

**CONTROL AFZAR TABRIZ**

**HUST M-11/I-11**

**CNC CONTROLLER**

**OPERATION MANUAL**

Version 1.0

February, 1995

No.248 / Felezkaran / resalat ave. / Tabriz / Iran

Tel : +98 411 4462448

Fax : +98 411 4461805



## CONTENTS

<b>1</b>	<b>FEATURES OF HUST M-11/I-11 CNC CONTROLLER</b>	<b>1-1</b>
<b>2</b>	<b>BASIC CONCEPTS OF PROGRAMMING</b>	<b>2-1</b>
2.1	A Part Program	2-1
2.2	Methods of Programming	2-1
2.3	The Composition of A Part Program	2-2
2.4	Coordinate System	2-4
2.4.1	Cartesian Coordinate System	2-4
2.4.2	Coordinate of Tool Position Command	2-5
2.4.3	Work Origin/Work Coordinate	2-7
2.4.4	Machine Origin	2-7
2.5	HUST M-11/I-11 Control Range	2-8
<b>3</b>	<b>PROGRAM DESIGN</b>	<b>3-1</b>
3.1	G-Codes Definition	3-1
3.2	Rapid Positioning Command, G00	3-3
3.3	Linear Cutting Command, G01	3-4
3.4	CNC and Master/Slave Mode (For I-11 Controller Only)	3-5
3.5	Circular Cutting Command, G02, G03	3-9
3.6	Spiral Cutting Command, G17, G18, G19	3-12
3.7	Dwell Command (Temporary Hold), G04	3-14
3.8	Clear Machine Coordinates, G08	3-14
3.9	English/Metric Measuring System Conversion, G20, G21	3-14
3.10	Move to The First Reference Point, G28	3-15
3.11	Return to The Previous Location From Reference Point, G29	3-15
3.12	Move to The Second Reference Point, G30	3-16
3.13	Skip Function, G31	3-16
3.14	Work-piece Dimension Enlarging and Shrinking, G50, G51	3-16
3.15	Work Coordinate System Setting, G92, G54~G59	3-17
3.15.1	Machine Origin (Home)	3-17
3.15.2	Work Coordinate System Setting	3-18
3.16	Mirror-effect Cutting, G68, G69	3-21
3.17	Absolute and Incremental Coordinate Command, G90, G91	3-22
3.18	Canned Cycle Command, G80, G81~G89, G98, G99 (For M-11 Only)	3-23
3.19	Auxiliary Functions, M-Codes and S-codes	3-28
3.20	Sub-program	3-29
<b>4</b>	<b>TOOL COMPENSATION FUNCTIONS</b>	<b>4-1</b>
4.1	Tool-tip Radius and Wear Compensation, G40, G41, G42	4-1
4.1.1	Start of Tool-tip Radius Compensation	4-2
4.1.2	Relationship Between The Radius Compensation and The Tool Path	4-3
4.1.3	Cancel of Tool-tip Radius Compensation	4-4

4.1.4	Notes On Tool Radius Compensation	4-5
4.2	Total Offset Compensation Set and Cancel, G43, G44, G49	4-6
<b>5</b>	<b>CONTROL PANEL KEY AND SCREEN DISPLAY</b>	<b>5-1</b>
5.1	Panel Keys and Function Keys Descriptions	5-2
5.2	Screen Display Modes	5-4
5.2.1	Power-on Display	5-4
5.2.2	Coordinate Display (POSITN)	5-5
5.2.3	Buffer Display	5-6
5.2.4	Program Display	5-7
5.2.5	Program Directory Display	5-7
5.2.6	MCM Parameter Display	5-8
5.2.7	Graphics Display	5-9
5.2.8	I/O/C/S/A-bit Status Display (PLCBIT)	5-10
5.3	Message Descriptions in Reversed Display	5-11
<b>6</b>	<b>PROGRAM EDITING</b>	<b>6-1</b>
6.1	Program Selection	6-1
6.2	Program Editing	6-2
6.3	Revision of An Old Program	6-4
6.4	Editing A Program in Teach Mode	6-7
6.5	Rules For Decimal Input	6-9
6.6	Notes on Program Editing	6-9
<b>7</b>	<b>MCM (MACHINE CONSTANT) PARAMETERS</b>	<b>7-1</b>
7.1	MCM Machine Constant Setting, G10	7-1
7.2	Machine Constant Descriptions	7-3
<b>8</b>	<b>MANUAL OPERATION</b>	<b>8-1</b>
8.1	Manual Operation	8-1
8.1.1	Homing to Machine Origin (HOME)	8-1
8.1.2	Manual JOG Feed Operation	8-2
8.1.3	G01 Manual Feed-rate Override (MFO%)	8-3
8.1.4	G00 Manual Feed-rate Override	8-3
8.1.5	Manual Spindle Speed Override (SSO%) and Rotating Direction	8-3
8.2	Manual Single Block Operation, MDI	8-4
8.2.1	Single Program Block Input and Execution	8-4
8.2.2	G54~G59 Work Coordinate System Setting, G10	8-5
8.2.3	Tool Offset Compensation Setting, G10	8-6
8.3	Auto Execution, AUTO	8-7
8.4	Single Block Execution in AUTO Mode, SINGLE	8-7
8.5	Option Stop (OPST)	8-8

8.6	Skip Function, SKIP	8-9
8.7	Program DRYRUN	8-10
8.8	MPG Hand-wheel Testing	8-10
8.9	Program Re-start, RE-STA	8-11
8.10	Round Corner Non-stop Operation	8-12
8.11	Feed Hold, (F-HOLD)	8-13
8.12	CLR-XR, CLR-YR, CLR-ZR, CLR-BR, Function	8-13
<b>9</b>	<b>PC ON-LINE OPERATION</b>	<b>9-1</b>
9.1	Program Transfer From PC To CNC Controller (READER IN)	9-1
9.2	Program Transfer From CNC Controller To PC (PUNCH OUT)	9-2
9.3	Transfer Part Program From PC to Controller And Execute The Program	9-2
9.4	PLC Ladder Transfer From To CNC and Ladder Test (Ladder IN and Ladder Simulation)	9-3
9.5	Program Format for PC On-line Transfer	9-4
9.6	RS232C Interface -- HUST's DNC.EXE software	9-4
9.7	RS232C Connection	9-6
<b>10</b>	<b>ERROR MESSAGE DESCRIPTIONS</b>	<b>10-1</b>



## 1 FEATURES OF HUST M-11/I-11 CNC CONTROLLER

- Controlled axis: X, Y, Z and B-axis.
- Voltage-driven servo system with max. response speed of 500 KPPS (i.e. 30 meters/min with 1  $\mu$  resolution).
- A CRT screen unit for cutting path display and program editing with display in X-Y-Z, X-Y, Y-Z, or Z-X axis.
- Program design by CAD/CAM on PC. Program input and DNC on-line execution from PC through RS232C interface.
- Memory capacity for CNC main board -- 20 K, with program storage up to 100.
- Battery backup for CNC program storage in case of power-off.
- MCM machine constant parameter setting to suit the specific features of the machine.
- Backlash error compensation for worn lead screw.
- Providing 6 sets of user defined work coordinate to simplify program design.
- Simultaneous use of the absolute and incremental coordinate in the program.
- Data storage for 16 sets of tool radius, offset, and wear compensation.
- Capable of executing program continuously or block by block.
- Function G-code for linear, circular, spiral cutting, mirror-effect cutting and canned cycle drilling. (Canned cycle drilling for M-11)
- Direct use of "R" or "I, J, K" incremental value for radius in circular cutting.
- Functions for "Option Skip", "Option Stop", "Feed Hold" and "Program Restart".
- Capable of setting Master/Slave mode operation. (For I-11)
- MPG hand-wheel test function for cutting product at the speed controller by MPG.
- Equipped with standard programmable logic controller (PLC).
- Equipped with interface connection for manual pulse generator (MPG hand-wheel).
- English/Metric programming.
- Providing DI/DO = 48/24.
- Self-diagnostic and error signaling function.

This operator's manual includes program editing, operation, daily aintenance, with examples and explanations for each command instruction.

If there are any problems in application, please fill out a problem sheet indicating the natures of the problem. Send it by either fax or mail. We will try our best to response to you as soon as possible.

Table 1-1 G-Code Definitions

G-code	Function
* 00	Rapid positioning (fast feed-rate)
* 01 #	Linear cutting (cutting feed-rate)
* 02	Circular (arc) cutting, CW
* 03	Circular (arc) cutting, CCW
04	Dwell (Temporary stop)
08	Clear machine coordinates in all axis
10	Data input
* 17	Spiral cutting, X-Y plane
* 18	Spiral cutting, Z-X plane
* 19	Spiral cutting, Y-Z plane
* 20	System measurement in INCH mode
* 21	System measurement in METRIC mode
28	Moving tool to the 1st reference point
29	Moving tool back to the specified position from the ref. point
30	Moving tool to the 2nd reference point (10 sets)
31 %	Skip function
* 40 #	Tool radius compensation - cancel
* 41	Tool radius compensation - set (comp. to left of path)
* 42	Tool radius compensation - set (comp. to right of path)
* 43	Tool offset compensation (+) direction
* 44	Tool offset compensation (-) direction
* 49	Tool offset compensation - cancel
* 50 %	Cutting size proportional function - cancel
* 51 %	Cutting size proportional function - set
* 54 #	The 1st work coordinate system
* 55	The 2nd work coordinate system
* 56	The 3rd work coordinate system
* 57	The 4th work coordinate system
* 58	The 5th work coordinate system
* 59	The 6th work coordinate system



Table 1-1 G-Code Definitions (Continued)

* 68	Mirror effect cutting across X-axis
* 69	Mirror effect cutting across Y-axis
* 80 \$	Hole drilling canned cycle - cancel
* 81 \$	Hole drilling canned cycle - set
* 82 \$	Hole drilling canned cycle - temporary stop at bottom
* 83 \$	Deep hole drilling canned cycle
* 84 \$	Thread Tapping cycle (Stop at bottom then backout in reverse direction)
* 85 \$	Boring cycle
* 86 \$	Boring cycle (Spindle stop at bottom)
* 89 \$	Boring cycle (Temporary stop at bottom)
* 90	Absolute coordinate
* 91	Incremental coordinate
* 92	Absolute coordinate origin setting
* 98 \$	Return to starting point "S" in fixed canned cycle
* 99 \$	Return to point "R" in fixed canned cycle
<p>* -- G-codes with "*" are modal G-codes.  # -- G-codes with "#" are of power-on default setting.  \$ -- Special function for M-11, milling machine.  % -- Optional function.</p>	



## 2 BASIC CONCEPTS OF PROGRAMMING

### 2.1 A Part Program

Prior to cutting a machine part by using a CNC cutting tool, a computer program, called a part program, must be created to describe the shape of the parts, which is based on some kind of coordinate system. The cutting tool will then follow these coordinates to do exact cutting. To create a part program, a concise machining plan is a necessity, which includes the coordinates for the machine part, coolant, spindle speed, tool type, I/O-bit, etc.. When designing a machining plan, the following factors must be considered:

- Determine the machining range requirement and select the suitable CNC machine tool.
- Determine the work-piece loading method and select the appropriate cutting tool and the tool holder.
- Determine the machining sequence and the tool path.
- Determine the cutting conditions such as spindle speed (S), feedrate (F), coolant, etc.

A part program is a group of sequential instructions formulated according to the machining plan. It can be edited either on a personal computer (PC), then transmitted to the CNC controller through RS232C interface or directly on the CNC controller using the editing keys. HUST 11-series can do both. They will be discussed later.

### 2.2 Methods of Programming

A CNC controller will execute the commands exactly in accordance with the instructions of the part program. So, the program design is the most important task in the whole CNC machining process. There are two ways to design a CNC part program and are to be briefly described as below:

#### 1. Manual Programming

Manual programming is a process that the whole process is manually done by hand including the coordinate calculations. It follows this sequence.

- Machine part drawing.
- Part shape description including coordinate calculations.
- Computer program design including spindle speed, feed rate, M-code, etc..
- Keying in the program instructions into the CNC controller or transmitted from PC.
- Testing the program.

The coordinate calculation is a simple process if the part shape is composed of lines or 90 degree angles. For curve cutting, however, the calculation will be more complicate and trigonometry will be required for correct answers. Once all calculations have been completed, the CNC part program is written in the formats to be discussed later.

The main disadvantage of manual programming, particularly when designing for a very complicate part, is time consuming and prone to making errors. In this case, automatic programming becomes more advantageous than the manual method.

