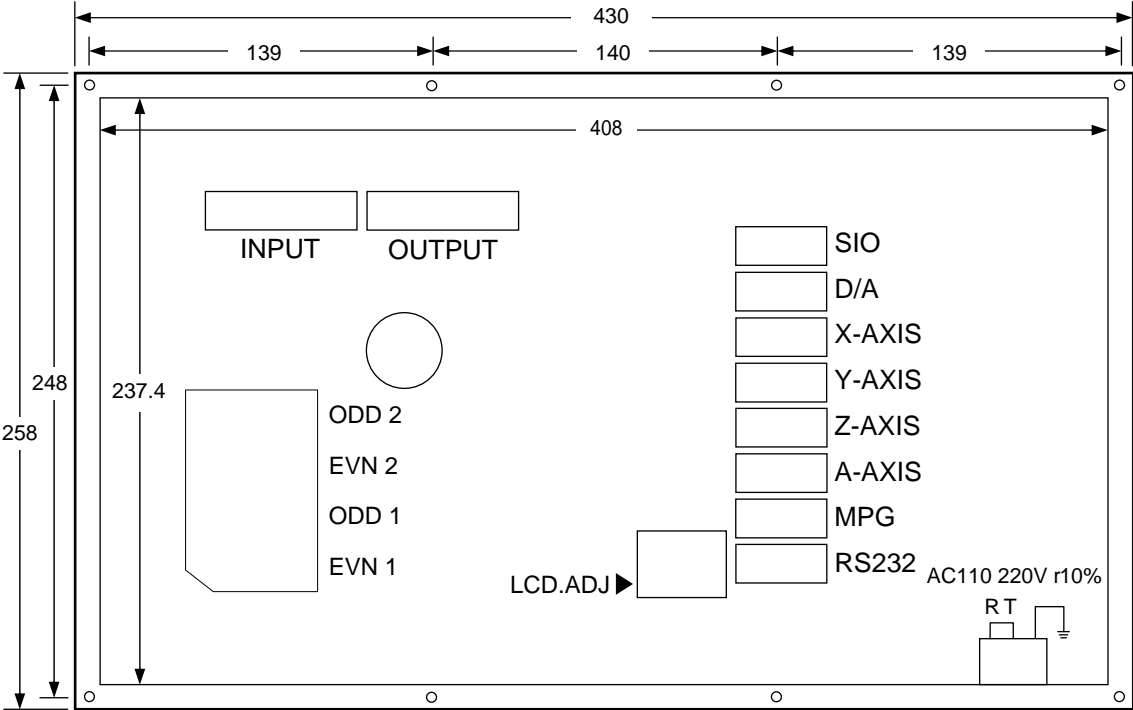


Fig. 5-5 H4CL-T Control Unit Dimensions (Rear View)

* **H4CL-T Series Cutout Dimensions**

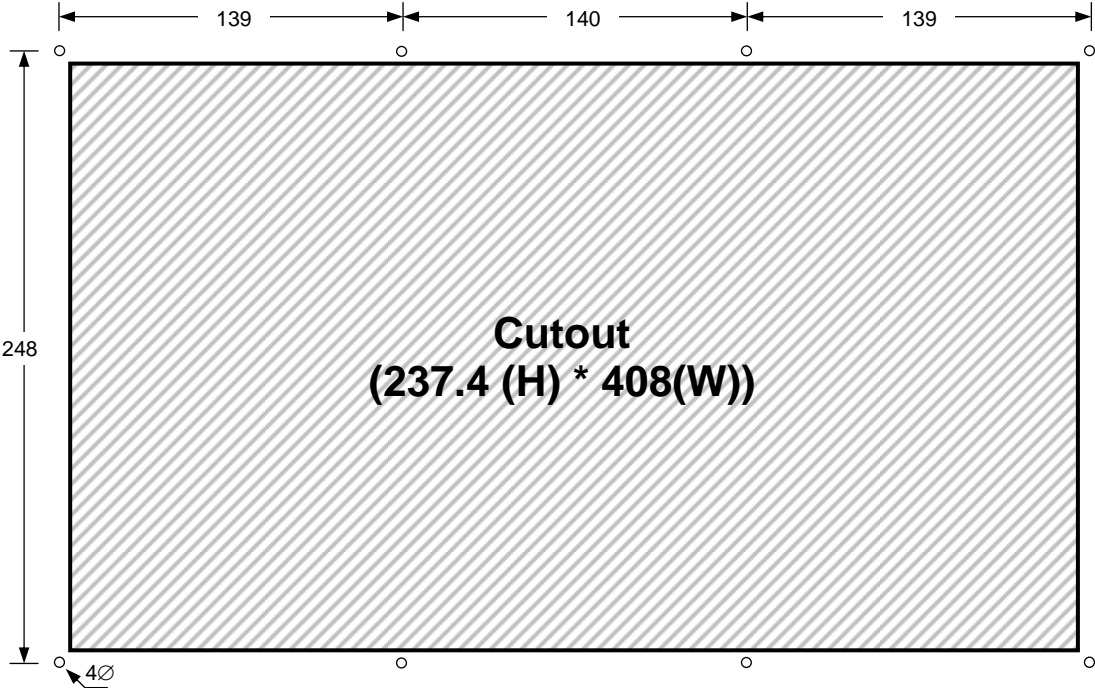


Fig. 5-8 H4CL-T Control Unit Cutout Dimensions

* **H4CL-T Series MDI Panel Cutout Dimensions**

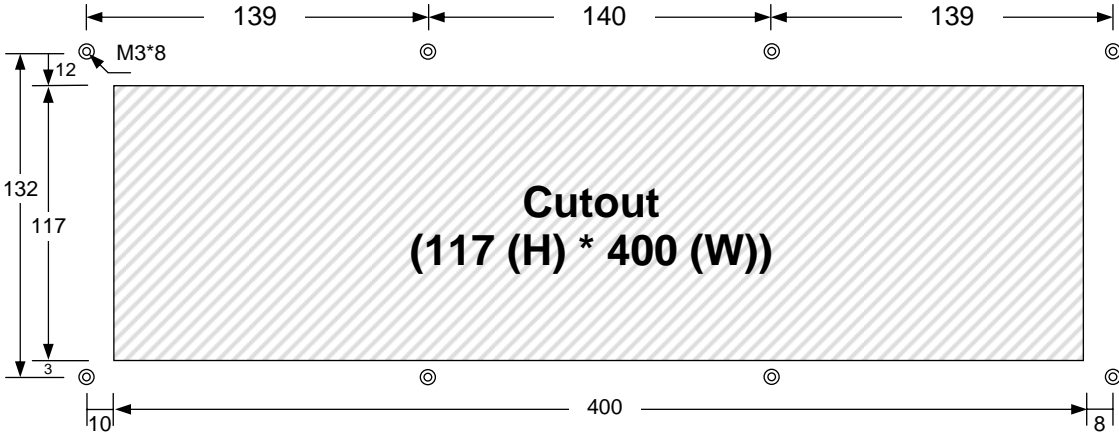


Fig. 5-9 H4CL-T MDI Panel Cutout Dimensions

5.3 Input/Output (I/O) Interface Connections

5.3.1 Input Board / Output Board (Terminal Block Type)

* NPN standard input board – 24 IN.

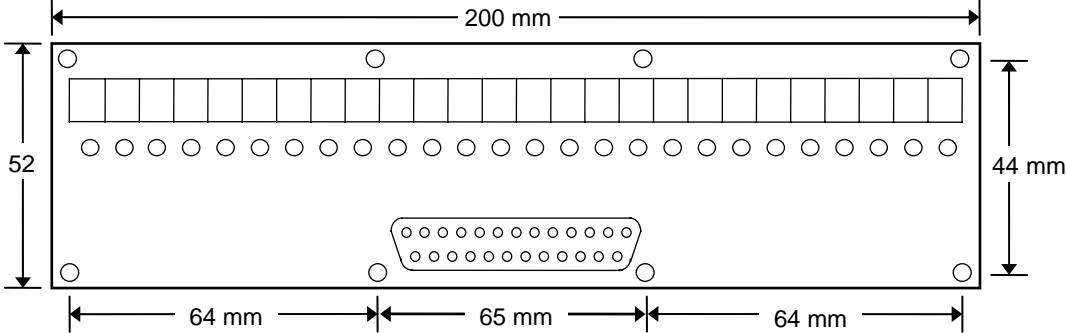


Fig. 5-10 NPN type 1 Input Board

* NPN type standard output board – 16 Outputs.

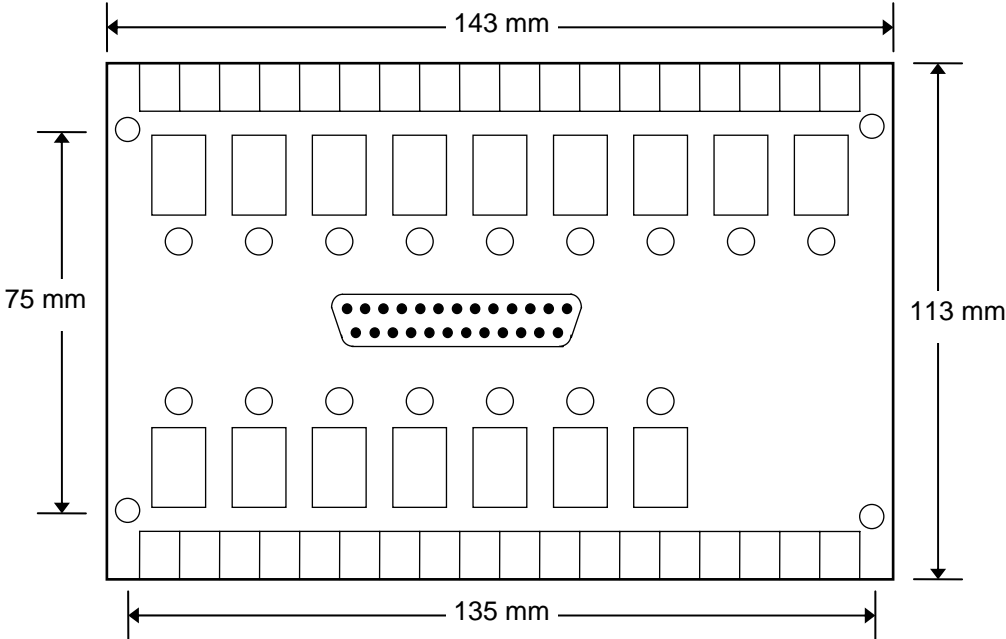
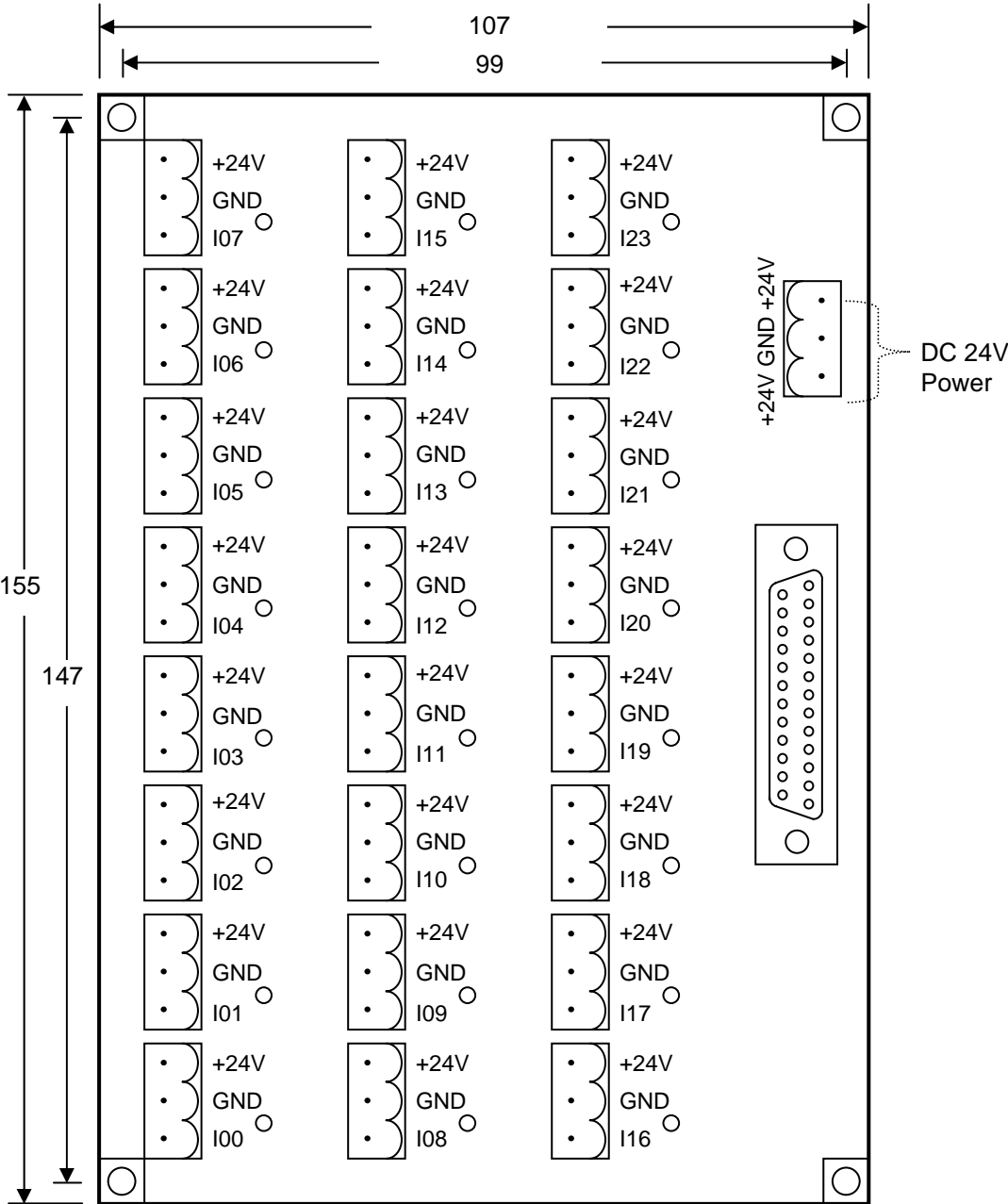


Fig. 5-11 NPN type 1 Output Board

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 illuminates.
- Note: Voltages exist at the +24V terminal of each input when the DC 24V power supply begins operation.