## 2. Automatic Programming

Automatic programming is a process in which the design work including coordinate calculation is done by computer. It follows this sequence.

- Computer added design for part drawing (CAD).
- Computer added manufacturing for CNC part program (CAM).
- Transferring program to CNC controller.
- Testing the program.

By making use of computer's high speed calculating capability, program designer can communicate with the computer in simple language, to describe the shape, size and cutting sequence of the part. The computer will transfer the motions of the machine tool into a part program, which is then transferred into CNC controller through RS232C interface. This process is called CAD/CAM. It is a necessary tool when designing a part program for a 3-D work-piece.

### 2.3 The Composition of A Part Program

A complete part program is composed of program BLOCKS, starting with a program number Oxxx, ended with M2, M30, or M99, and in between with a series of CNC instructions. A CNC instruction is a command to order the cutting tool to move from one location to another with the specified speed, or the peripheral equipment to do some mechanical work. The cutting is done when the cutting tool moves.

An example of a complete part program containing nine (9) blocks is as follows:

```
N20 G00 X0.000 Y0.000 Z0.000
N30 M3 S1000
N40 G1 X10.000 Y10.000 Z10.000 F200
N50 X20.000
N60 Y10.000
N70 Z10.000
N80 G0 X0.000 Y0.000 Z0.000
N90 M5
N100 M2
```

A block of program can have one to several instructions and it has a general form as follow. The block sequence number "Nxx" can be omitted. If you do not key in the block number, HUST 11-series has a special function "Auto-N" to automatically generate the number for you during or after program editing (see Chap 6). The program execution starts from top to bottom block and has nothing to do with the order of block sequence number. Each instruction starts with an English letter (A~Z), followed by a integer or floating number, depending on what type of instruction the number is associated with. If the number represents a coordinate, it can be positive (+) or negative (-).

N \_\_\_ G \_\_\_ X(U) \_\_\_ Y(V) \_\_\_ Z(W) \_\_\_ B \_\_\_ F \_\_\_ S \_\_\_ D(H) \_\_\_ M \_\_\_

In general, the program instructions can be divided into four categories.

1. Function command : G-code. A CNC command to instruct the cutting tool to do an action, such as straight, circular or thread cut, compound cut, etc.
2. Position command : X, Y, Z, U, V, W. A coordinate command to instruct the cutting tool to stop the cutting action at the location specified -- an end point. The end point of the current block is the starting point of the next block.  
(Motion command)
3. Feed-rate command : F-code. A command to instruct the cutting tool how fast to do the cutting.
4. Auxiliary command : M, S, D, H, etc. A command to instruct the peripheral equipment to do an action, such as spindle speed, coolant on/off, program stop, etc.

Each command code has a fixed format and a special meaning to the CNC controller and it must be strictly followed when designing a program. The system will not accept the command if the format is in error. Otherwise, a machine error will result. Followings are the command codes that are used in HUST 11-series.

- D : Tool radius compensation number.
- F : Feed-rate in mm/min or mm/revolution, a decimal.
- G : Function G-code, an integer.
- H : Tool offset compensation number.
- I : The X-axis component of the arc radius @ the start point, a decimal.
- J : The Y-axis component of the arc radius @ the start point, a decimal.
- K : The Z-axis component of the arc radius @ the start point, a decimal.
- K, L : Repetition counter, integer.
- M : Control code for peripheral machine tool, integer.
- N : Program block (sequence) number, integer.
- O : Program number, integer.
- P : Dwell time; subprogram code; or parameter in canned cycles, integer.
- Q : Parameter in canned cycles, integer.
- R : Arc radius or "R" point in canned cycles, decimal.
- S : Spindle speed, integer.
- U : Incremental coordinate in X-axis, decimal.
- V : Incremental coordinate in Y-axis, decimal.
- W : Incremental coordinate in Z-axis, decimal.
- X : Absolute coordinate in X-axis, decimal.
- Y : Absolute coordinate in Y-axis, decimal.
- Z : Absolute coordinate in Z-axis, decimal.

## 2.4 Coordinate System

The machining action of a cutting tool is accomplished when the tool is moving along a specific path from point A to point B, which represent the shape or the contour of a machine part. In order for the tool to follow the specific path, a computer program describing the shape of the machine part must be created and the shape or the contour is described by the Cartesian coordinate system.

### 2.4.1 Cartesian Coordinate System

HUST 11-series uses the customarily 3-D/2-D Cartesian coordinate system as shown in Fig 2-1 and Fig 2-2. The 2-D system can be represented by X-Y, Y-Z, or Z-X-axis. The intersecting point of the two axes is the origin,  $X=Y=0$ ,  $Y=Z=0$ , or  $Z=X=0$ .

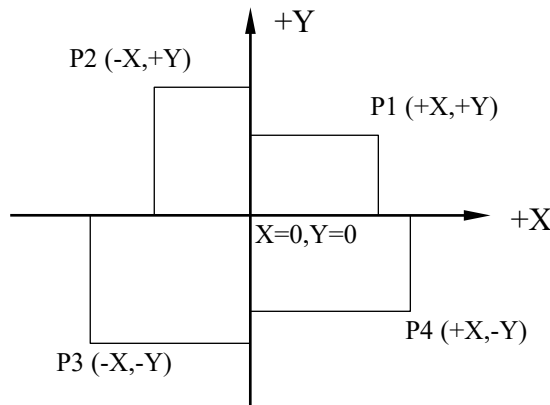


Fig 2-1 2-D Coordinate System

Fig 2-2 is 3-D system (right-hand rule) with the intersecting point designated as origin  $X=Y=Z=0$ . The direction of normal rotation for each axis is indicated by the direction of the four fingers when you grab the axis by the right hand with your thumb pointing to the (+) direction of that axis. The fourth axis can be treated as normal linear axis or the spindle axis by the setting of MCM Parameter #37.

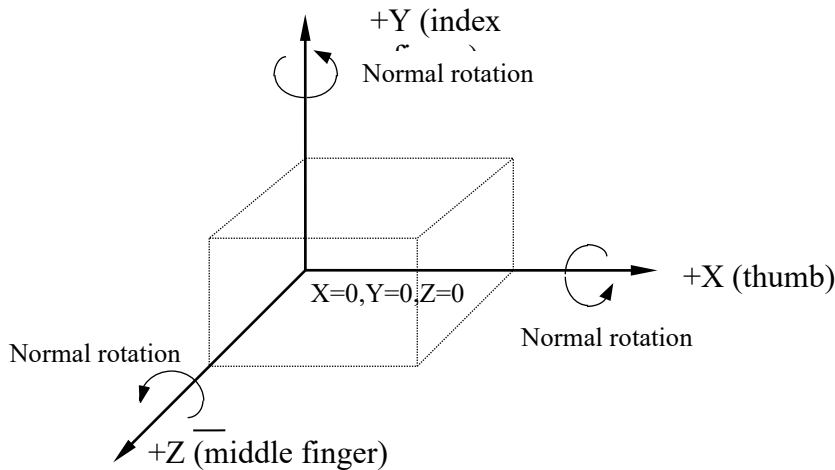


Fig 2-2 3-D Coordinate System (Milling Machine Type 1)

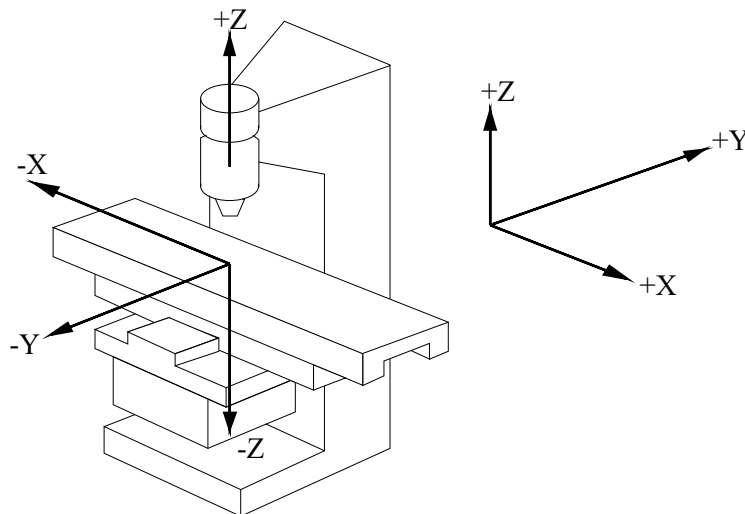


Fig 2-3 3-D Coordinate System (Milling Machine Type 2)

Note that in the 3-D system, the Z-axis must be parallel to the spindle axis. Once the direction of Z-axis is determined, the X and Y-axis can be determined by using the right-hand rule with the thumb pointing to the +X-axis. The fourth axis (B-axis) can be used as spindle axis to control RPM. All four axes X, Y, Z, and B can be used as linear axes or rotating axes for angle indexing.

#### 2.4.2 Coordinate of Tool Position Command

The instruction for tool position command in HUST 11-series can be in either absolute coordinate or incremental coordinate as follows:

- X, Y, Z, B : Absolute coordinate command. The cutting tool moves to the position specified by the absolute coordinate X, Y, Z, B.
- U, V, W : Incremental coordinate command. The cutting tool moves to the position with an incremental amount specified by U, V, W.

- Absolute Coordinate

The origin is the reference. The coordinates of all points describing the shape of the work-piece (machine part) are calculated from the origin. The coordinates can be positive (+) or negative (-), depending on its relative position with respect to the origin.

```

N10 G00 X0.000 Y0.000    -- Move to P0 work origin with high speed
N20 G90                 -- Set absolute coordinate
N30 G01 X12.0 Y12.0 F200. -- P0 to P1
N40 X26.0 Y16.0         -- P1 to P2
N50 X38.0 Y30.0         -- P2 to P3
N60 M2                  -- Program end

```

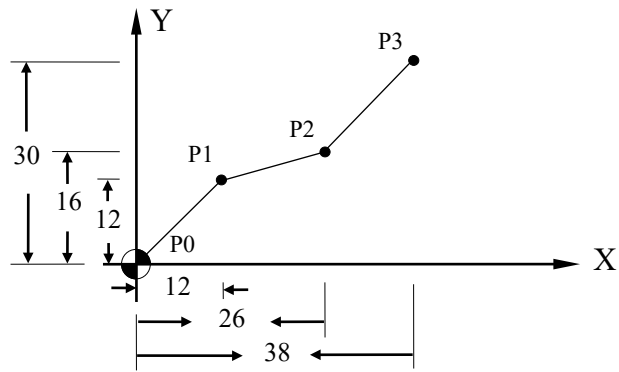


Fig 2-4 Absolute Coordinate

- Incremental Coordinate

The coordinates of all points describing the shape of the work-piece (machine part) are calculated from the end point of the previous block. They are the amount of coordinate increase from the last point. The incremental coordinates can be either positive (+) or negative (-), depending on its relative position with respect to the end point of the previous block. They are positive (+) if the cutting tool is going in the direction of the +X/+Z-axis, negative (-), otherwise.

```

N10 G00 X0.000 Y0.000    -- Move to P0 work origin with high speed
N20 G91                  -- Set incremental coordinate
N30 G01 X12.0 Y12.0 F200. -- P0 to P1
N40 X14.0 Y4.0           -- P1 to P2
N50 X12.0 Y14.0          -- P2 to P3
N60 M2                   -- Program end
    
```

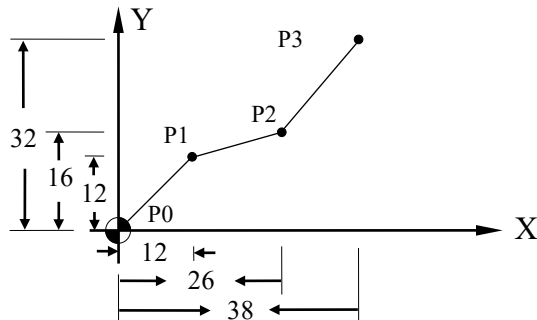


Fig 2-5 Incremental Coordinate

Note that the G91 command sets the X, Y, Z-coordinate as incremental values and the U, V, W will be rendered in-effective with G91. The power-on default coordinate is in absolute value, the above program can be re-written as:

```

N10 G00 X0.000 Y0.000    -- Move to P0 work origin with high speed
N30 G01 U12.0 V12.0 F200. -- P0 to P1
N40 U14.0 V4.0           -- P1 to P2
N50 U12.0 V14.0          -- P2 to P3
N60 M2                   -- Program end
    
```



The absolute and the incremental coordinate command can be used together in a single block program as shown in the example below.

```
N10 G00 X0.000 Y0.000    -- Move to P0 work origin with high speed
N30 G01 X12.0 V12.0 F200. -- P0 to P1
N40 X26.0 V4.0           -- P1 to P2
N50 X38.0 V14.0         -- P2 to P3
N60 M2                  -- Program end
```

Or

```
N10 G00 X0.000 Y0.000    -- Move to P0 work origin with high speed
N30 G01 U12.0 Y12.0 F200. -- P0 to P1
N40 U14.0 Y16.0          -- P1 to P2
N50 U12.0 Y30.0         -- P2 to P3
N60 M2                  -- Program end
```

In the absolute coordinate, the calculation error of one point will not affect the positioning of next point. In the incremental coordinate, however, an error of a point will affect the positioning of all subsequent points. There isn't any rule as to when to use the incremental or the absolute coordinate. The mixed use of both coordinates appears to be the most convenient.

### 2.4.3 Work Origin/Work Coordinate

The work origin is the coordinate origin as described before. It is also called the program origin. This is the reference point for all coordinate calculations and the coordinate so obtained is called work coordinate. The reason to call it as work origin is to differentiate it from the machine origin to be discussed in the next section.

The work origin can be anywhere inside the machine working range. The user should determine the location of this point before making any coordinate calculations. Once the origin is selected, store the coordinate of this point with respect to the machine origin in MCM parameter #1 (see Chap 7). The best selection is the one that will make the coordinate calculation simple and easy.

### 2.4.4 Machine Origin

The machine origin is the HOME location for the cutting tool. This is the reference point for the coordinate determination of the work origin and the tool offset compensation (see Chap 4). The coordinate obtained using the machine origin as calculation base is called the machine coordinate.

The exact location of the machine origin is determined by the location of the home limit switch on each axis. When user executes X-, Y-, Z-, or B-Home on a HUST CNC controller, the cutting tool will move to the machine origin. The exact distances between the machine origin and the work origin must be accurately measured using a fine instrument, such as a Linear scale. Otherwise, the completed part will be in an error.

When the electric power is interrupted for any reasons, execute HOME on each axis before resuming any cutting.

## 2.5 HUST M-11/I-11 Control Range

The minimum/maximum programmable range for HUST M-11/I-11 controller is as follows. Please note that the control range may be limited by the working range of user's machine.

	Metric, mm	English, inches
Min. setting unit	0.001	0.0001
Max. setting unit	8000.000	800.0000
Min. moving unit	0.001	0.0001
Max. moving unit	8000.000	800.0000
Min. setting time	0.1 seconds	0.1 seconds
Max. setting time	8000.0 sec.	8000.0 sec

	Metric Unit / English Unit
G-code	G00~G99 (The 1st 0 may be omitted)
M-code	M00~M99 (The 1st 0 may be omitted)
S-code	S1~S99999 rpm
F-code	0.01~80000.00 mm/min or 0.001~8000.000 in/min
X, Y, Z, U, V, W, I, J, K	0.001~±8000.000 mm or 0.0001~±800.0000 inches
R (Radius)	0.001~±4000.000 mm or 0.0001~±400.0000 inches
G04	0~8000.000 seconds
Program number	0~99
Memory capacity	20 K
Lead screw compensation	0~255 pulses (related to tool resolution)
Max. response speed	500 KPPS

### 3 PROGRAM DESIGN

This chapter will discuss the meanings and of command codes, such as G, F, M and S-code, and the format of their usage.

#### 3.1 G-code Definition

G-codes followed by one or two numbers are special command codes in HUST CNC system and they are from G00~G99. The first "0" can be omitted. Each G-code has its own specific function (see Table 3-1). G-code commands are divided into two groups:

##### 1. One-shot G-codes

A One-shot G-code is effective only in the program block where it was encountered. Once program starts executing the next block, it's no longer effective.

Example:

```
N10 G0 X30.000 Y40.000
N20 G4 X2.000      ....G04 is one-shot G-code, effective only in this block.
N30 X20.000 Y50.000  ....G04 no longer effective in this block. G0 is.
```

##### 2. Modal G-codes

A Modal G-code is a G-code that remains effective until another G-code in the same group is encountered. Following G-codes are in the same group for HUST M-11/I-11 controller.

G00, G01, G02, G03	Same group		
G17, G18, G19	Same group		
G40, G41, G42	Same group,	G43, G44, G49	Same group
G54~G59	Same group,	G68~G69	Same group
G80~G89	Same group,	G90~G91	Same group
G98~G99	Same group		
G92	A group by itself. <u>Repeated setting in the program can cause potential danger for the tool to run into the equipment.</u>		

Example:

```
N10 G0 X30.000 Y40.000  ....G0 is effective in this block.
N20 X40.000 Y5.000     ....No G-code specified, G0 remains effective.
N30 Y20.000           ....No G-code specified, G0 in N10 still effective.
N40 G1 X50.000 F200   ....G1 is effective from this block, NOT G0.
```

Normally, only one G-code is allowed in a program block. If several G-codes are accidentally specified in a block, only the last G-code specified is effective.

Example:

```
G00 G1 X10.000  ....Only G01 is effective.
```

Table 1-1 G-Code Definitions

G-code	Function
* 00	Rapid positioning (fast feed rate)
* 01 #	Linear cutting (cutting feed rate)
* 02	Arc cutting, CW (cutter at rear)
* 03	Arc cutting, CCW (cutter at rear)
04	Dwell (Temporary stop)
08	Clear machine coordinates in all axis
10	Data input
* 17 #	Spiral cutting, X-Y plane
* 18	Spiral cutting, Z-X plane
* 19	Spiral cutting, Y-Z plane
* 20	System measurement in INCH mode
* 21	System measurement in METRIC mode
28	Tool moves to the 1st reference point
29	Moves back to the specified position from the ref. point
30	Tool moves to the 2nd reference point (10 sets)
31 %	Skip function
* 40 #	Tool radius compensation - cancel
* 41	Tool radius compensation - set (left)
* 42	Tool radius compensation - set (right)
* 43	Tool offset compensation (+) direction
* 44	Tool offset compensation (-) direction
* 49	Tool offset compensation - cancel
* 50 %	Graphics proportional function - cancel
* 51 %	Graphics proportional function - set
* 54 #	The 1st work coordinate system
* 55	The 2nd work coordinate system
* 56	The 3rd work coordinate system
* 57	The 4th work coordinate system
* 58	The 5th work coordinate system
* 59	The 6th work coordinate system
* 68	Mirror effect cutting across X-axis
* 69	Mirror effect cutting across Y-axis
* 80 \$	Hole drilling canned cycle - cancel
* 81 \$	Hole drilling canned cycle - set
* 82 \$	Hole drilling canned cycle - temporary stop at bottom
* 83 \$	Deep hole drilling canned cycle
* 84 \$	Thread Tapping cycle (Stop at bottom then backout in reverse direction)
* 85 \$	Boring cycle
* 86 \$	Boring cycle (Spindle stop at bottom)
* 89 \$	Boring cycle (Temporary stop at bottom)
* 90	Absolute coordinate
* 91	Incremental coordinate
* 92	Absolute coordinate setting
* 98 \$	Return to starting point "S" in fixed canned cycle
* 99 \$	Return to point "R" in fixed canned cycle
<p>* -- G-codes with "*" are modal G-codes.  # -- G-codes with "#" are of power-on default setting.  \$ -- Special function for M-11, milling machine.  % -- Optional function.</p>	

### 3.2 Rapid Positioning Command, G00

Format: G00 X(U)\_\_\_\_ Y(V)\_\_\_\_ Z(W)\_\_\_\_ B\_\_\_\_

X, Y, Z : Position code in absolute coordinate.

U, V, W : Position code in incremental coordinate if absolute coord. system specified.

B : If B-axis set as a linear axis, add B\_\_\_\_ in the format

This code is used to move the cutting tool from the current location (or the end point of previous block) to the coordinate specified by X(U), Y(V), Z(W) at high speed while the tool is NOT physically doing any cutting. It can control the movement of 1~4 axes. The moving speed is based on the setting of MCM parameter #34. (See Chapter 7)

The position codes X(U), Y(V) and Z(W) associated with G00 command are the target coordinate for the cutting tool to move to. The starting location is the one prior to G00 command or the current position. The moving path will be the resultant of the incremental coordinate between these two points. The position code can be positive (+) or negative (-).

When calculating the moving speed on each axis, the controller will use the longest distance as a base. If the speed on one of the calculated value exceeds the MCM setting value, the controller will use that MCM value to re-calculate the moving speed for the rest of the axes. following is an example.

Caution -- To avoid damages to the cutting tool, be sure there is NOT any obstruction along the tool path when applying G00 command.

G00 Example:

```
G90
G00 X100.000 Y50.000 Z20.000
```

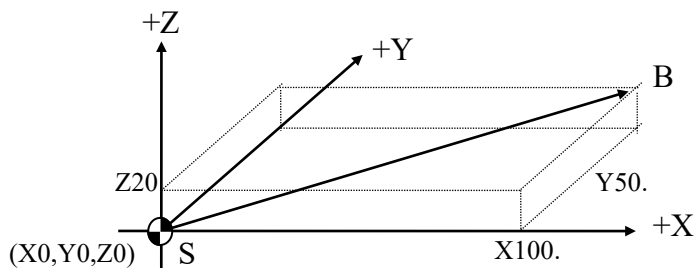


Fig 3-1 G00 Rapid Positioning

The target location is (100,50,20) with the start coordinate of (0,0,0). Therefore, the moving distances are 100 mm, 50 mm, and 20 mm in X, Y, and Z axis, respectively. Assume that the rapid feedrate of MCM parameter 34 is 10000.00 mm/minute. Then, the moving speeds along each axis are:

X-axis -- Since X-axis has the longest moving distance (100 mm), the moving speed will be 10000.00 mm/minute as set in MCM parameter #34.

Y-axis -- Moving distance is 50.00 mm. The moving speed along Y-axis will be proportionally calculated as  
 $(50 \text{ mm}/100 \text{ mm}) * 10000.00 = 5000.00 \text{ mm/minute}$ .

Feed-rate for Y-axis is 5000.00 mm/minute.

Z axis -- Moving distance is 20.000 mm. The moving speed along Z-axis will be proportionally calculated as

$$(20 \text{ mm}/100 \text{ mm}) * 10000.00 = 2000.00 \text{ mm/min}$$

Feed-rate for Z-axis is 2000.00 mm/min.

### 3.3 Linear Cutting Command, G01

Format: G01 X(U)\_\_\_\_ Y(V)\_\_\_\_ Z(W)\_\_\_\_ B\_\_\_\_ F\_\_\_\_

X, Y, Z : Position code in absolute coordinate

U, V, W : Position code in incremental coordinate

B : If B-axis set as a linear axis, add B\_\_\_\_ in the format

F : Cutting feed-rate

G01 is for the linear cutting motion and can control 1~3 axes at the same time. The cutting speed is determined by F-code. The smallest setting value for F-code is 0.02 mm/min or 0.2 in/min. F-code can be applied with any G-code including G00 block, but it does not affect G00 feed-rate.

The feed-rate (F-code) is a modal code. If the cutting rate is a constant for all program blocks, only one feed-rate in the beginning block needs to be defined. Unless the feed-rate is redefined, the F-code remains effective. The formula of cutting speed calculation for each axis is as below. U, V and W are incremental values.