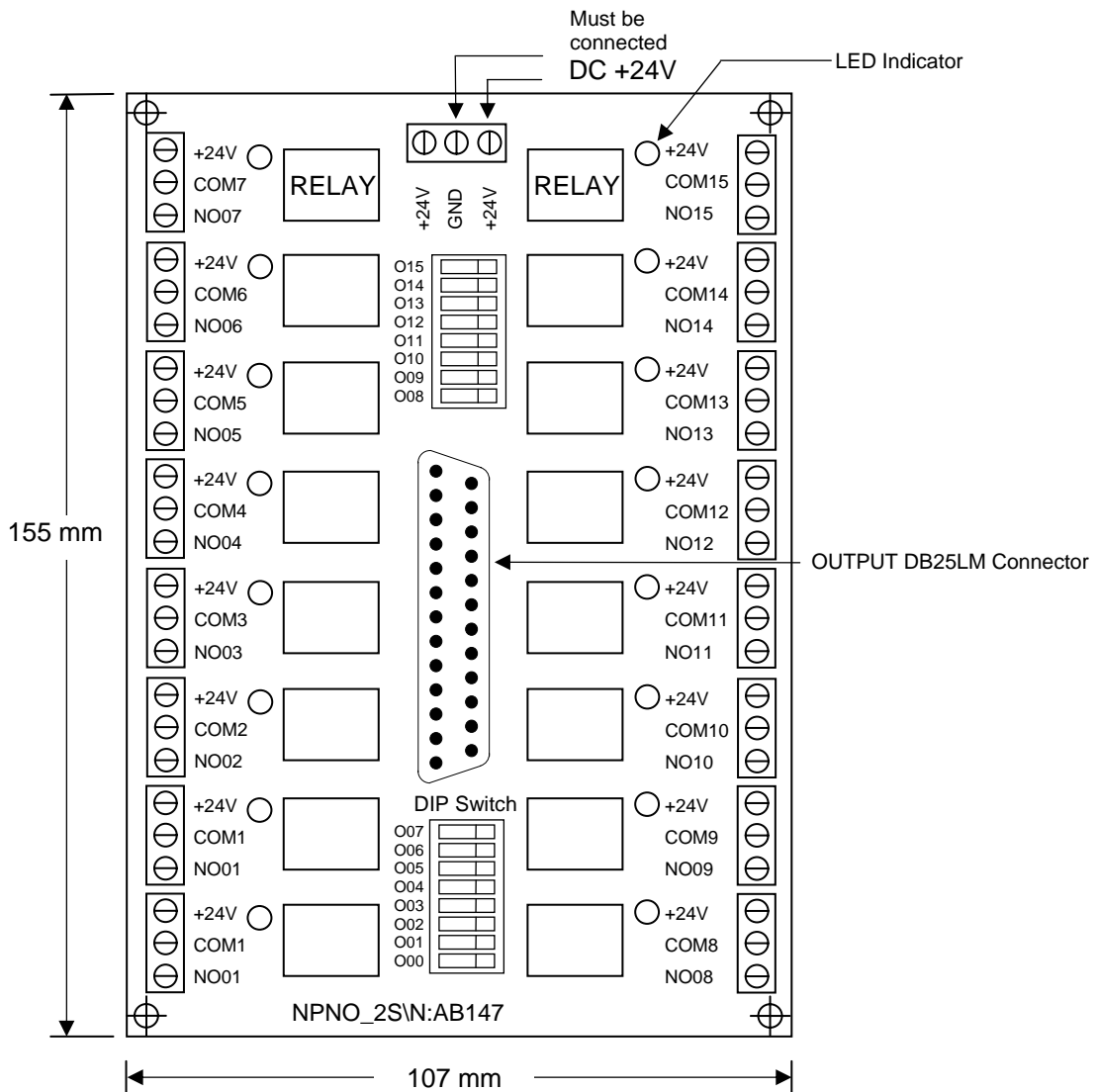


Unit: mm

Fig. 5-12 Input Board

*** Output Board**

- The standard NPN input board provides a 16-input interface.
- During output, the corresponding indicator illuminates.
- If the DIP switch is “OFF”, the COM and NO are the general switch contacts. If the DIP switch is “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 switched 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. 5-13 Output Board

5.3.3 I/O Connector Pin Assignment

The I/O connections and pin assignment are described in this section.

The I/O connector designation and pin assignment are shown in Table 5-1 and Fig. 5-14. Note that there are 24 input points and 16 output points.

Table 5-1 I/O Connector Designations and Pin Assignment

HUST CNC	Connector Designation	I/O Pin Assignment
H4CL-T	Output	O000 ~ O015
	Input	I000 ~ I023

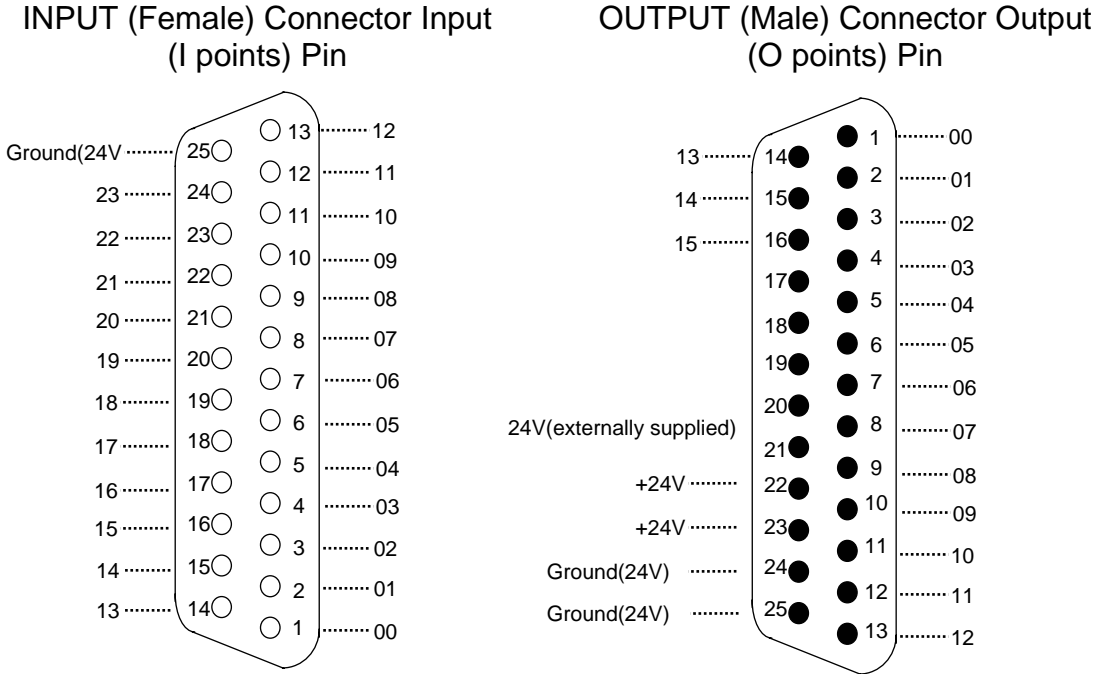


Fig. 5-14 I/O Connector Pin Assignment (NPN-type)

- I/O signals are directly connected to the I/O points when these boards are used.
- I/O signals are connected to the input and output connectors when the input board and output board are not used. (In this case, the transistors in the controller are likely to blow if the line fails or the current existing on the circuit exceeds the rated current.)
- The output points of the H4CL-T Series are transistor circuit of the open collector.

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-T has 24 input points.

*** Input Signal Specification:**

Input voltage: 0 V.
 Input current: 8 mA.

The H4CL-T Series should be fed with the DC 24V power supply from the output board.

The DC 24 power supply will not provided.

*** 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-T) of the numerical control unit when the input board is not used.

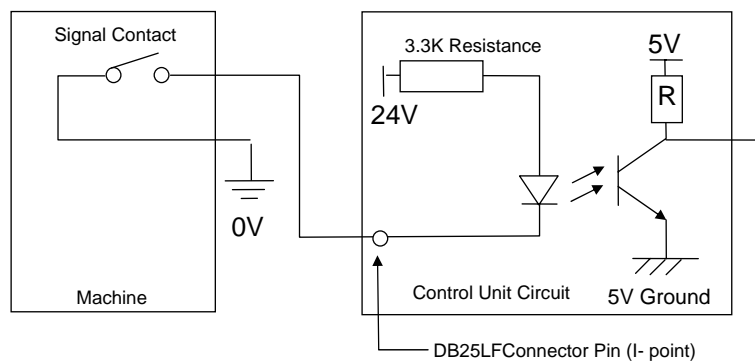


Fig. 5-15 Input Signal Connection (direct input to the control unit)

* **Input Signal Connection Diagram (input to the control unit via the input board)**

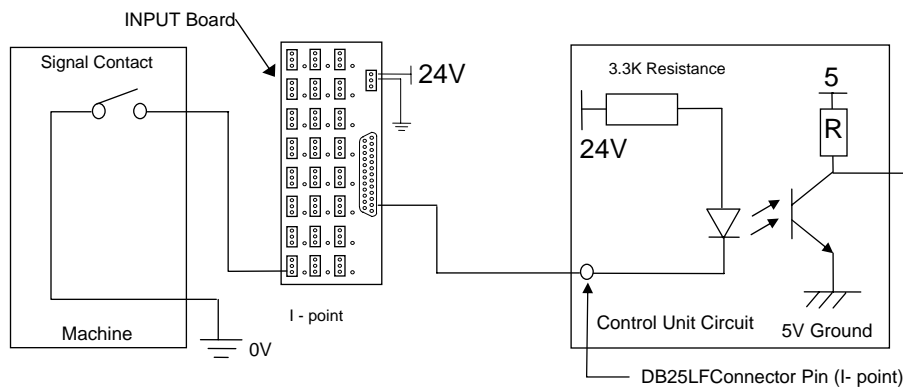


Fig. 5-16 Input Signal Connection (input to the control unit via the input board)

5.3.5 Output Signals

The output signal is transmitted to the external machine from the control unit. The output circuit of the H4CL-T 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 voltage of 250V and 1A current.

(b) Each contact of the H4CL-T transistor circuit supplies up to 24V voltage and 100mA current when the output relay board is not used.

* **Output Signal Connection (direct input to the machine from the control unit)**

The output signal is directly connected to the output points (the output connectors of the H4CL-T) of the numerical control unit when the output relay board is not used.