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

$$\text{Feed-rate in Y-axis, } F_y = \frac{V}{\sqrt{U^2 + V^2 + W^2}} * F \quad (2)$$

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

Following is a G01 example written in 3 different forms that are doing the same linear cutting. The starting point is at X=0.0, Y=0.0, Z=0.0. (Fig 3-2)

#### 1. G90 absolute coordinate command

```
N1 G90
N2 G01 X25.000 Y20.000 Z10.000 F100.00    ... P1
N3 X60.000 Y50.000 Z40.000                ... P2
```

#### 2. G91 incremental coordinate command

```
N1 G91
N2 G01 X25.000 Y20.000 Z10.000 F100.00    ... P1
N3 X35.000 Y30.000 Z30.000                ... P2
```

## 3. G90 absolute command with incremental codes

```

N1 G90
N2 G01 U25.000 V20.000 W10.000 F100.00      ... P1
N3 U35.000 V30.000 W30.000                  ... P2

```

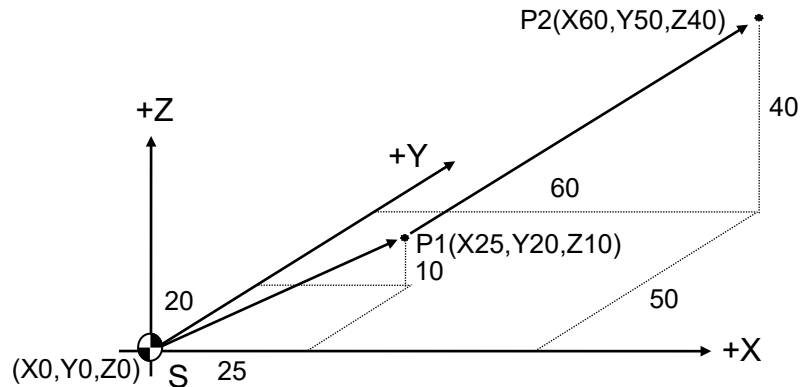


Fig 3-2 G01 Example

### 3.4 CNC and Master/Slave Mode (For I-11 Controller Only)

When the CNC controller executing a program block, the servo motor is always subjected to a motion sequence as: Acceleration--Constant feed-rate specified in the block--Deceleration. When the controller proceeds to execute the next block, the servo motor will repeat the same motion sequence. The CNC and Master/Slave mode for HUST I-11 controller is specially designed to govern how the acceleration/deceleration between blocks is to be connected.

#### CNC mode --

The servo motor will decelerate to a complete stop at the end of the first block, then the motor will accelerate to the feed-rate on the next block.

#### Master/Slave mode --

In this mode, the user select one axis as a Master axis and the rest will automatically become Slave axes. The acceleration/deceleration connection of motor speed between blocks will NOT come to a complete stop. Instead, the motors for both master and slave axes will decelerate or accelerate to the feed-rate of the next block from the feed-rate of the current block. The feed-rate (F) in the block is for the master axis and the feed-rate for the slave axes will be calculated according to their displacements. If the feed-rate for the master axis is zero (0), the feed-rate of the slave axis will be used for calculation.

MCM parameter #121 is for setting CNC and Master/Slave mode as follows:

- Setting=0, CNC mode
- Setting=1, X-axis as Master axis, the rest as Slave axes.
- Setting=2, Y-axis as Master axis, the rest as Slave axes.
- Setting=3, Z-axis as Master axis, the rest as Slave axes.
- Setting=4, B-axis as Master axis, the rest as Slave axes.

HUST 11-series system provides three (3) types of motor acceleration/deceleration, i.e. Linear, Exponential, and "S" curve. They are determined by the setting of MCM parameter #123 and will be explained in the examples below.

- **CNC mode:** MCM #121=0

MCM #121 setting	MCM #123 setting	G00 Acc./Dec. type	G1, G2 Acc./Dec. type
0	0	Linear	Exponential
0	1	"S" curve	"S" curve

Example 1: Fig 3-3, CNC mode, motor acceleration/deceleration in Exponential curve for G01 (MCM #123=0), Absolute coordinate.

```

N05 G00 X0. Y0. Z0.
N10 G01 X100. F1000.          --- Fx=1000., Fy=Fz=0
N20 G01 X200. Y100. Z50. F500. --- Fx=Fy=500., Fz=250.
N30 G01 X300. F250.          --- Fx=250., Fy=Fz=0.
N35 G01 X350. F100.          --- Fx=100., Fy=Fz=0.
    
```

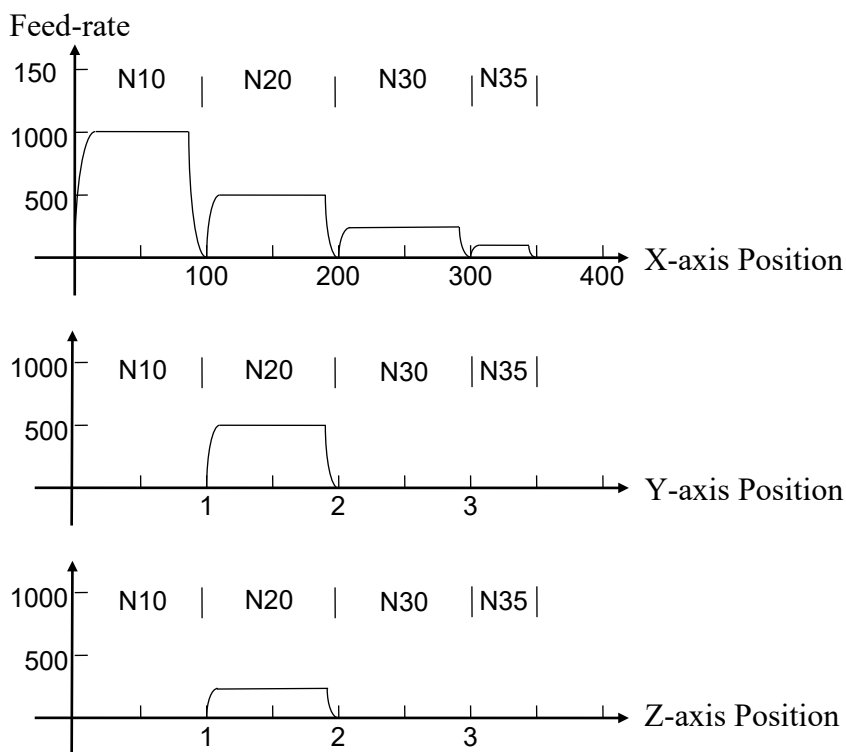


Fig 3-3 CNC mode with G01, Exponential Accel./Decel.



Example 2: G01 U100.0 V200.0 F2000.00 (CNC mode)

Calculate the G01 feed-rate for X and Y-axis. Assuming MCM #34 settings as F<sub>trx</sub>=2000.00 mm/min, F<sub>try</sub>=1000.00 mm/min.

Composite vector for X and Y-axis =  $(100^2 + 200^2)^{1/2} = 223.6$

X-axis Feed-rate F<sub>x</sub> = 2000.0 \* (100/223.6) = 894.4

Y-axis Feed-rate F<sub>y</sub> = 2000.0 \* (200/223.6) = 1788.9

1788.9 > 1000.0 (F<sub>try</sub>), so the G01 feed-rate will be limited as:

F<sub>x</sub> = (894.4/1788.9) \* 1000.0 = 500.00

F<sub>y</sub> = (1788.9/1788.9) \* 1000.0 = 1000.00

- **Master/Slave mode:** MCM #121=1, 2, 3, or 4

MCM #121 setting	MCM #123 setting	G00 Acc./Dec. type	G1, G2 Acc./Dec. type
1, 2, 3, 4	0	Linear	Linear
1, 2, 3, 4	1	"S" curve	"S" curve

Example 1: Fig 3-4, X-axis as master axis, motor accel./decel. in Linear type for G01 (MCM #123=0), Absolute coordinate.

Fig 3-4A, X-axis as master axis, motor accel./decel. in "S" type for G01 (MCM #123=1), Absolute coordinate.

Note that the feed-rate in each block is for master axis. The feed-rate for slave axes will be adjusted according to their coordinate increment with respect to the master axis.

```
N10 G01 X100. F1000.
N20 X200. Y100. Z50. F500.
N30 X300. F250.
```

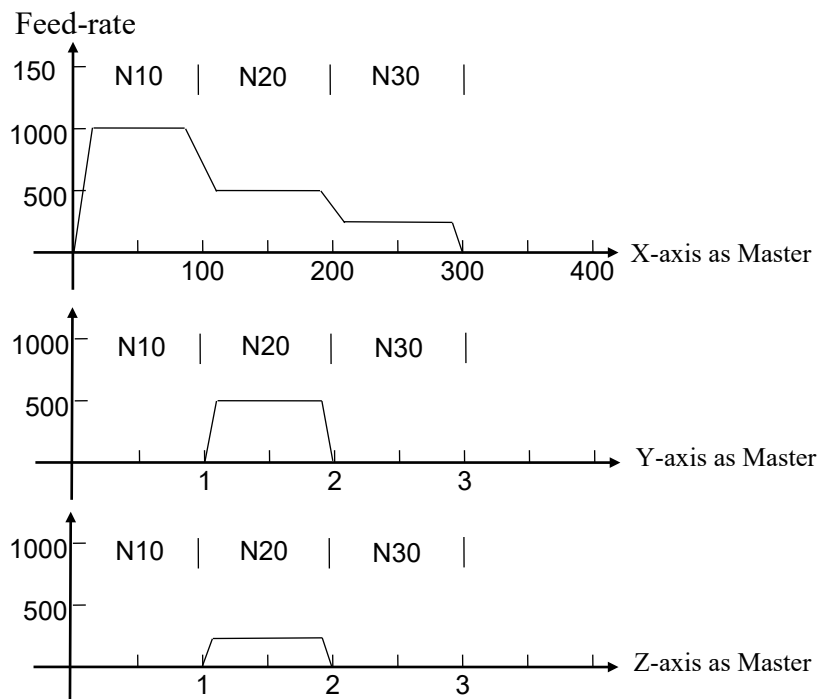


Fig 3-4 Master/Slave Mode, Linear Accel./Decel.

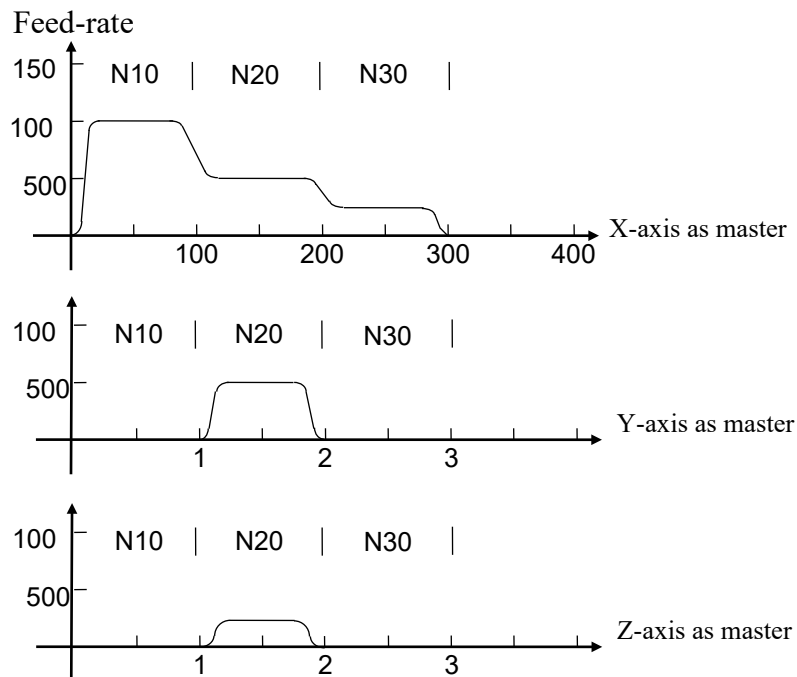


Fig 3-4A Master/Slave Mode, "S" Curve Accel./Decel.

Example 2: Fig 3-5, X-axis as master axis with constant feed-rate, motor accel./decel. in Linear type for G01 (MCM #123=0), Absolute coordinate.

```
N10 G01 X100. Y50. Z0. F1000.
N20 X200. Y75. Z50.
N30 X300. Y175. Z100.
```

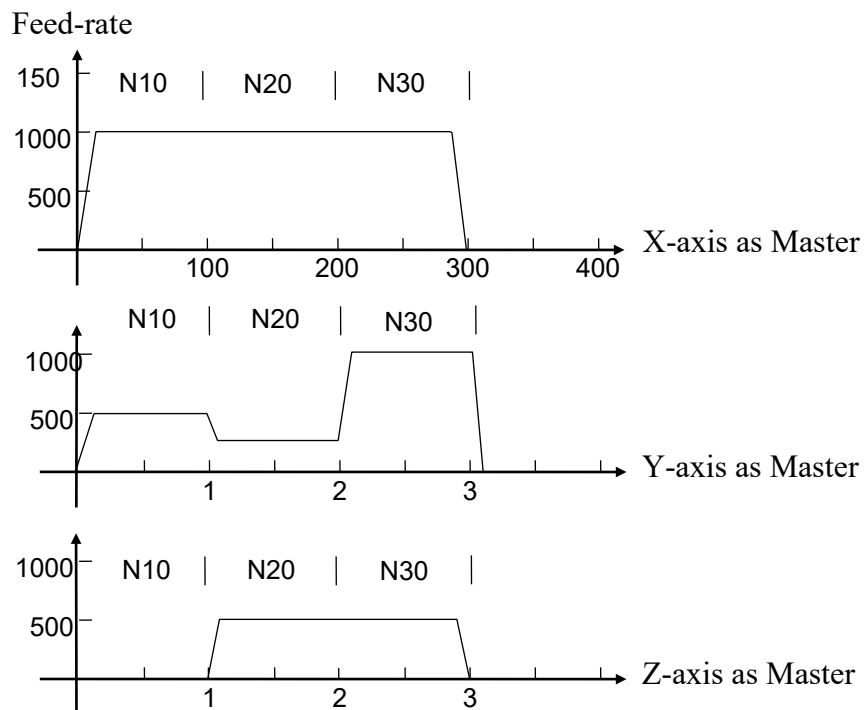


Fig 3-5 Master/Slave Mode with Constant F-rate for Master, Linear Accel./Decel.

In Example 2, the feed-rates of the slave axes are adjusted according to their incremental ratios. Note the small time required for accel./decel. between blocks and the distance traveled in this small amount of time can be estimated by:

$$\text{Distance} = 0.5 * \frac{(F1 - F2)}{60} * \frac{T}{1000}$$

F1, F2 = The feed-rates of 1st and 2nd block of the Slave axis  
T = The setting value of G01 in MCM #44

Therefore, if F1y=500. mm/min (N10 block), F2y=250. mm/min (N20) and MCM #44= 500 ms, the distance traveled at the beginning of N20 block for Y-axis is 1.04 mm. You can reduce this distance by reducing the setting of MCM #44.

Example 3: G00 U100.0 V300.0 (X-axis as master axis)

Calculate the G00 feed-rate for master and slave axis with MCM #34 settings  
as Ftx=2000.00 mm/min, Ftry=4000.00 mm/min.

Master axis      Fx = 2000.0  
Slave axis        Fy = (300/100)\*2000.00 = 6000.00  
Fy > Ftx, so the G00 feed-rate will be limited as:  
Master axis      Fx = (4000/6000)\*2000.00 = 1333.33  
Slave axis        Fy = 4000.00

### 3.5 Circular (Arc) Cutting Command, G02 And G03

Format:

X-Y plane:    G17      --- power-on default  
G02 (or G03) X(U)\_\_\_\_ Y(V)\_\_\_\_ I \_\_\_\_ J \_\_\_\_ F \_\_\_\_      (1)

X-Z plane:    G18  
G02 (or G03) X(U)\_\_\_\_ Z(W)\_\_\_\_ I \_\_\_\_ K \_\_\_\_ F \_\_\_\_      (2)

Y-Z plane:    G19  
G02 (or G03) Y(V)\_\_\_\_ Z(W)\_\_\_\_ J \_\_\_\_ K \_\_\_\_ F \_\_\_\_      (3)

The arc center variables I-J, I-K or J-K can be replaced by radius "R" and the arc cutting plane is determined by G17 (power-on default), G18 and G19. The arc cutting is the special case of the spiral cutting to be discussed in the next section.

Four (4) elements are required to do a circular cutting:

1. Arc cutting command code -- G02 or G03.

Depending on the arc cutting plane, the direction of circular cutting path is defined by command codes G02 (CW) and G03 (CCW) as in the figure below:

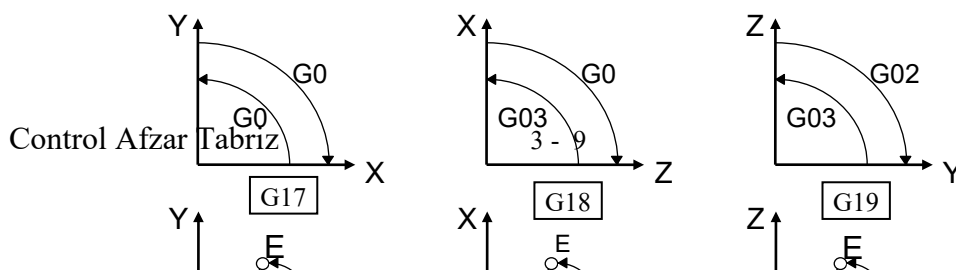


Fig 3-6 Directions of G02 and G03

2. The end point of the arc -- X(U), Y(V), Z(W).

U, V, W are the incremental coordinates from the start point (S) to the end point (E). The start point is the current position or the end point of the last block.

3. The center of the arc -- I, J, K or R

I, J and K are the X, Y, Z-axis components of the arc radius, respectively and R is the radius of the arc. Either representation is acceptable. I, J, K can be (+) or (-) and their meanings are identical to U, V, W. The range for "R" is -4000.~+4000. mm or -400.~+400. inches. Do not use R representation if the arc angle is in the range of  $-1^{\circ}\sim+1^{\circ}$  or  $179^{\circ}\sim181^{\circ}$ .

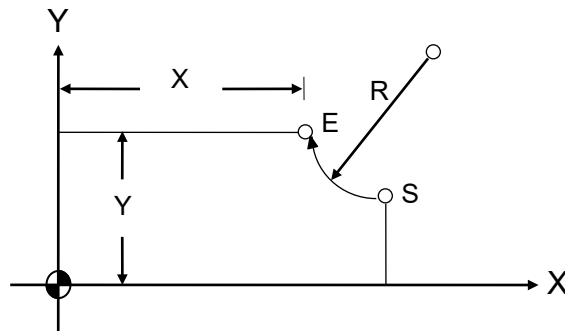


Fig 3-7 G2 Circular Cutting with "R" Specified

4. Arc cutting rate -- F-code

The minimum rate is 0.2 mm/min or 0.02 in/min.

When applying radius R method, be careful in determining the sign of radius R.

1. Use "+R" if arc angle  $< 180^{\circ}$ .
2. Use "-R" if arc angle  $> 180^{\circ}$ .

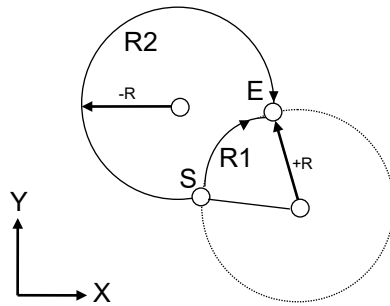


Fig 3-8 Arc Cutting with +R and -R

Example: The following four blocks will do the same arc cutting.

The start point X=50.0, Y=15.0

The end point X=30.0, Y=25.0

Radius R=25.0, or I=0.0, J=25.0

1. G02 X30.0 Y25.0 J25.0 F300.
2. G02 U-20.0 V10.0 J25.0 F300.
3. G02 X30.0 Y25.0 R25.0 F300.
4. G02 U-20.0 V10.0 R25.0 F300.

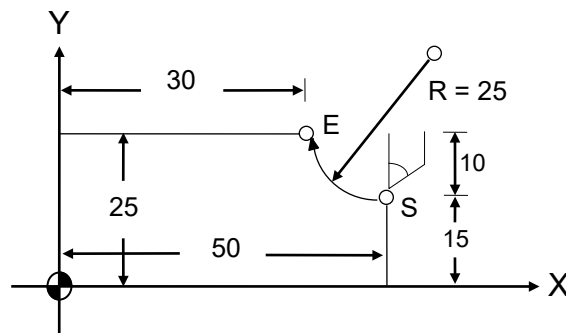


Fig 3-9 Arc Cutting Example

Notes on circular cutting:

1. G02, G03 command block must be followed by a G00 or G01 command block to signal to CNC the completion of the circular cutting. Otherwise, Error 25 will be displayed.
2. When cutting a circle, only the I, J, K method for arc center specification can be used. Radius "R" method will NOT yield satisfactory result.

Example: G90  
G00 X40.0 Y0.0  
G03 X40.0 Y0.0 I50.0 F100.0

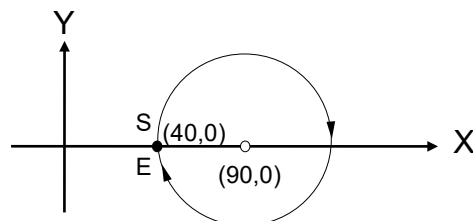


Fig 3-10 Cutting A Circle

3. The F-value is the tangential cutting speed at the cutting point, which will be affected by the length of the arc radius. The reason is that the HUST 11-system adopts a constant max. error of  $1 \mu\text{m}$  for arc cord height.

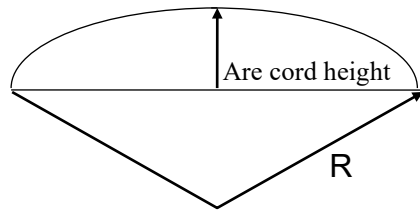


Fig 3-10A Arc Cord Height

4. When the calculated tangential cutting speed for the arc is greater than the programmed F-value, the programmed F-value will be used for the cutting. Otherwise, the calculated value will be used. The max. tangential cutting speed is estimated with the formula:

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

Where R= Arc radius in mm.

### 3.6 Spiral Cutting Command, G17, G18, G19

As seen in the last section, these commands stand alone in a block. The meaning is to execute a circular cutting on the specified plane by G17~G19 and do the linear cutting on the third axis. The tool radius compensation is only effective on the circular cutting plane. The following figure depicts the sign (+/-) convention of the three coordinates.

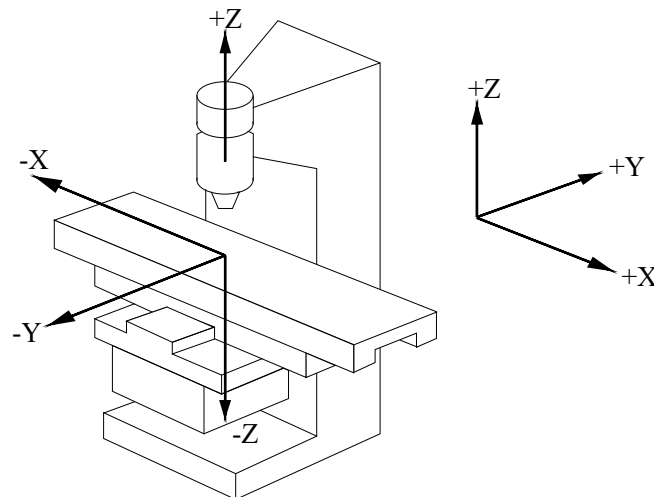


Fig 3-11 Sign (+/-) Convention Of The Three Coordinates (Right-hand Rule)

#### G17, X-Y Circular Cutting Plane

If you look down the machine from the top (along Z-axis), you have X-Y circular cutting plane and Z-axis as the linear axis.

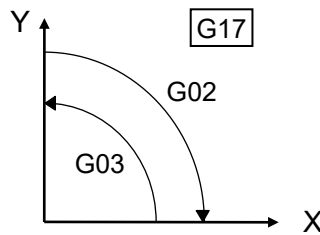


Fig 3-12

Format: (see Fig 3-6 and Fig 3-12)

G17  
G02 (or G03) X(U)\_\_\_Y(V)\_\_\_ I \_\_\_ J \_\_\_ Z \_\_\_ F \_\_\_ (1)

### G18, Z-X Circular Cutting Plane

If you look at the machine from the back along the Y-axis, you have Z-X circular cutting plane and Y-axis as the linear axis.

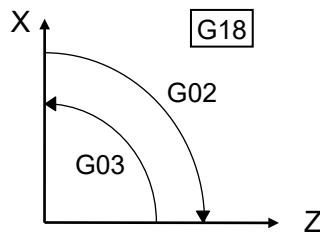


Fig 3-13

Format: (see Fig 3-6 and Fig 3-13)

G18  
G02 (or G03) X(U)\_\_\_Z(W)\_\_\_ I \_\_\_ K \_\_\_ Y \_\_\_ F \_\_\_ (2)

### G19, Y-Z Circular Cutting Plane

If you look at the machine from the right along the X-axis, you have Y-Z circular cutting plane and X-axis as the linear axis.

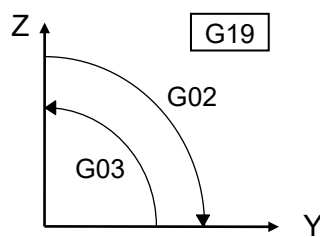


Fig 3-14

Format: (see Fig 3-6 and Fig 3-14)

G19  
G02 (or G03) Y(V)\_\_\_Z(W)\_\_\_ J \_\_\_ K \_\_\_ X \_\_\_ F \_\_\_ (3)

On all three formats, if the displacements are zero on the linear axes, you have circular cutting formats as shown in the last section. Note that the I, J, K representation can be replaced by "R" value also.