When the H4CL-T 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 5-17)

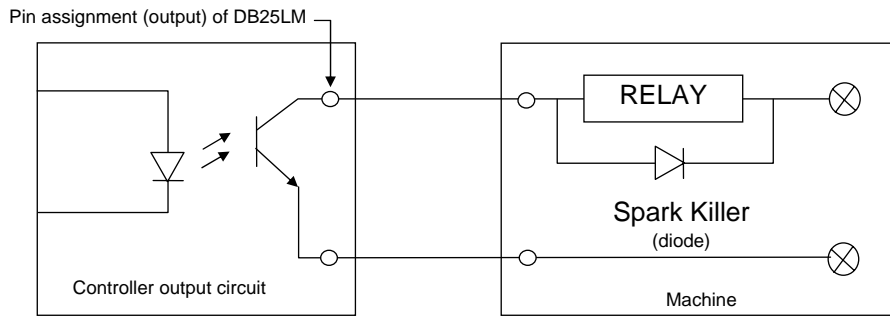


Fig. 5-17 Output Line (without the output relay board)

* **Output Signal Connection (the signal is transmitted to the machine via the output relay board)**

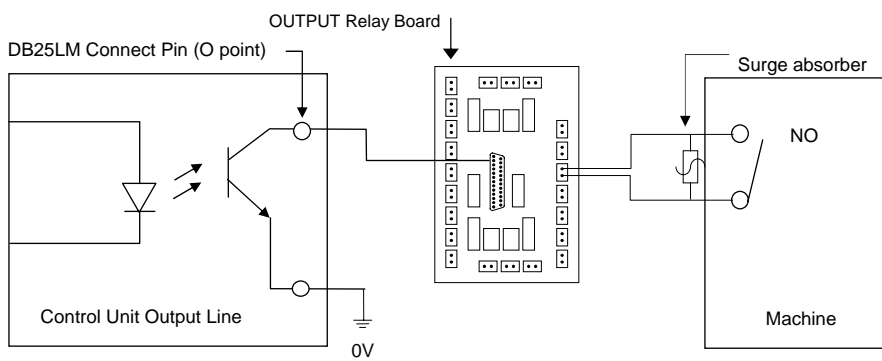


Fig. 5-18 (to the machine via the output relay board)

5.4 Connection Diagram

5.4.1 Connector Type

The connector types on the back of the H4CL-T 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-T connector designation is marked on the back of the control unit case and their corresponding types are listed below:

Connector Name	Connector Designation	Connector Type
Output Interface	Output	DB25LM (Male)
Input Interface	Input	DB25LF (Female)
I/O (expandable)	SIO	DB15LM (Male)
D/A Spindle Analog Output	D / A	DB15LF (Female)
X-axis Servo	X-AXIS	DB9LF (Female)
Z-axis Servo	Z-AXIS	DB9LF (Female)
Spindle	SPINDLE	DB9LF (Female)
MPG	MPG	DB9LM (Male)
RS232C Interface Connector	RS232	DB9LF (Female)

Table 5-2 H4CL-T Connector Designations and Types

5.4.3 H4CL-T Connection (Y-shaped terminal)

* H4CL-T Main Connection Diagram

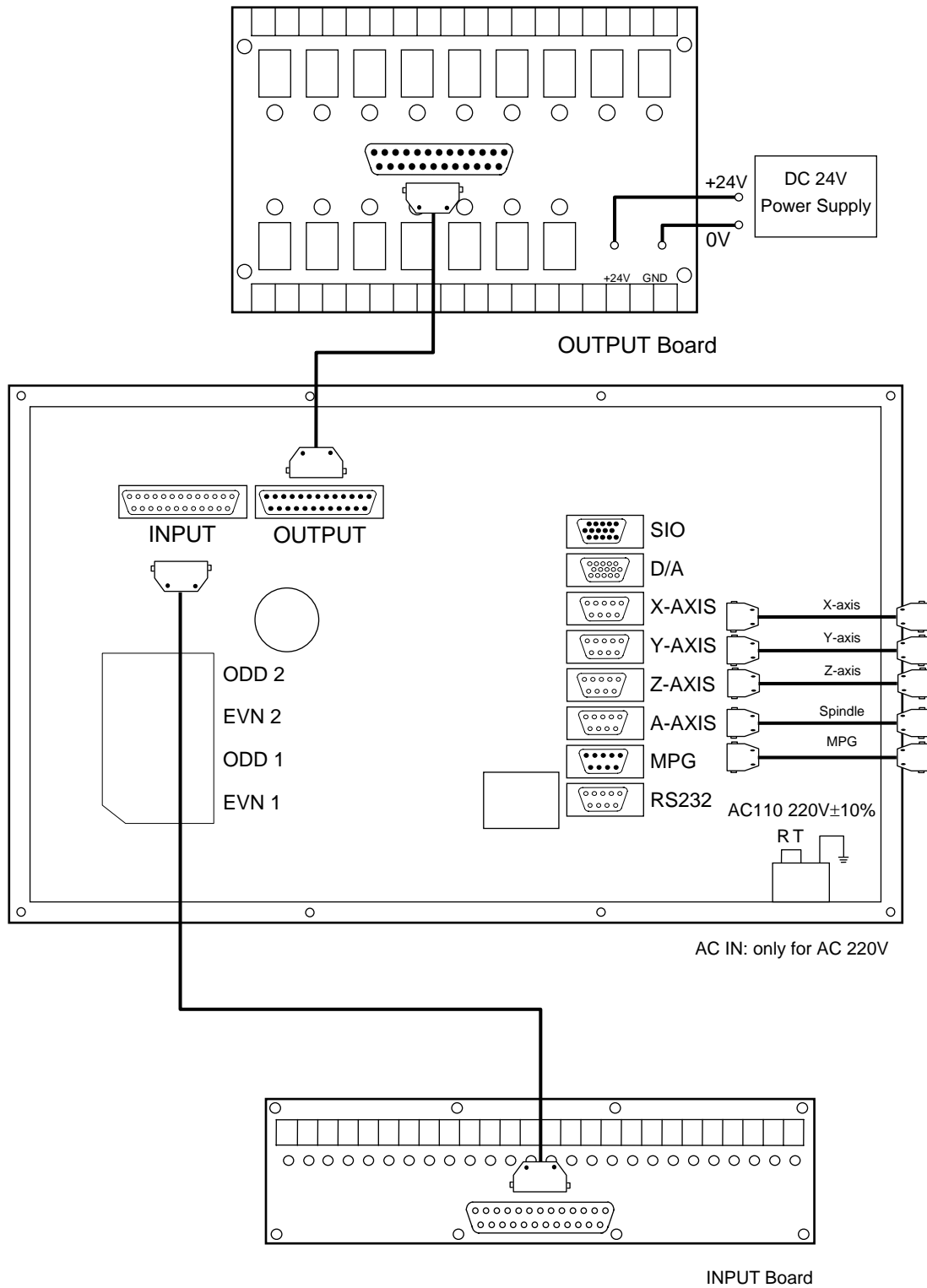


Fig. 5-19

* **Emergency-Stop Line-1**

It is recommended to connect as Fig. 5-20. In doing so, the software and hardware is controlled in series and the user can press the emergency button to turn off the servo even if an abnormality is found in the software.

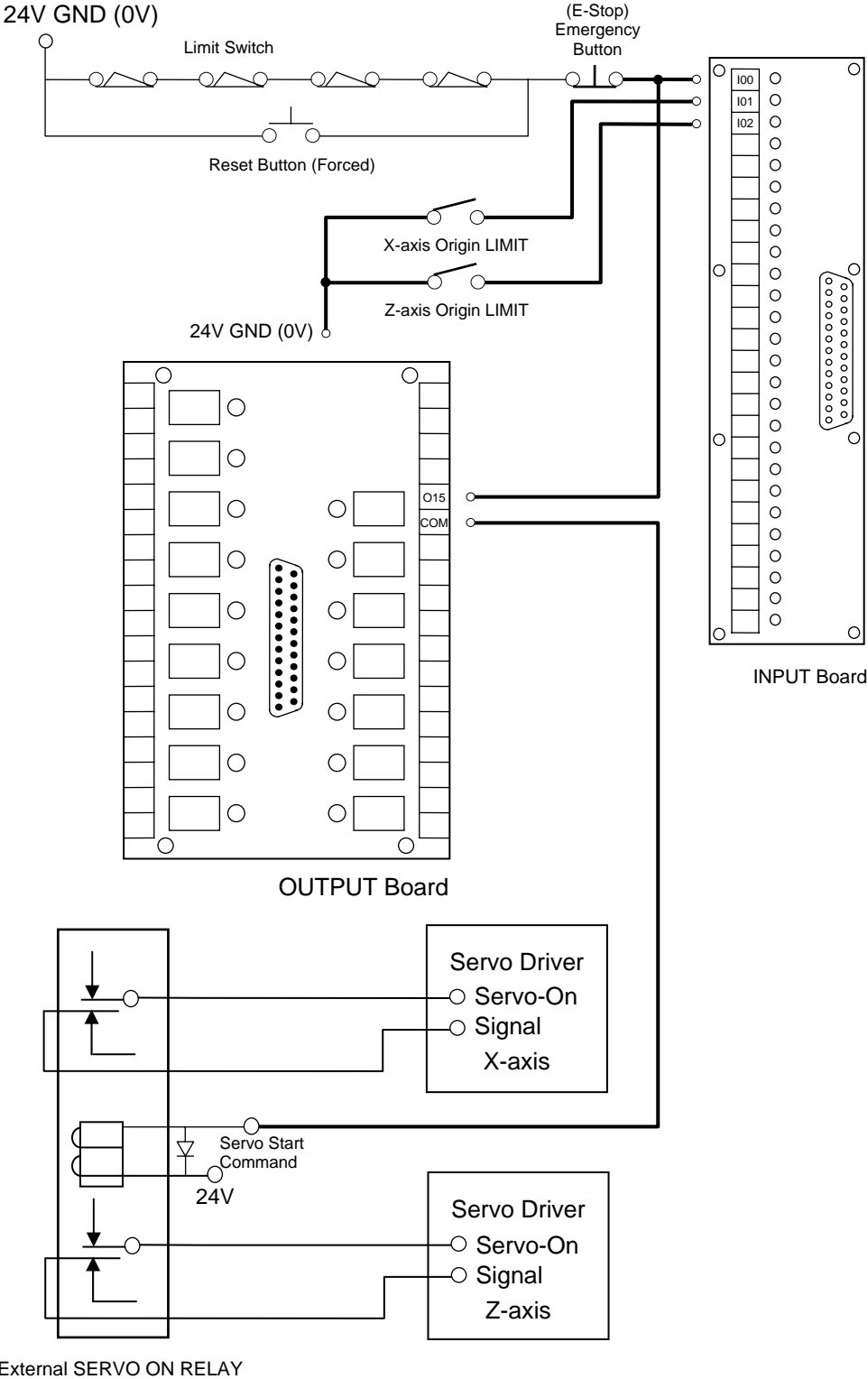


Fig. 5-20

* **Emergency-Stop Line-2**

Fig.5-21 is a simplified connection diagram.

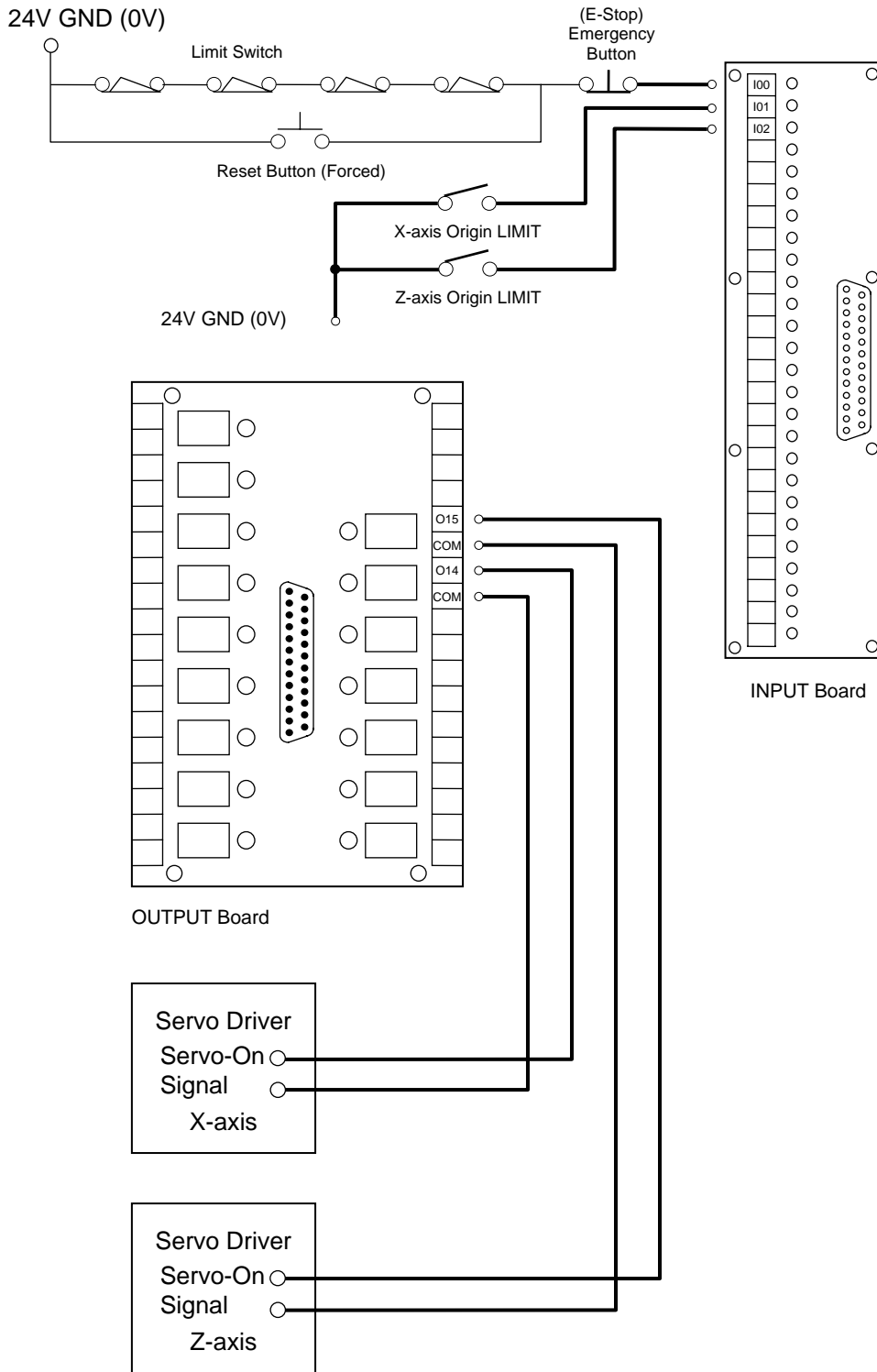


Fig. 5-21

* H4CL-T Spindle Connection

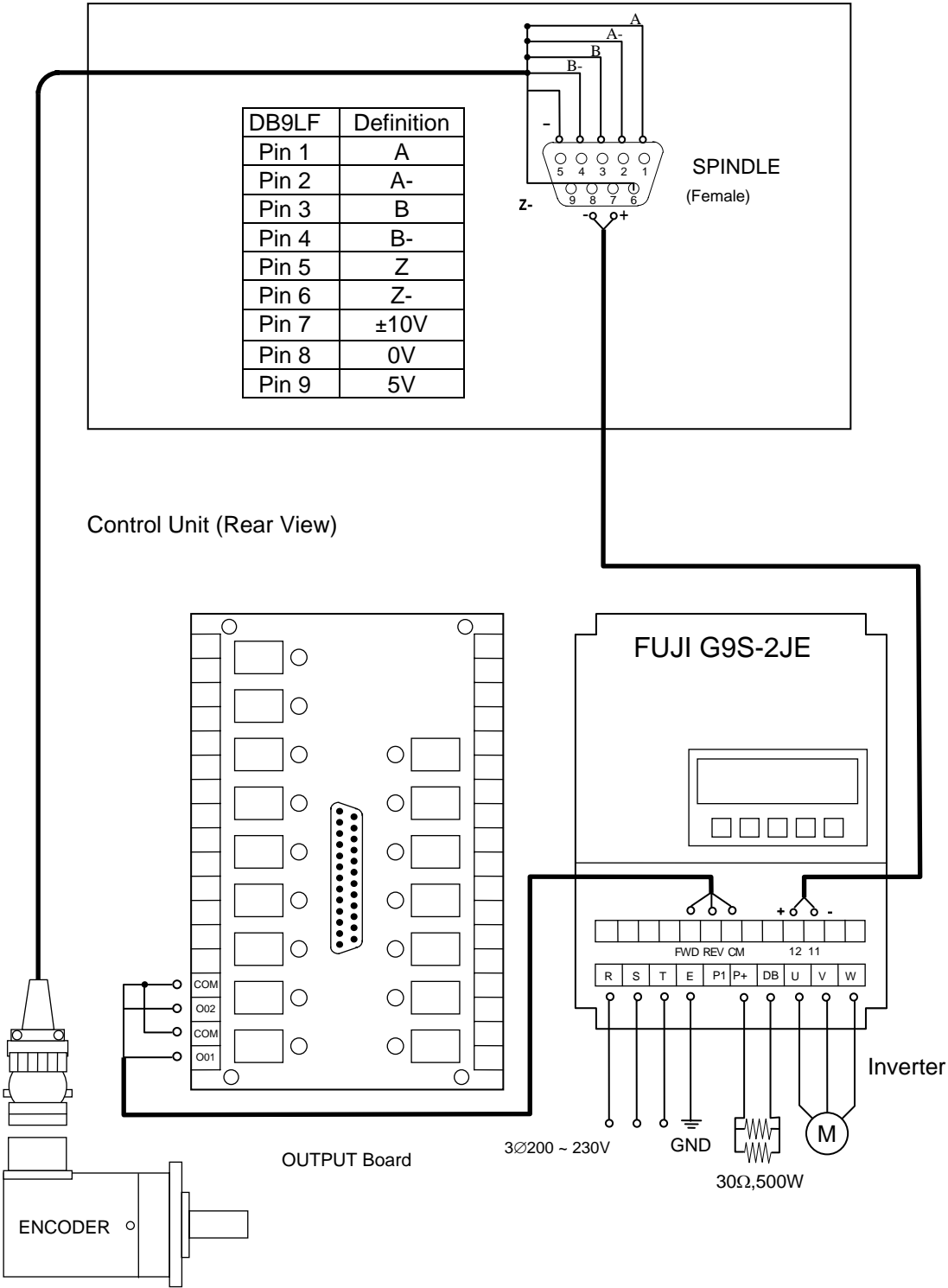


Fig. 5-22

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.

5.4.4 H4CL-T Connection (CE Standard)

* **H4CL-T Main Connection Diagram**

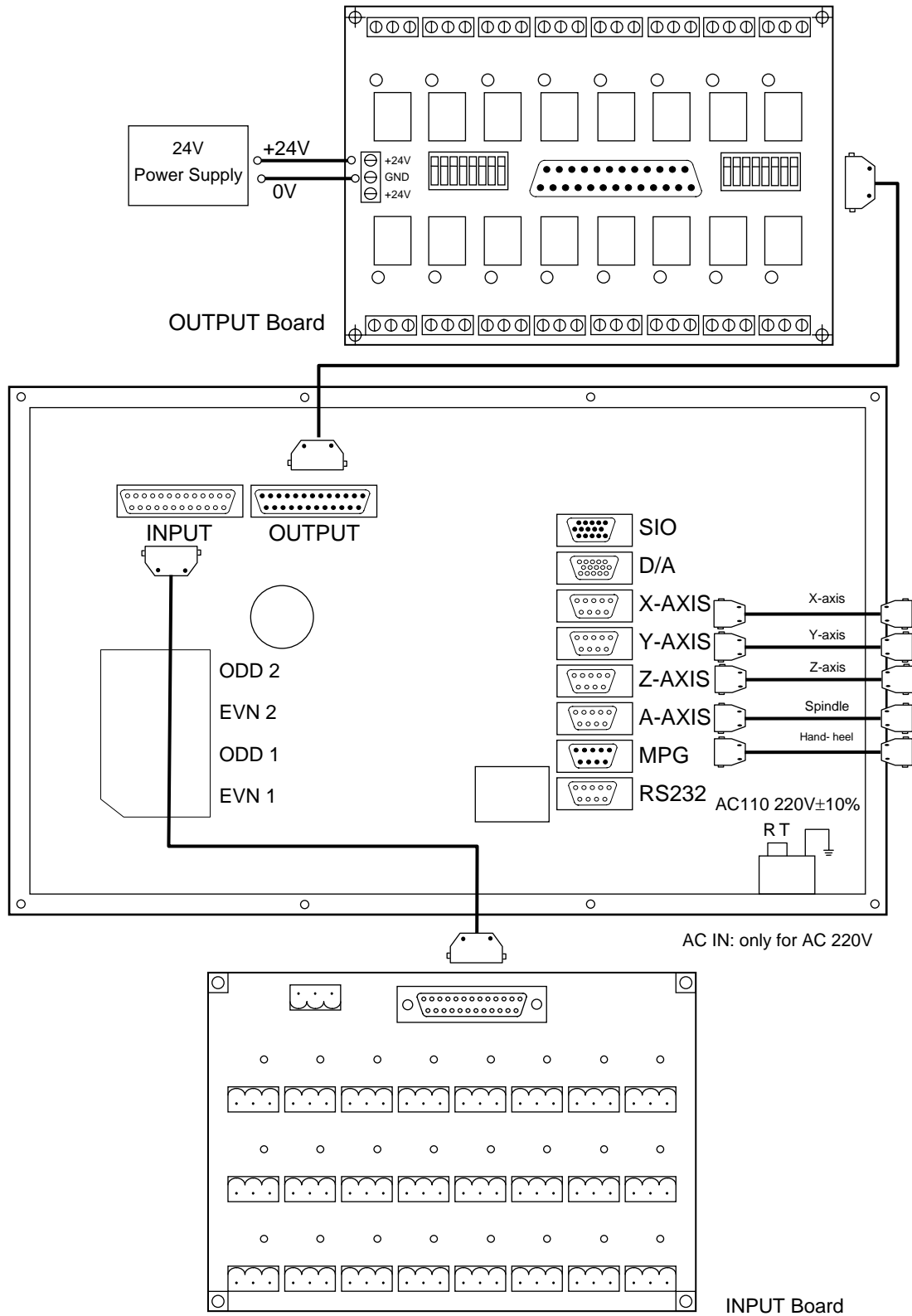


Fig. 5-23

**** Emergency-Stop Line-1**

It is recommended to connect as Fig. 5-24. In doing so, the software and hardware is controlled in series and the user can press the emergency button to turn off the servo even if an abnormality is found in the software.

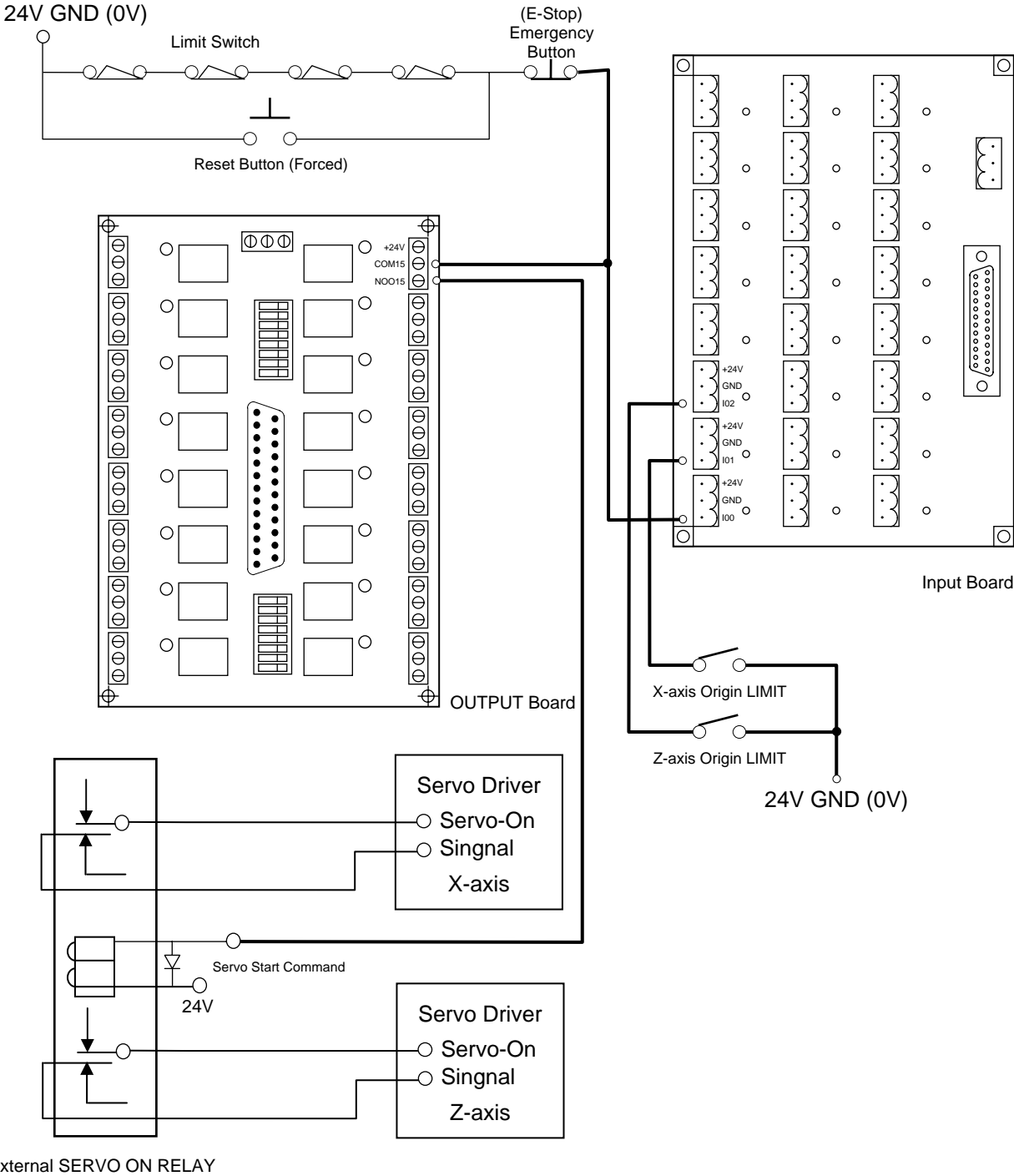


Fig. 5-24

* **Emergency-Stop Line-2**

Fig.5-25 is a simplified connection diagram.