Example: N10 G17

N20 G03 X80.0 Y30.0 R30.0 Z40.0 F100.0

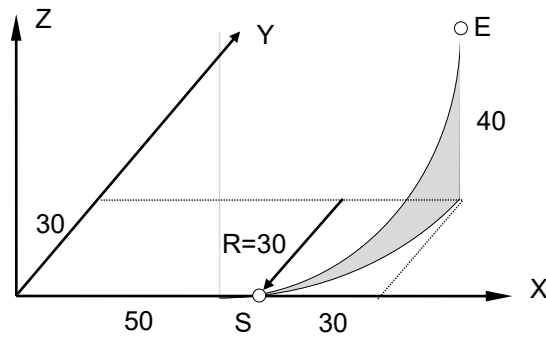


Fig 3-15

### 3.7 Temporary Hold (Stop, Dwell), G04

Format: G04 X\_\_\_\_\_

X: Holding period in seconds, ranging 0.01~8000.0 seconds.

Under some circumstances during cutting, it becomes necessary to hold (stop) the cutting action for certain period before proceeding to the next block. In this case, G04 function can be used for this purpose.

Example:

```
G01 X10.000 Y10.000 F100.
G04 X2.00          .... Hold for 2 seconds, then process to next block
G00 X0.000 Y0.000
```

### 3.8 Clear Machine Coordinates, G08

Format: G08 ..... Clear machine coord. in all axes  
 or G08 X\_\_\_\_Y\_\_\_\_Z\_\_\_\_B\_\_\_\_ ..... Clear machine coord. in X and Z-axis  
 or G08 any combination of X, Y, Z, B from 1~4 axes.

The number associated with X, Y, Z or B does not have any meaning, but you have to have a number to input X, Y, Z and B into the CNC buffer. When G08 is used in the program, the machine coordinates accumulated up to the G08 command for the specified axis (axes) will be cleared regardless of what numbers are with X, Y, Z and B-axis.

### 3.9 English/Metric Measuring System Conversion, G20, G21

Format: G20 -- System measuring unit in INCH mode  
 G21 -- System measuring unit in METRIC mode. This is the power-on and RESET default.



### 3.10 Move To The First Reference Point, G28

Format: G28 ..... Move to the 1st ref. point in all axes  
 or G28 X\_\_\_Y\_\_\_Z\_\_\_B\_\_\_ ..... Move to the 1st ref. point in X,Y, Z, B-axis  
 or G28 any combination of X, Y, Z, B from 1~4 axes.

The coordinates of the first reference point is set in the MCM parameter #60. The number associated with X,Y, Z, B does not have any meaning, but you have to have a number to input X,Y, Z, B into the CNC buffer. When encountering this command during cutting, the tool will move to the first reference point as set in MCM parameter #60 for the axis specified in the G28 block, regardless of what numbers are with X,Y, Z, B-axis.

The coordinates of MCM parameter #60 are determined by users, based on the machine origin being at X=Y=Z=B=0. This reference point is normally selected at some convenient location during machining. Therefore, if X=Y=Z=B=0 is selected for MCM #60, G28 command will cause the tool moving to machine origin.

Note that prior to the G28 command, the tool offset compensation must be canceled and the tool offset compensation cancel command should not be used in the same block as the G28 command.

Example:

```
G40      .... Tool offset compensation canceled
G28 X10. .... Tool moves to the 1st ref. point in X-axis, no motion in Y, Z, B-axis.
```

### 3.11 Return To The Previous Location From The Reference Point, G29

Format: G29 ..... Return from the ref. point in all axes  
 or G29 X\_\_\_Y\_\_\_Z\_\_\_B\_\_\_ ..... Return from the ref. point in X and Z-axis  
 or G29 any combination of X, Y, Z, B.

The G28 command moves the tool to the first reference point. G29 command works just the opposite. It moves the tool from the reference point to the last position, prior to G28 command, as indicated by X, Y, Z, B in the program block. G29 command can not be used alone, instead it is used following a G28 or G30 command. Again, the number associated with X, Y, Z, B does not have any meaning, but you have to have a number to input X, Y, Z, B into the CNC buffer.

Example: (only G29 in the block)

```
G01 X60.00 Y0.00 Z30.00 ..... Tool at the location of X60., Y0., Z30.
G28      .... Tool moved from (X60, Y0., Z30) to the 1st ref. point.
G29      .... Tool returns from the ref. point to (X60, Y0., Z30)
```

### 3.12 Move To The Second (2nd) Reference Point, G30

Format: G30 X\_\_\_ Y\_\_\_ Z\_\_\_ B\_\_\_ P\_\_\_

P: The group number (1~10) of MCM parameter #61

The method of application for this command is the same as for G28. The coordinates of this reference point are set in the MCM parameter #61 with total of 10 groups of ref. points.

### 3.13 Skip Function, G31

Format: G31 X(U)\_\_\_ Y(V)\_\_\_ Z(W)\_\_\_ B\_\_\_

For Skip function G31 to be effective, it must be used in combination with an input signal (I-18 for HUST 11-series) to be received during the execution of G31 block. Once an input signal is detected, the tool will forgo the unfinished operation of current block and starts executing the next block. If no input signal is received during the execution of G31 block, the tool will move to the coordinates as specified with the cutting speed as G01.

When G31 is doing linear cutting, the feed-rate will be the one in effect (G00 or G01). G31 is an one-shot G-code.

Example: N10 G1 X10. Z10.

```

.....
N40 G40
N50 G31 U100.0 F100.
N60 G1 V25.00
N70 X90.00 Y30.00
    
```

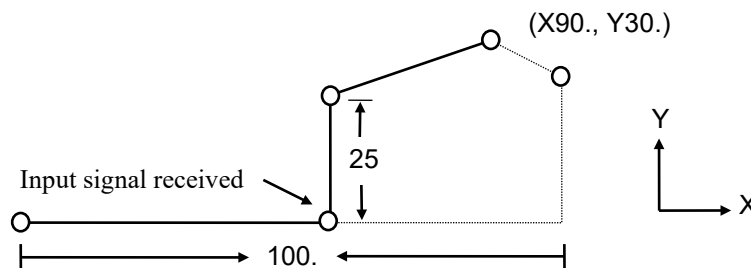


Fig 3-16 G31 Skip Function

In Fig 3-16, the dashed line represents the original path without SKIP function and the solid line is the actual tool path when the SKIP function signal is received. Prior to using SKIP function, do NOT use the tool compensation command.

### 3.14 Work-piece Dimension Enlarging and Shrinking, G50, G51

Format: G51 I\_\_\_ J\_\_\_ K\_\_\_ (Command set)  
 G50 (Command cancel)

I, J, K in this command are the proportionalities for the dimension to be enlarged or shrunk. Their ranges are 0.01~100 and the the max. and min. ratio in the block should be kept below 100. G51 can execute 1~3 axis at the same time and the dimension for the axis not specified will remain unchanged.

When using G51 command, it's better for the tool to return to the starting point, then use G50 to cancel it before doing other cutting. The reason is that G51 is a modal G-code and it remains effective unless G50 is used. The danger of this is that the tool may hit the travel limit and cause an accident, particularly if an enlarging scale is used.

Example:

```
N10 G1 X0.0 Y10.0 Z10.0
N20 G51 I2.0 J2.0    --- Dimension in X and Y-axes to be enlarged by 200%
N30 X40.0 Y40.0    --- Actual cutting X=80.0 mm, Y=60.0 mm
N40 G50
```

### 3.15 Work Coordinate System Setting, G92, G54~G59

There are two coordinate systems for HUST CNC machine tool. They are:

1. Machine Coordinate (Home)
2. Work Coordinate:
  - Basic Coordinate System (G92) -- To be set in the program
  - Work Coordinate System (G54~G59) -- To be set in the MCM parameters (Suggested method)

#### 3.15.1 Machine Coordinate (Home)

The origin of the machine coordinate system is a fixed point on the machine. Its location is normally determined by the locations of the over-travel limit switches (OTLS). When you execute HOME from the control panel, the tool or the machine table will move toward the OTLS, then reverse back to look for the encoder GRID signal. When it locates the GRID, the tool stops. This location is the HOME position or Machine origin. Machine origin is the calculation basis for all work coordinates and the reference point coordinates. Before you do any cutting, be sure to execute HOME.

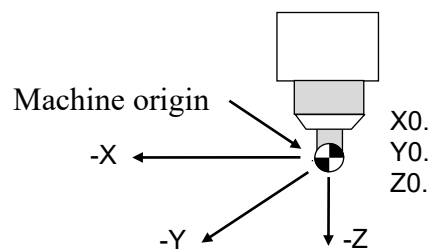


Fig 3-17 Machine Origin

Occasionally, for the convenience of doing cutting, it becomes necessary to set another origin that is slightly shifted from the machine origin. Such origin is called HOME SHIFT. The amount of shift is set in MCM #82. When you execute HOME, the tool will rest at the HOME position but the machine coordinate will show the home-shift values. If the setting

values in MCM #82 is zero (0), the HOME SHIFT is the HOME position. The methods to find HOME position are:

1. Manually execute HOME from the control panel. (See Chap 8)
2. Use G28 or G30 by setting the reference coordinates in the MCM to zero for all axes.

### 3.15.2 Work Coordinate System Setting, G92, G54~G59

#### Basic Coordinate System, G92

Format: G92 X\_\_\_ Y\_\_\_ Z\_\_\_ B\_\_\_

X, Y, Z, B : The coordinate of the current tool position with respect to the desired work origin location.

G92 is used to set indirectly the work origin in the program. This command specifies the coordinates (X, Y, Z, B) of the current tool position with respect to the desired work origin. All the coordinates in the program after G92 command are calculated based on this origin. Such coordinate system is called Basic Coordinate System. Followings are some examples to depict the usage of G92.

Example 1: Assume that the current tool position is at the machine origin.

N1 G92 X0.0 Y0.0 Z0.0	... Machine origin as work origin @ X0.Y0.Z0.
N2 G0 X-70. Y-40. Z-40.	... Fast move to P1
N3 G1 Y-20.0 F500.0.	... Linear cut to P2 @ 500mm/min
N4 X-40.000	... Linear cut to P3
N5 Y-40.000	... Linear cut to P4
N6 X-70.000	... Linear cut to P1
N7 G0 X0.0 Y0.0 Z0.0	... Fast move to work origin, i.e. machine origin
N8 M2	... Program end

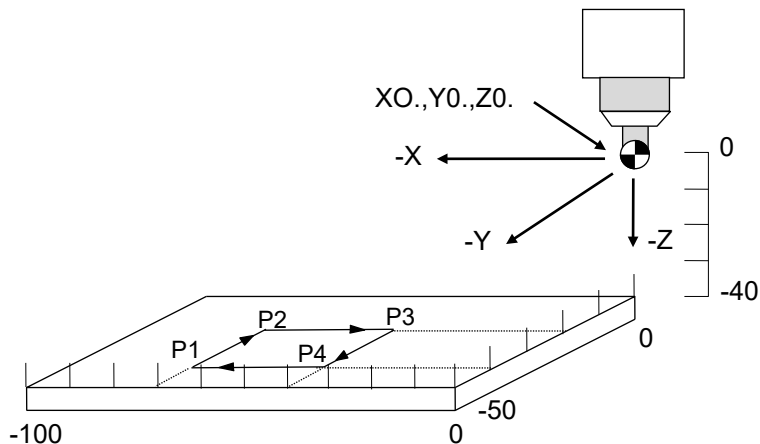


Fig 3-18 Machine Origin As Work Origin

Example 1 shows that the coordinate readings for P1~P4 present some difficulties when you use machine origin as work origin. If you use P1 as work origin, the coordinate calculations for P2~P4 will be lot more easier. Followings are two examples.

Example 2: The current tool position is at the machine origin. Use P1 as work origin.

```

N1 G0 X-70.0 Y-40.0 Z-40.0 ... Fast move to P1
N2 G92 X0.00 Y0.0 Z0.0 ... P1 set as work origin
N3 G1 Y20.000 F500.000 ... Linear cut to P2 @ 500mm/min
N4 X30.000 ... Linear cut to P3
N5 Y0.000 ... Linear cut to P4
N6 X0.000 ... Linear cut to P1
N7 G0 X70.00 Y40.00 Z40.00 ... Fast move to machine origin
N8 M2 ... Program end

```

Example 3 is the same as example 2 except that, in Example 3, you set the work origin when the tool is at the machine origin position.

```

N1 G92 X70.0 Y40.0 Z40.0 ... P1 set as work origin
      (X70, Y40, Z40 is the coord. of the current tool position w.r.t. P1)
N2 G0 X0.000 Y0.000 Z0.000 ... Fast move to P1
N3 G1 Y20.000 F500.000 ... Linear cut to P2 @ 500mm/min
N4 X30.000 ... Linear cut to P3
N5 Y0.000 ... Linear cut to P4
N6 X0.000 ... Linear cut to P1
N7 G0 X70.00 Y40.00 Z40.00 ... Fast move to machine origin
N8 M2 ... Program end

```

When applying G92 command, be aware of the ending location of your tool. If the end point is not at the starting point and you execute the program again, the resulting tool path will be different. This may cause some potential dangers for the tool to accidentally hit the travel limit. Therefore, unless it's absolutely necessary to use G92 in the program, HUST suggests the use of G54~G59 for the work origin setting. If you use G92, be sure to set the values of G54 in the MCM parameters to zero (0) to avoid any confusions.

### **Work Coordinate System, G54~G59 (Recommended method)**

Format: G54~G59 X\_\_\_ Y\_\_\_ Z\_\_\_ B\_\_\_

X, Y, Z, B : The coordinate (with respect to the specified work origin) for the current tool to move to.

HUST 11-series provides 6 sets of work origin system that are stored in the MCM parameter #1, in which the 1st line is the work origin coordinate for G54, the 2nd line for G55, and so forth. The coordinates of these work origins are the coordinates with respect to the machine origin. The coordinate data input into MCM #1 can be done by:

1. G10 command in MDI mode --- See Chap 8
2. Direct input in MCM mode --- See Chap 7

The application of these work origins in the program is accomplished by G54~G59 command as explained in the examples below. The advantage of using these work origins is the simplification of the coordinate calculations for the work-piece. Fig 3-19 shows six geographic cutting patterns with six work origins for G54~G59 as follows:

```

G54, coord. settings in the 1st line of MCM #1 -- X-70.0, Y-10.0
G55, coord. settings in the 2nd line of MCM #1 -- X-80.0, Y-30.0

```

G56, coord. settings in the 3rd line of MCM #1 -- X-80.0, Y-50.0  
 G57, coord. settings in the 4th line of MCM #1 -- X-70.0, Y-50.0  
 G58, coord. settings in the 5th line of MCM #1 -- X-40.0, Y-60.0  
 G59, coord. settings in the 6th line of MCM #1 -- X-20.0, Y-40.0

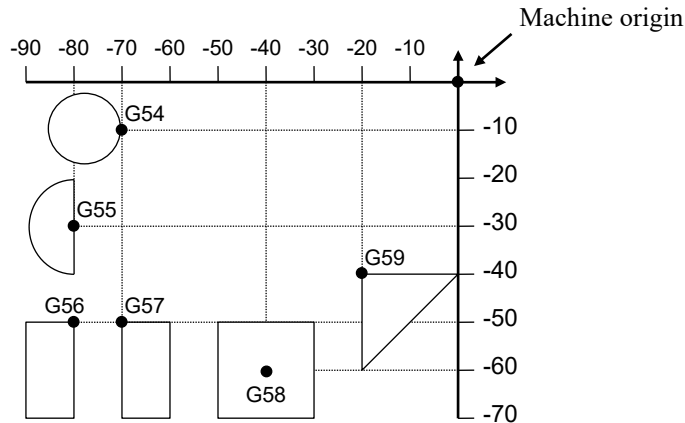


Fig 3-19 G54~G59 Work Origin

Once the work origins have been set in the MCM #1, the cutting patterns patterns can be accomplished using G54~G59 commands as shown below. Only G54 and G55 are shown in the example, but G56~G59 can be done the same way. Note that the program coordinates are also changed when the work origin is changed.

```

N1 G0 -- Feed-rate set at fast move mode
N2 G54 X0. Y0. -- Move to G54 work origin
N3 G2 I-7.0 F200.0 -- Cut a circle in CW with R=7.0
N4 G0 -- Feed-rate set at fast move mode
N5 G55 X0. Y0. -- Move to G55 work origin
N6 G1 V10.0 F300. -- Line cut along Y-axis @ 300 mm/min
N7 G3 V-20.0 R10.0 F300. -- Cut a half-circle in CCW with R=10.0
N8 G1 V10.0 F300. -- Line cut along Y-axis @ 300 mm/min
N9 G28 -- Move to the 1st reference point
N10 M2 -- Program end
    
```

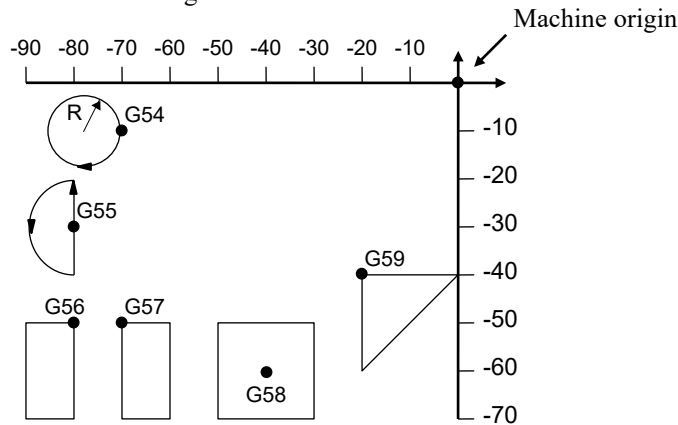


Fig 3-20 The Application of G54~G59

Notes on G54~G59 command:

1. Power-on default is the G54 command.
2. When using G54~G59 command, the associated X, Y, Z-values in the program block are the coordinates with respect to the specified work origin and the tool will move to that location.

### 3.16 Mirror-effect Cutting, G68, G69

Format: G68 -- Mirror-effect cutting, the X-axis as mirror

G69 -- Mirror-effect cutting, the Y-axis as mirror

The mirror-effect cutting is by using a sub-program to design a cutting pattern, then using G68, G69 command to accomplish the mirror-effect cutting as shown in Fig 3-21. In application, G68 or G69 forms a single program block. For G68, the sign of the X-coord. will be reversed ( $+ \rightarrow -$  or  $- \rightarrow +$ ) and for G69, the sign of the Y-coord. will be reversed. Therefore, all you have to do in cutting the patterns as in Fig 3-21 is writing a sub-program for pattern 1, then use G68 and G69 commands as follows:

```

M98 P___ -- Sub-program call to cut pattern 1 (+X, +Y)
G68      -- Reverse the sign for X-coord.
M98 P___ -- Sub-program call to cut pattern 2 (-X, +Y)
G69      -- Reverse the sign for Y-coord.
M98 P___ -- Sub-program call to cut pattern 3 (-X, -Y)
G68      -- Reverse the sign for X-coord.
M98 P___ -- Sub-program call to cut pattern 4 (+X, -Y)
G69      -- Reverse the sign for Y-coord and back to the position for pattern 1.
M2

```

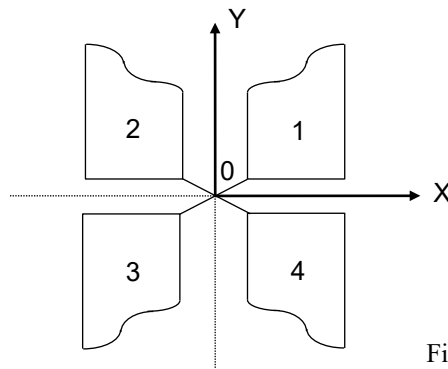


Fig 3-21 Mirror-effect Cutting

Note that G68 and G69 are modal G-code, you have to use one more G69 (or G68) as shown in the program above to reverse the coordinate sign back to normal. The mirror-effect cutting can also be cancelled by RESET key. When mirror-effect cutting is in effect, the "X-mirror" or "Y-mirror" will be shown at the top of CRT screen in reversed cursor.

### 3.17 Absolute and Incremental Coordinate Command, G90, G91

```

Format: G90 -- Absolute coord. mode, power-on default
        G91 -- Incremental coord. mode for X, Y, Z, B.

```

The absolute coordinate system (G90) is the power-on default. When in absolute mode, you can use the position command codes (X, Y, Z, or U, V, W) to obtain the desired absolute/incremental coord. descriptions. However, when G91 command is in effect, all X, Y, Z, B-coordinates represent incremental values and the U, V, W-codes are rendered ineffective.

Example 1: Absolute coord. setting (Fig 3-22)

```

N1 G90
N2 G1 X20.0 Y15.0 -- P0 to P1
N3 X35.0 Y25.0   -- P1 to P2
N4 X60.0 Y30.0   -- P2 to P3

```

Example 2: Incremental coord. setting (Fig 3-22)

```

N1 G91
N2 G1 X20.0 Y15.0    -- P0 to P1
N3 X15.0 Y10.0      -- P1 to P2
N4 X25.0 Y5.0       -- P2 to P3
    
```

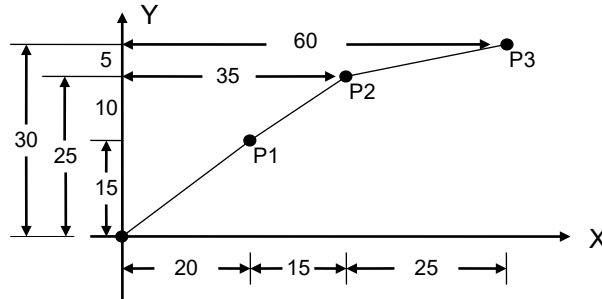


Fig 3-22 G90, G91 Example

**3.18 Canned Cycle Command, G80, G81~G89, G98, G99 (For M-11 Only)**

These G-code commands are for M-11 milling machine only, NOT for I-11 CNC.

HUST M-11 CNC controller provides several canned cycle functions to simplified the program design. All these canned cycle functions can be summarized as in Fig 3-23 below.

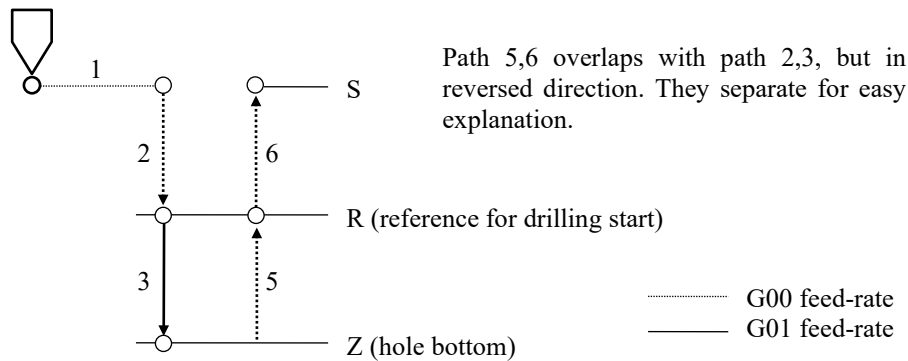


Fig 3-23 Canned Cycle Cutting Sequence

1. Fast move to the start (S) point on X-Y plane.
2. Fast move to the reference point for drilling start (R) along Z-axis.
3. Hole drilling to the bottom (Z) along Z-axis with G01 feed-rate.
4. Mechanical action at the hole bottom -- dwell or spindle rotation reversed.
5. Drill bit retracted to R-point. The moving speed depends on the command specified.
6. Fast move back to the start point S.

When applying the canned cycle function, you must use the followings:

- M03 -- Spindle rotation in the normal direction.
- M04 -- Spindle rotation in the reversed direction
- M05 -- Spindle stopped.

Format For Canned Cycle:



G90 or G91  
 G98 or G99  
 G81~G89 X\_\_\_Y\_\_\_Z\_\_\_P\_\_\_Q\_\_\_R\_\_\_F\_\_\_K\_\_\_  
 G80 or G00 or G01

- X, Y : Coordinate of hole location, absolute or incremental.
- Z : Coordinate of hole bottom, absolute or incremental.
- P : Dwell time at hole bottom, in 1/1000 seconds.
- Q : Amount of feed for each cut (for G83), in mm.
- R : Coord. of the tool reference point for drilling start or tool retraction, absolute or incremental.
- F : Feed-rate.
- K : Number of repetition for hole drilling.