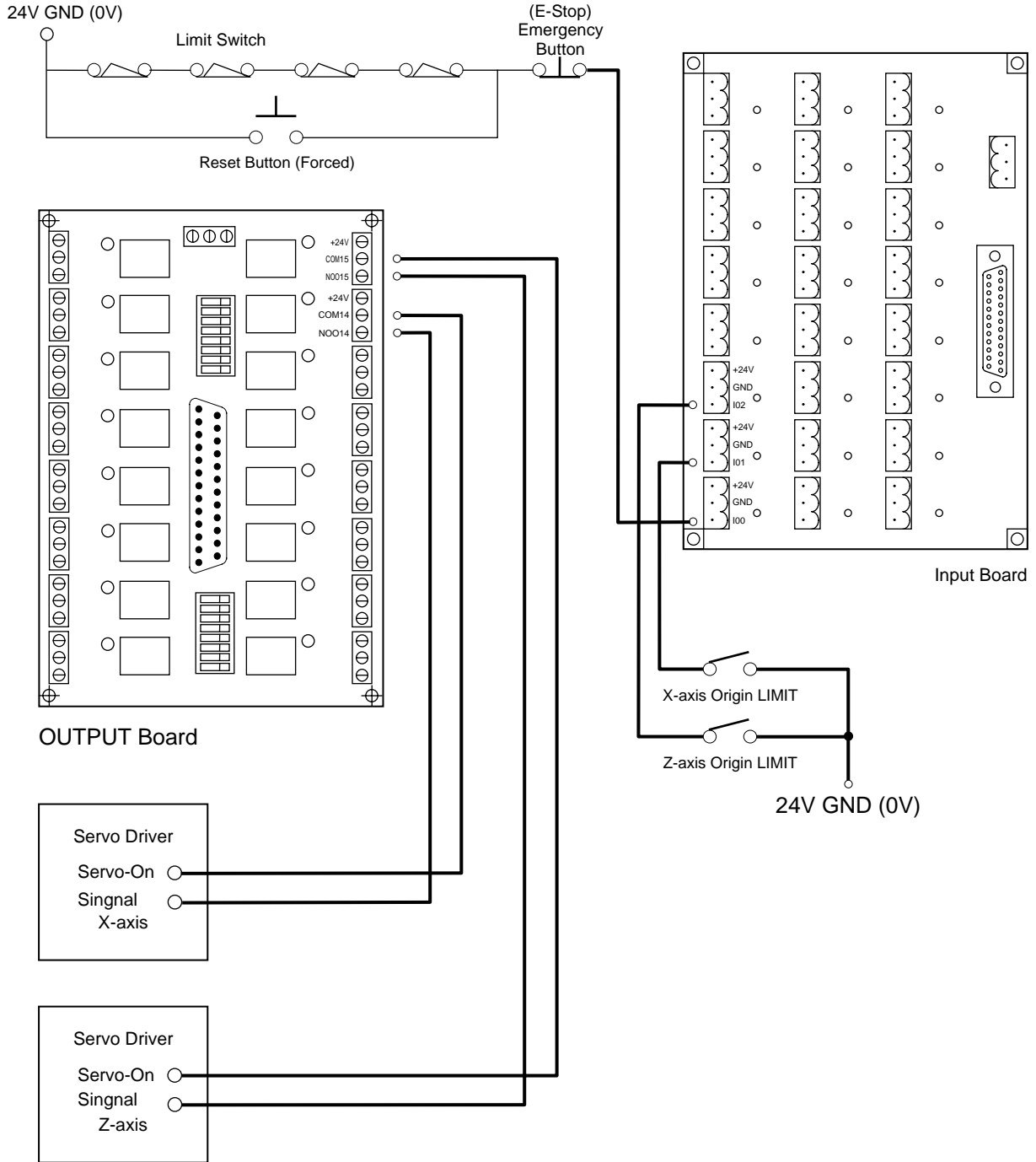


Fig. 5-25

* Servo Motor Connection Diagram (The MITSUBISHI J2S motor is used as an example.)

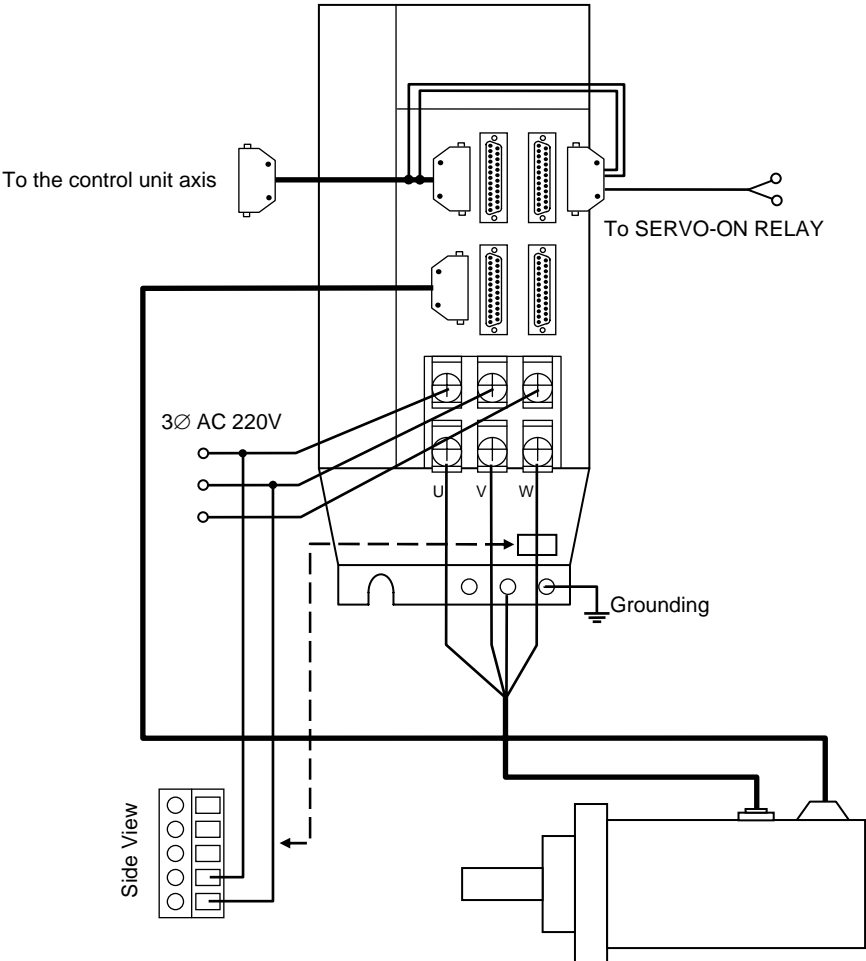


Fig. 5-26

5.4.5 H4CL-L Auxiliary panel + SIO wiring and explanation

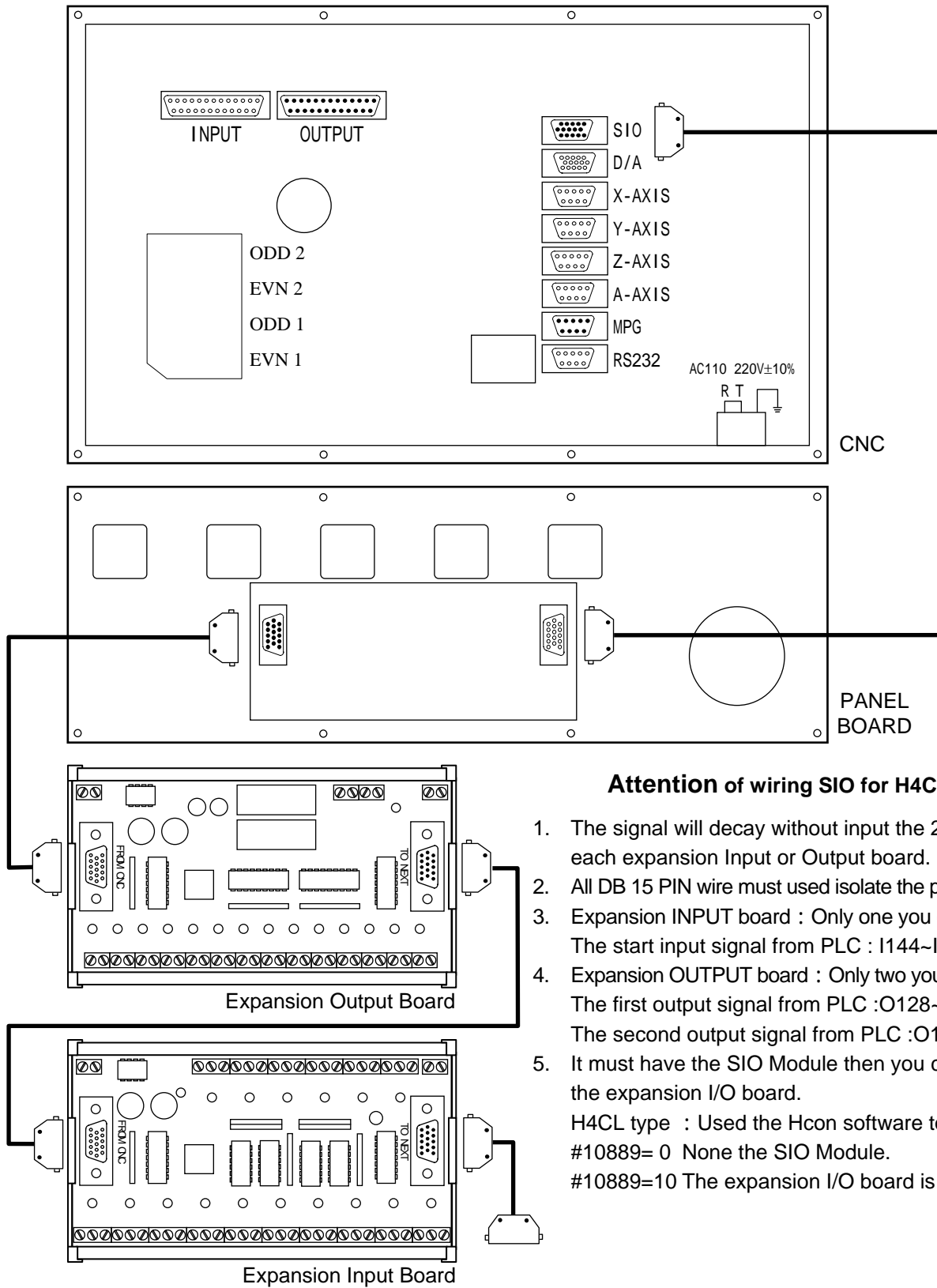


Fig. 5-27

*** Expansion Output Board size**

- 1. There have 16 output for one piece of expansion OUTPUT board.
- 2. O000 O013 is the type of Open-Collector. (24V 100 mA)
- 3. The other two is for Contact type. (5A)

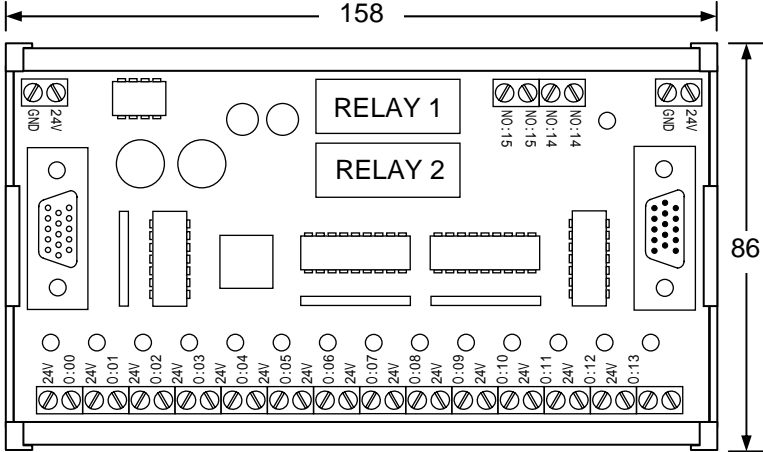


Fig. 5-28 Expansion OUTPUT board

*** Expansion Input Board size**

- 1. There have 16 input for one piece of expansion INPUT board. Input is for NPN type.

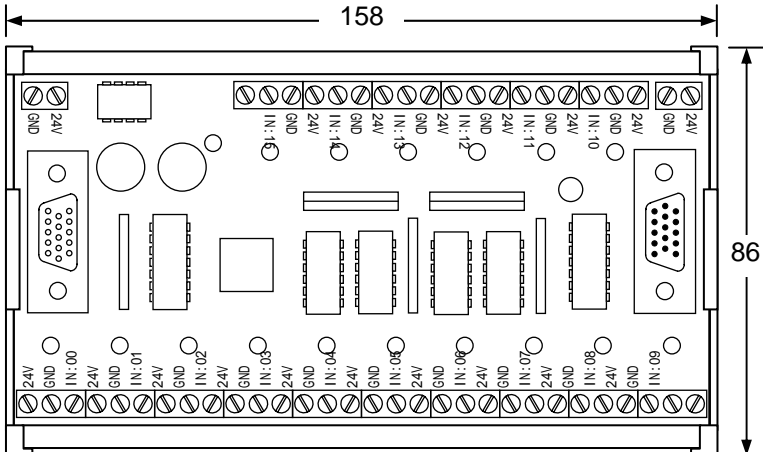


Fig. 5-29 Expansion INPUT board

5.4.6 Connection Method for Servo Drivers & Pulse Generators

The servo driver is connected to the connectors of the X-, and Z-axis, and the spindle encoder and inverter to the spindle. As shown in Fig. 5-30, the pulse generator is connected to the MPG. Servo driver and pulse generator connections vary depending on their brands. Refer to their operating manuals for details.

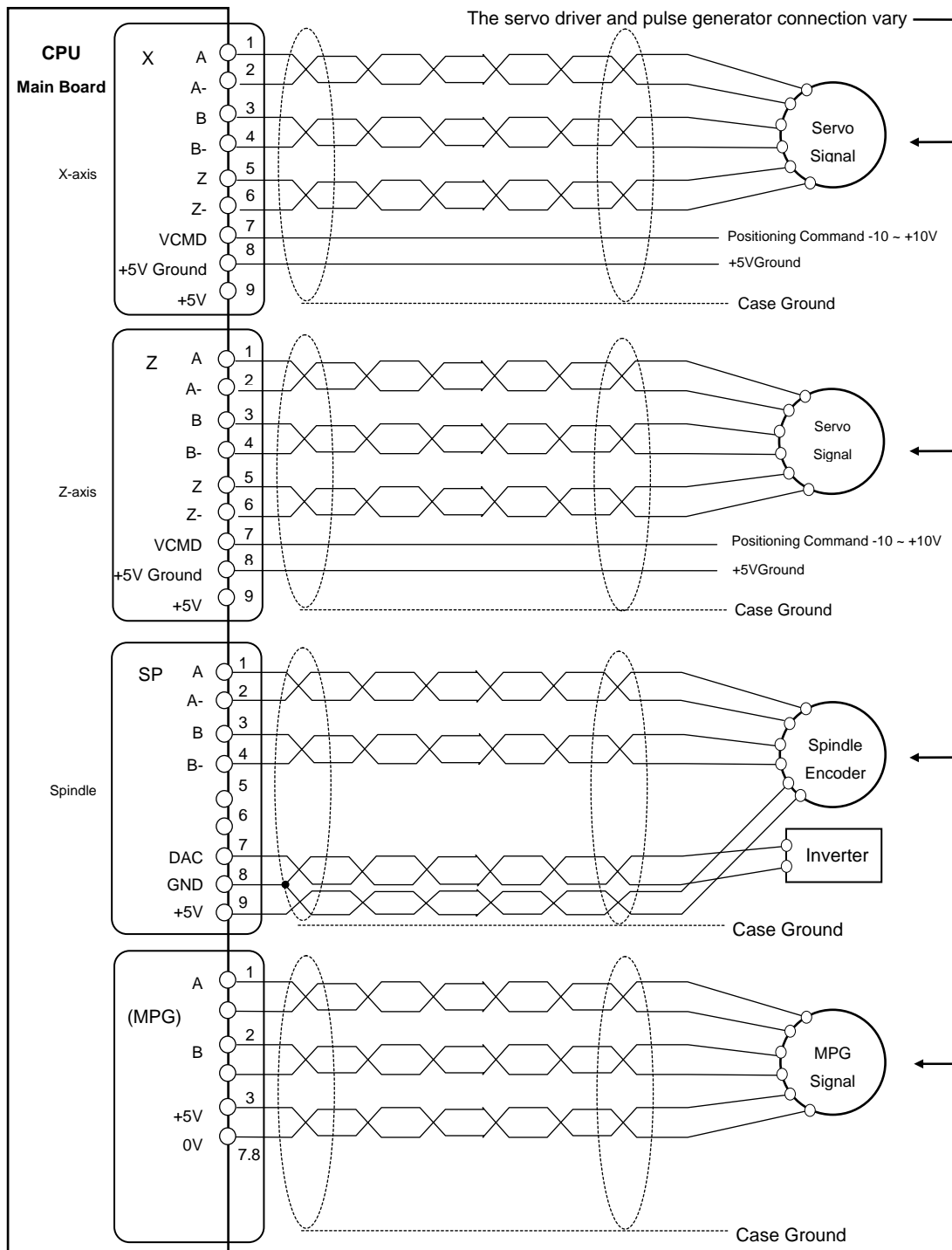


Fig. 5-30 Axes, Spindle, and MPG connection

5.4.7 System AC Power Connection

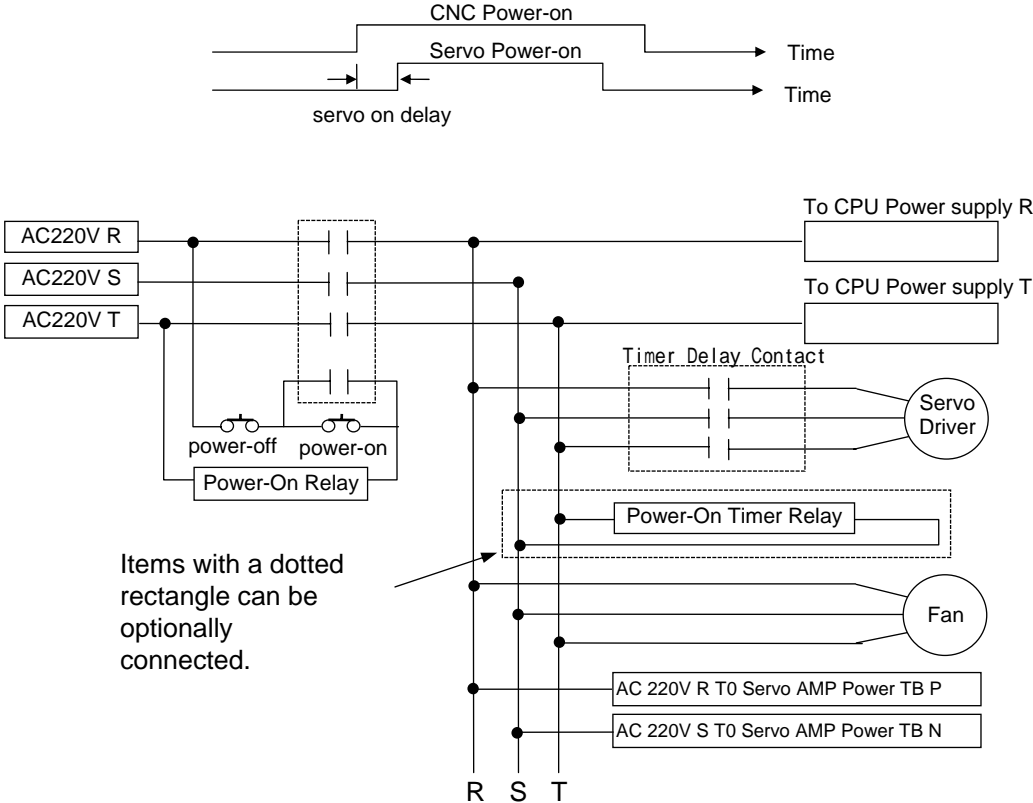


Fig. 5-31 System AC Power Connection

5.4.8 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.

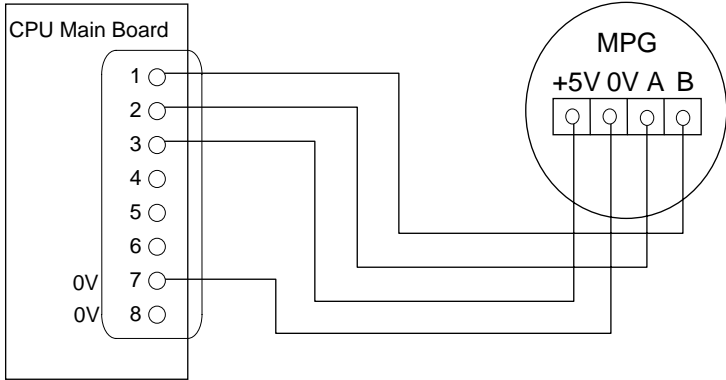
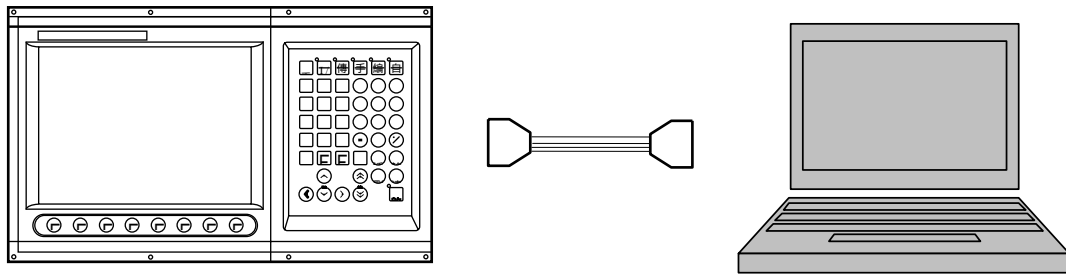


Fig. 5-32 MPG Connection

5.4.9 RS232 Connector Pin Assignment and Connection

Fig. 33 shows the connection method for the H4CL-T control unit and PC. Observe the following notes when connecting:

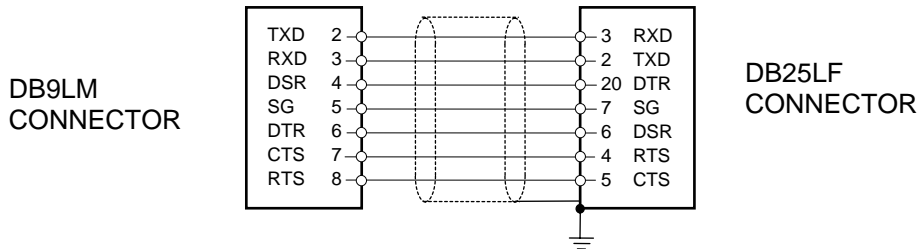
1. The connection between the RS232 C port and PC should not be more than 15 meters.
2. If the connection is established in an environment exposed to electrical noise, such as an electric discharge machine or welding machine, establish the connection with twisted cables or avoid installing in such an environment. The control unit and PC should not share the same power outlet with the electric discharge machine or welding machine.
3. The interface voltage of the PC should range from 10 to 15V.



HUST Control Unit H4CL-T →

RS232

PC COM2



HUST Control Unit H4CL-T →

RS232

PC COM1

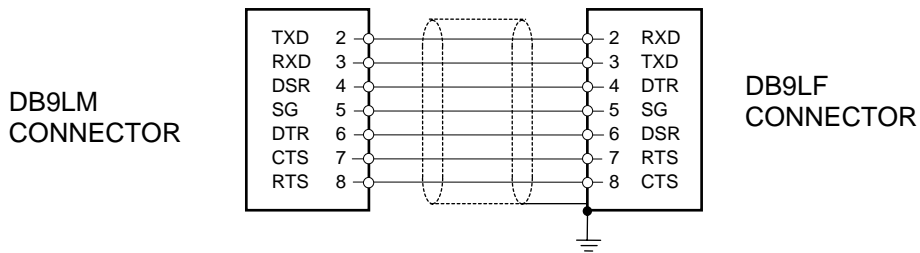


Fig. 5-33 RS232 C Connection

5.4.10 D/A Explanation and Wiring (For Spindle)

D/A for spindle control, the explanation and wiring is below:

* D/A Explanation

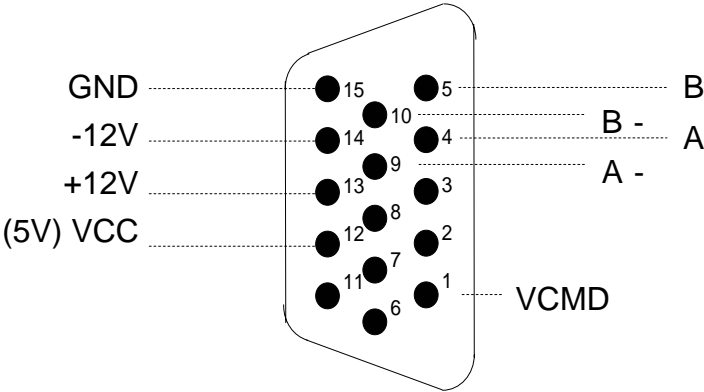


Fig 5-34 D/A Connector

* D/A Wiring

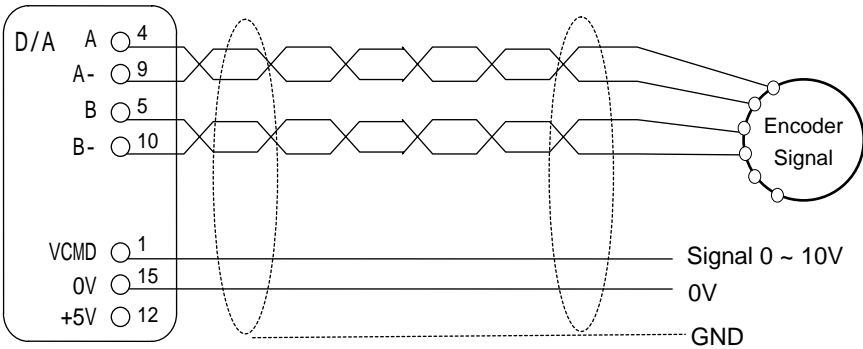
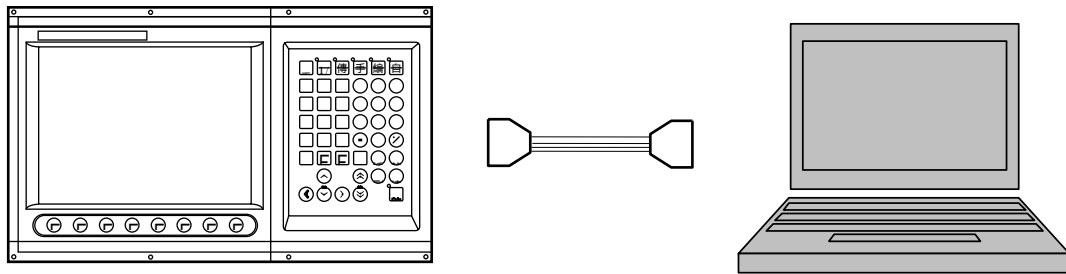


Fig 5-35 D/A Wiring

5.4.9 RS232 Connector Pin Assignment and Connection

Fig. 33 shows the connection method for the H4CL-T control unit and PC. Observe the following notes when connecting:

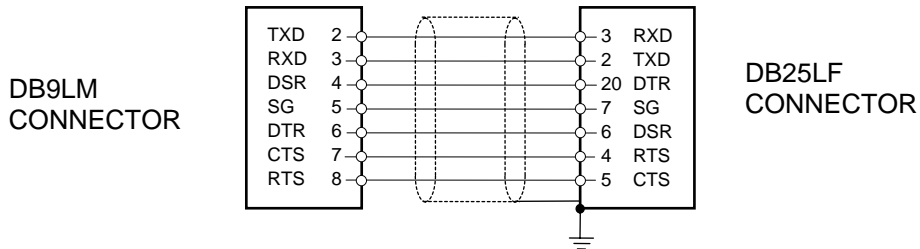
1. The connection between the RS232 C port and PC should not be more than 15 meters.
2. If the connection is established in an environment exposed to electrical noise, such as an electric discharge machine or welding machine, establish the connection with twisted cables or avoid installing in such an environment. The control unit and PC should not share the same power outlet with the electric discharge machine or welding machine.
3. The interface voltage of the PC should range from 10 to 15V.



HUST Control Unit H4CL-T →

RS232

PC COM2



HUST Control Unit H4CL-T →

RS232

PC COM1

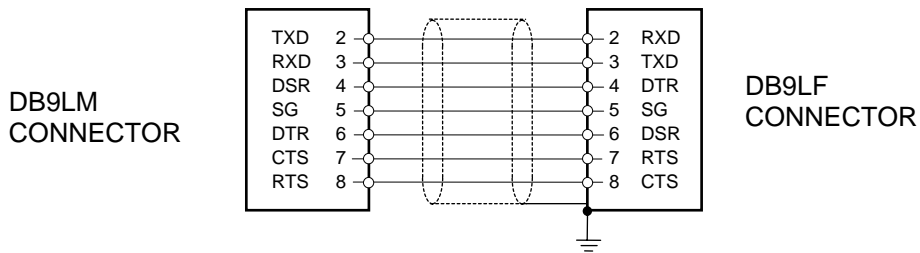


Fig. 5-33 RS232 C Connection

5.4.10 D/A Explanation and Wiring (For Spindle)

D/A for spindle control, the explanation and wiring is below:

* D/A Explanation

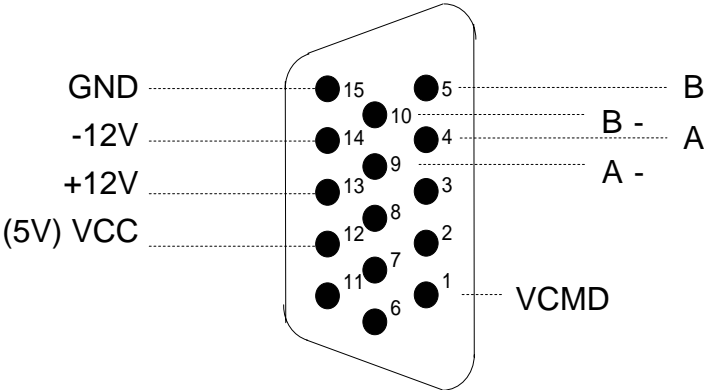


Fig 5-34 D/A Connector

* D/A Wiring

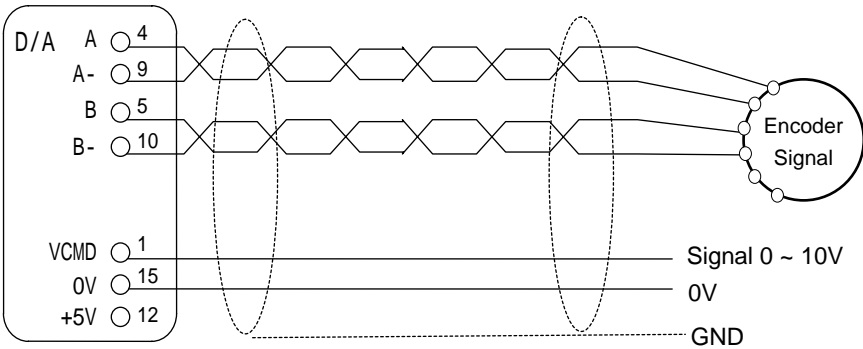


Fig 5-35 D/A Wiring

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-T Series controller. (Fig 6-1). Possible error messages of the H4CL-M Series controller and their solutions are described in the following:

PRNO: 0		CNT: 0	
● X	0.000	MPG x	100
● Z	0.000	G00 MFO:%	100
		G01 MFO:%	100
		SSO MFO:%	100
Spindle:	0	OF-X:	0.000
Feedrate:	1.000	OF-Z:	0.000
		M:	0
		T:	0
		S:	0
			TIME: 0
Error message → Error 02. X AXIS ERR			AUTO STOP
	SINGLE	RESTART	OP-STOP

Fig. 6-1

* **ERROR-01 MCM DATA ERROR OR BATTERY FAIL**

Message:

MCM parameter settings are incorrect or the memory battery has failed.

Recommended Remedy:

1. Check the MCM parameters are correct.
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 controller 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 occurred 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 is correct. Also check the MCM parameter settings.
3. Make sure that the machine and motor functions appropriately and the wiring is correct.
4. Check the connections.

* **ERROR 8 -- Exceed 80 Characters For One Block**

Message:

A block in the program contains more than 80 characters.

Recommended Remedy:

Check the program block and make sure each the block contains less than 80 characters.

* **ERROR 10 – RS232 Error**

Message:

An error occurred in RS232C communication signal of the controller.

Recommended Remedy:

1. Check and make sure that the baud rate of the controller 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 occurred in program memory. The battery may have no power or the data exceeds the memory capacity.

Recommended Remedy:

Enter “MDI” mode and execute the “**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 problem persists after recharging, contact your dealer to replace the battery. (if the controller has not been switched on for more than one year, the data in the memory is lost.)

* **ERROR 13 -- Error G-Code Command**

Message:

An incorrect G-code exists in the program data of the H4CL-T 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:

Manually move the X-axis tool back to the travel limit.

* **ERROR 16 -- Z-axis Over-travel**

Message:

The Z-axis tool moves beyond the pre-set hardware over-travel limit.

Recommended Remedy:

Manually move the Z-axis tool back to the travel limit.

* **ERROR 18 -- 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.

* **ERROR 20 --Software Over-Travel**

Message:

The program runs beyond the software over-travel limit.

Recommended Remedy:

Check the program or reset the MCM “Software Over-Travel Limit”. range

* **ERROR 22 -- 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 do 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 intersecting point are correct.

* **ERROR 30.1 –Battery is Low**

Message:

The controller battery (BT1) has failed.

Recommended Remedy:

Turn on the controller for four hours to recharge the battery. If the problem persists after recharging, contact your dealer to replace the battery.

* **ERROR 31 -- None PLC**

Message:

The PLC program does not exist.

Recommended Remedy:

Check that the PLC exists in the FLASHROM.

* **ERROR 36 –Error in RS232 Program Number**

Message:

The part program number transmitted from RS232C is incorrect.

Recommended Remedy:

Check DNC10 or HCON part program number for correctness.

* **ERROR 37 -- NC Alarm**

Message:

An error occurred in the external control device.

Recommended Remedy:

Check the external control device. Remove the error and RESET the function.

* **ERROR 50~99**

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	NC
I02	Z-axis Home LIMIT	NC
I03	Foot Switch (turned on using the pedal)	Restorative
I04	Auto/Semi-auto	
I05	Selective skip	
I06	Program lock	
I07	Selective dwell	
I08		
I09		
I10	Reset (button)	Reserved for external
I11		
I12	Tool 1 Positioning Signal	
I13	Tool 2 Positioning Signal	
I14	Tool 3 Positioning Signal	
I15	Tool 4 Positioning Signal	
I16	Tool 5 Positioning Signal	
I17	Tool 6 Positioning Signal	
I18	Tool 7 Positioning Signal	
I19	Tool 8 Positioning Signal	
I20	Tool Locking Signal	
I21	Charging Signal	
I22	First spindle origin signal (motor side)	First spindle closed-loop spindle positioning control Applicable only when GR is not 1:1
I23	First spindle origin signal (spindle side)	

I05 Descriptions

“Optional Skip”

When I05 = 1, the “Optional Skip” is selected and the material charging process is interrupted.

I05 = 1 (High) The Optional Skip is selected.

I05 = 0 (Low) The Optional Skip is not selected.

When I05 = 1, the block with “/1” in the program will be skipped. When I05 = 0, the block with “/1” in the program is executed.

Example:

N10 M03 S1000

N20 G01 X20. F1.2

N30 X50. /1 ←

If I04 of block N30 is 1, this block is skipped and block N40 will be executed directly after block N20.

N40 X0.

M05

M02

“Charging”

When I104=0, the “Charging” function is selected.

When I04 = 0, “manual” charging is available in the manual mode using the foot switch (I03).

When I04 = 0, “automatic” charging is available in the auto mode using the M10 (Chuck Open) and M11 (Chuck Close) commands.

* **Output Planning**

Output	Description	Remarks
O00	Spindle Rotation CW	
O01	Spindle Rotation CCW	
O02	Coolant	
O03	Alarm Indicator	
O04	Spindle Chuck	
O05	Lubricant	
O06	Chuck Release Indicator - ON	
O07		
O08	Tool Changing Rotation CW	
O09	Tool Changing Rotation CCW	
O10		
O11	Charging Output	
O12		
O13	Workpiece count reached preset target - ON	
O14	SERVO-ON X	
O15	SERVO-ON Z	

* **OUTPUT PLANNING (Servo spindle mode)**

OUTPUT	Description	Note
O00	s SERVO ON	
O01		
O02	Coolant	
O03	Alarm light	
O04	Spindle chuck	
O05	Lubricant	
O06	Power head 3	M24 ON ; M25 OFF
O07	Spindle dwell	M84 ON ; M85 OFF
O08	Tool changing CW	
O09	Tool changing CCW	
O10	Power head 1	
O11	Feeding output	
O12	Power head 2	
O13	On when Counter reaches the default setting.	
O14	SERVO-ON X	
O15	SERVO-ON Z	

When using multiple spindles, O010 is for the second spindle SERVO ON ;
O012 is for the third spindle 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	Chuck Released	O04=1	
M11	Chuck Closed	O04=0	
M15	Counter +1 #6501+1		
M20	Power head 1 ON	O010=1	Applicable while spindle number =1
M21	Power head 1 OFF	O010=0	Applicable while spindle number =1
M22	Power head 2 ON	O012=1	Applicable while spindle number =1
M23	Power head 2 OFF	O012=0	Applicable while spindle number =1
M24	Power head 3 ON	O006=1	Applicable while the first spindle positioning control is closed-loop
M25	Power head 3 OFF	O006=0	
M84	Spindle dwell ON	O007=1	Applicable while the first spindle positioning control is closed-loop
M85	Spindle dwell OFF	O007=0	
M63	Second spindle rotating CW		O010:enable second spindle junction
M64	Second spindle rotating CCW		
M65	Second spindle stop		
M73	Third spindle rotating CW		O012: enable third spindle junction
M74	Third spindle rotating CCW		
M75	Third spindle stop		

M-code	Description	I/O	Remarks
M50	First spindle switches to standard axial mode		C axis
M51	First spindle switches to spindle mode		
M60	Second spindle switches to standard axial mode		A axis
M61	First spindle switches to spindle mode		C axis

*** G71 ~ G76 Compound Canned Cycle Parameters**

PARAMETERS	X-AXIS	Z-AXIS
G71,G72 Cutting depth	0.000 inch	0.000 inch
G71,G72 Retraction	0.000	
G73 Total cutting depth	0.000 inch	0.000 inch
G73 Cutting cycle	0	
G74,G75 Retraction	0.000	
G76 Finish cutting cycle	0	
Chamfer	0	
Tool-tip angle	0	
Min. cutting depth	0.000	inch
Depth for finish cut	0.000	inch

				⏪ PAGE	
	SYS-MCM	VERSION	ERR-MSG		

Refer to Chapter III “G/M-Code” for more information about G-codes.

Fig. 7-1

*** PLC Parameters**

PARAMETERS	
Graphic Mode , Z-axis Scale	0.000
=0 Radius,=1 Diameter setting	0
Tool Numbers	0
Tool CW Time (10ms)	50
Tool CCW Time (10ms)	50
LUB. Interval Time (10ms)	50
LUB. On Time (10ms)	50
U232 Device ? 0:No 1:Yes	0
Control Mode 0:Panel 1:Key	0
Panel Type 0:SW 1:Key	0
<div style="text-align: right;"> ⬆ PAGE ⬇ </div>	
Change the password	BN-MCM
CLEAR ALL-PGM	LD-MCM
CLEAR OFFSET	

Fig. 7-2

Tool Counts:

Lathe tool changing steps:

1. Tool Changing rotation CW O08=1
2. Turn to the desired tool number (INPUT). Manually change the next tool.
3. O08=0
4. Pause 50x5=250 ms (Timer =79)
5. Tool changing rotation CCW O09=1
6. Wait for the tool locking signal I20=1
7. CCW rotation continues for (Time-4) time (Timer=78)
8. O09=0; Tool changing is completed.

Note: Tool Counts Setting


- 1 or > 8 No changing**
- > 1 and 8 Changing tools**
- 2 ~ 8 tools can be specified**

Example: Tools =5

The cycle "1,2,3,4,5,1,2,3..." is applicable to manual tool changing. If T-code is used for tool changing, T202 indicates that the current tool will be changed with the second tool and the second Tool Offset is selected. T603 indicates that no tool is selected for changing because 6 is greater than 5, but the third Tool Offset is selected.

8. Attachment - B - zDNC Operating Instructions

1. Getting Started

Click  on the desktop to execute zDNC
zDNC

2. Open the Option Setting Screen

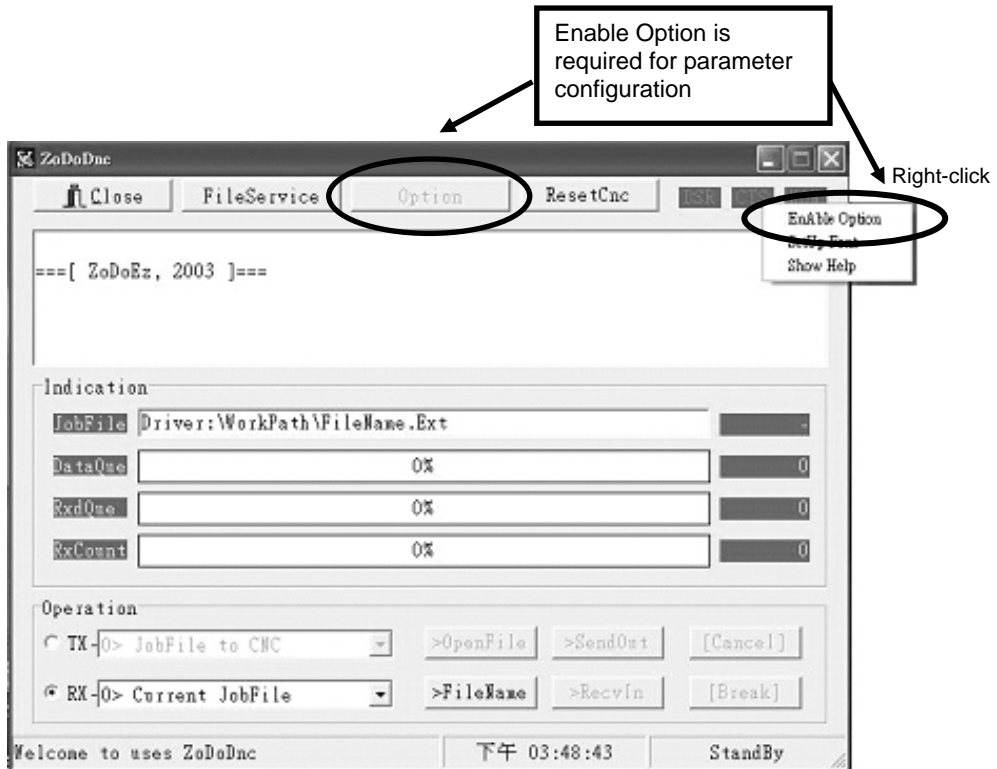


Fig 8-1

3. Display Settings

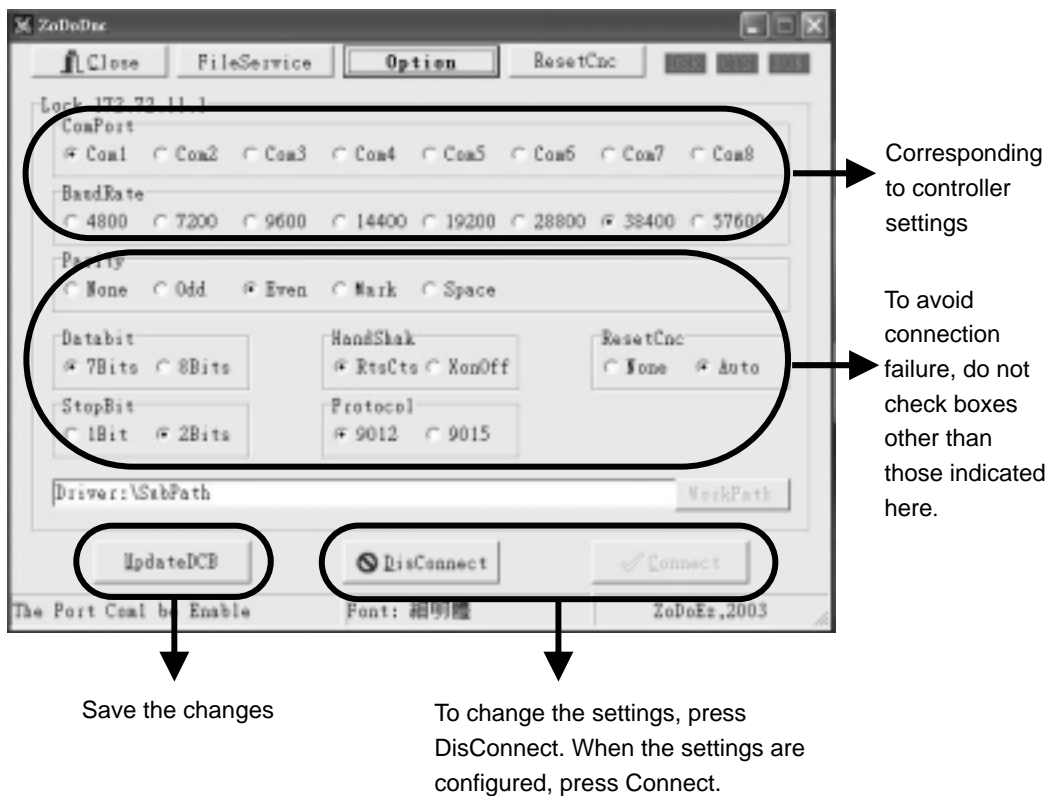


Fig 8-2

4. PC TO CNC

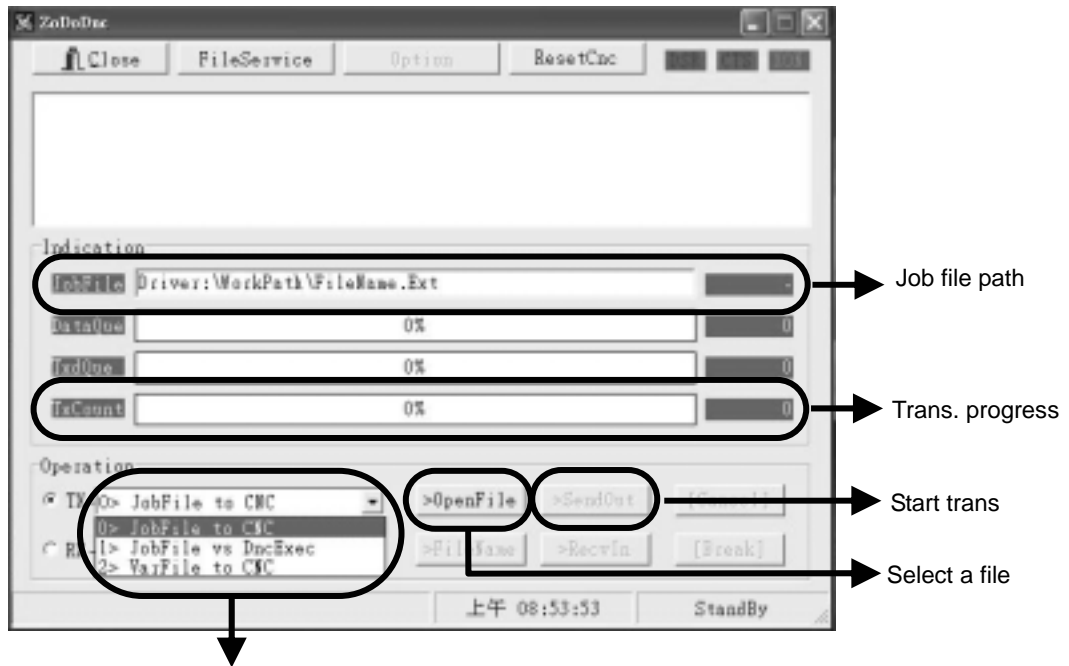


Fig 8-3

5. CNC TO PC

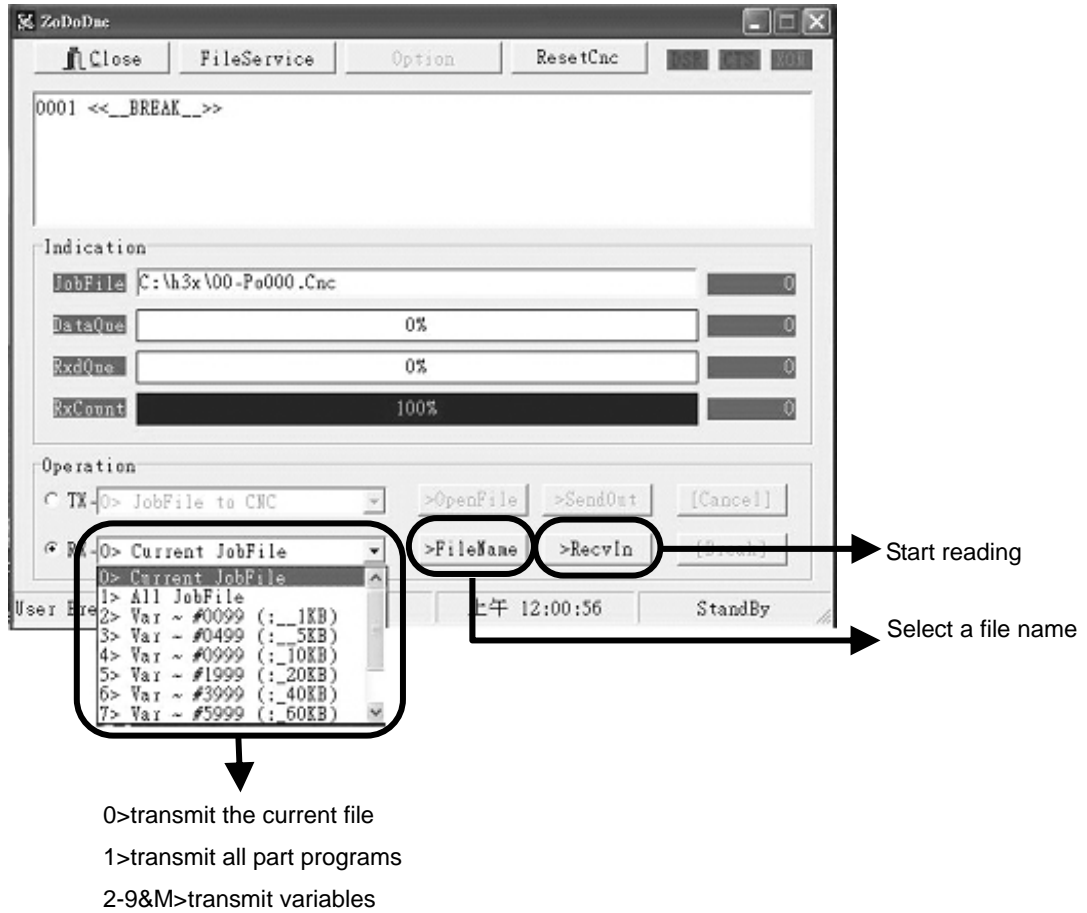


Fig 8-4

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.