Note that all the canned cycle commands, position codes, and the associated parameters are modal codes. Unless re-specified, they will remain effective each cycle.

1. G90 or G91 -- Absolute or incremental coord. setting

When in G90 absolute mode, R and Z-values are the coord. from Z=0. In G91 mode, they are increments from the start point S. Due to this, the resulting hole depths are different even with the same R, Z parameters, see Fig 3-24.

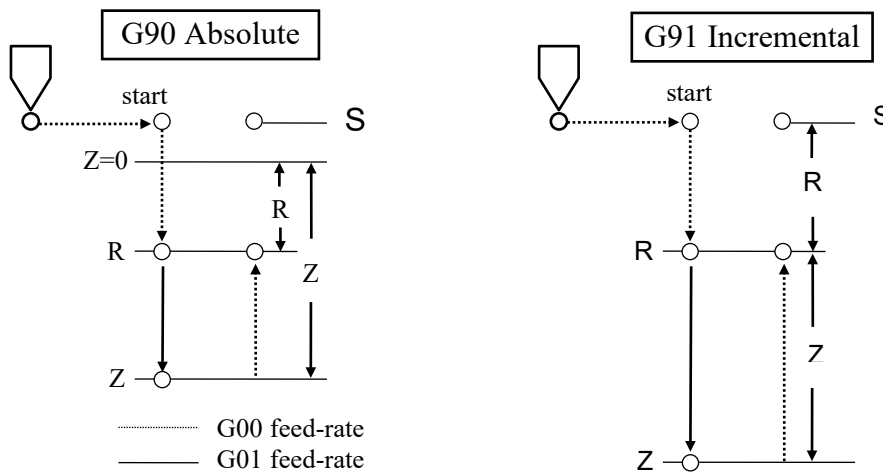


Fig 3-24 G90, G91 Effect On R, Z-value

2. G98 Or G99 -- Reference point setting of drill bit retraction

- G98 -- Drill bit retracted to the start point S.
- G99 -- Drill bit retracted to the drilling start point R.

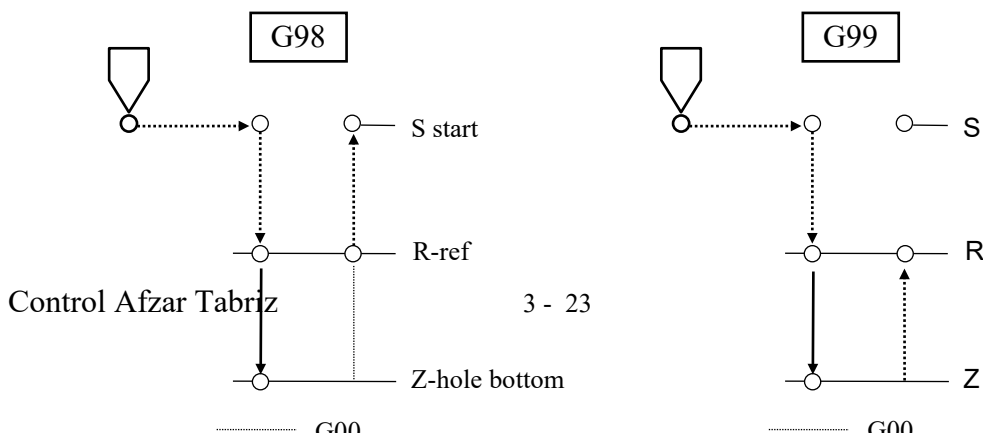


Fig 3-25 G98, G99 Setting

3. G81~G89 -- Canned cycle commands

The application of these commands is tabulated below.

	Application	Drill Rate	Action at Bottom	Retraction Rate
G81	Drilling	G01	--	G00
G82	Drilling	G01	Dwell	G00
G83	Deep hole drilling	G01	--	G00
G84	Thread tapping	G01	Dwell then spindle reversed	G01
G85	Boring	G01	--	G01
G86	Boring	G01	Spindle stopped	G01
G89	Boring	G01	Dwell	G01

4. G80 -- Canned cycle cancellation. All canned cycle commands must be cancelled by G80, G00 or G01.

G81 Drilling Canned Cycle

Format: G81 X\_\_\_ Y\_\_\_ Z\_\_\_ R\_\_\_ K\_\_\_ F\_\_\_

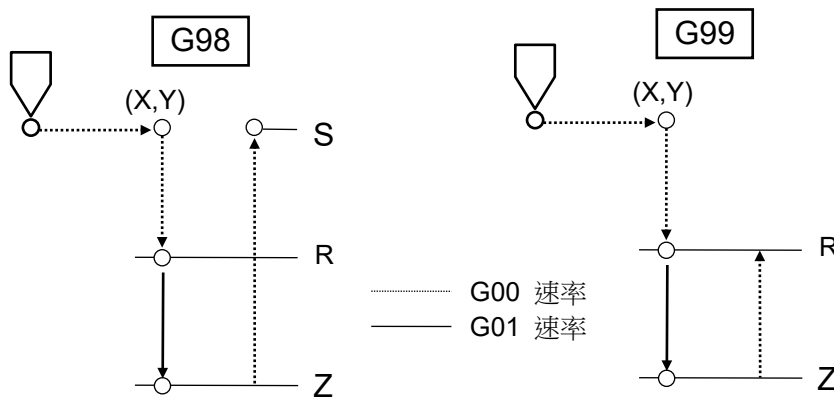


Fig 3-26 G81 Canned Cycle

G82 Drilling Canned Cycle -- Dwell at hole bottom

Format: G82 X\_\_\_ Y\_\_\_ Z\_\_\_ P\_\_\_ R\_\_\_ K\_\_\_ F\_\_\_

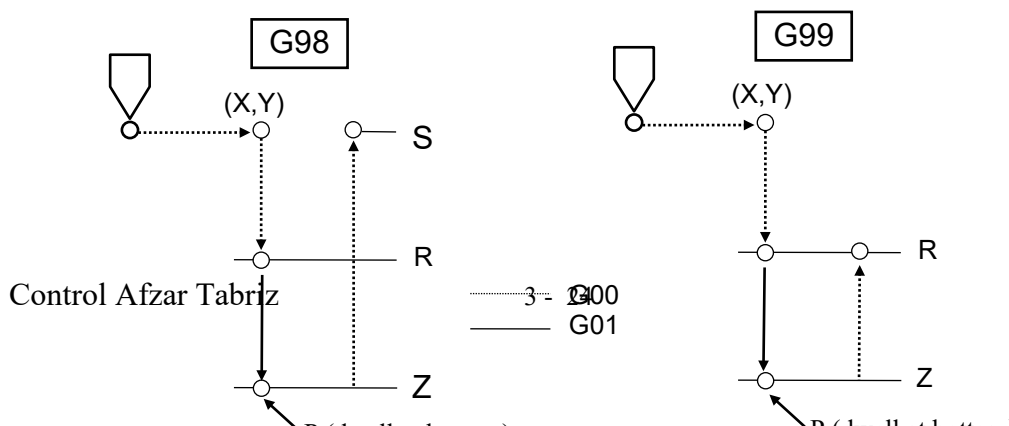


Fig 3-27 G82 Canned Cycle with Dwell at Hole Bottom

The difference between G81 and G82 is that G82 has a dwell time (P) when the drill bit reaches bottom. The dwell time is input as an integer in 1/1000 seconds.

### G83 Deep Hole Drilling Canned Cycle

Format: G83 X \_\_\_ Y \_\_\_ Z \_\_\_ R \_\_\_ Q \_\_\_ K \_\_\_ F \_\_\_

A deep hole cannot be finished in one forward action. Instead, it'll be finished in several up-and-down repetitions as specified by K. Q is the drilling depth for each repetition. The small "d" in Fig 3-28 is the location where the feed-rate is changed over from G00 to G01 and this number is set in MCM parameter #33.

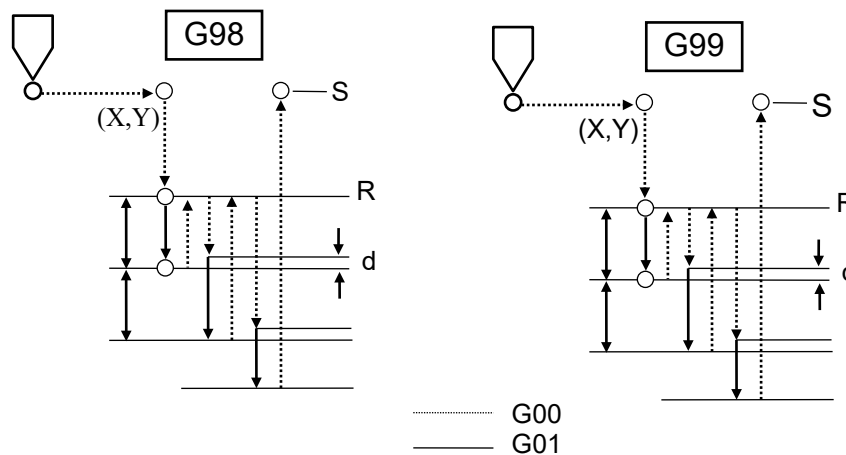


Fig 3-28 G83 Deep Hole Drilling Canned Cycle

### G84 Thread Tapping Canned Cycle

Format: G84 X \_\_\_ Y \_\_\_ Z \_\_\_ R \_\_\_ P \_\_\_ K \_\_\_ F \_\_\_

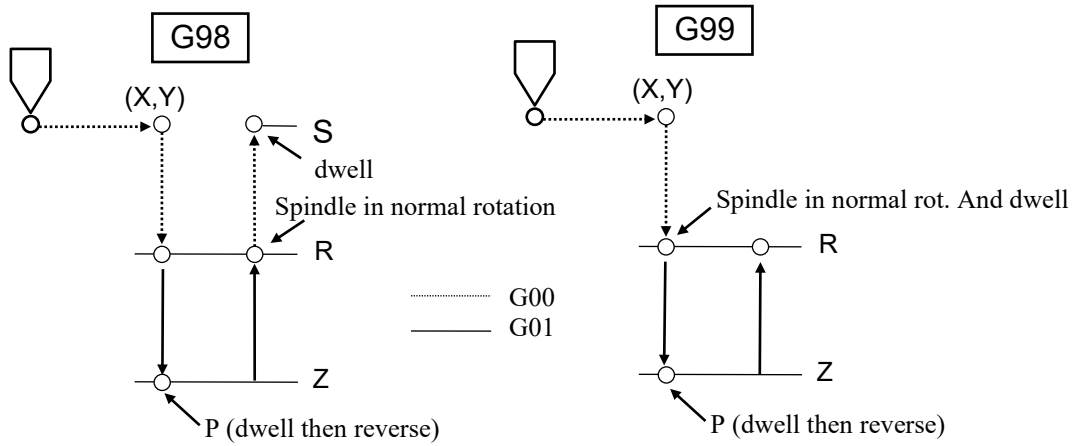


Fig 3-29 G84 Thread Tapping Canned Cycle

G85 Boring Canned Cycle

Format: G85 X \_\_\_ Y \_\_\_ Z \_\_\_ R \_\_\_ K \_\_\_ F \_\_\_

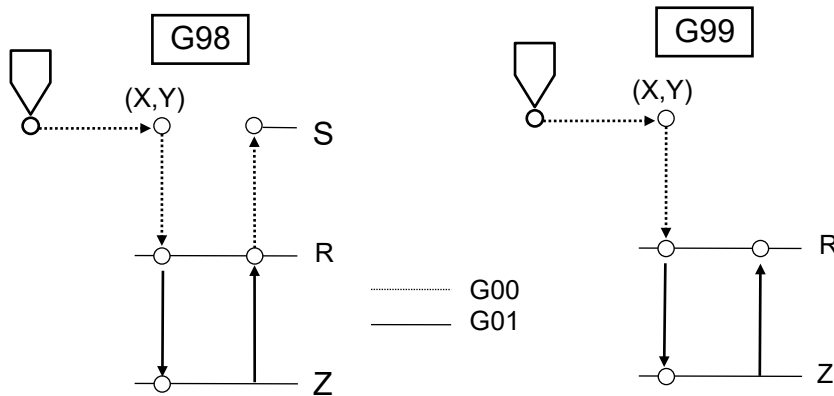


Fig 3-30 G85 Boring Canned Cycle

G86 Boring Canned Cycle with Spindle Stopped at Hole Bottom

Format: G86 X \_\_\_ Y \_\_\_ Z \_\_\_ R \_\_\_ K \_\_\_ F \_\_\_

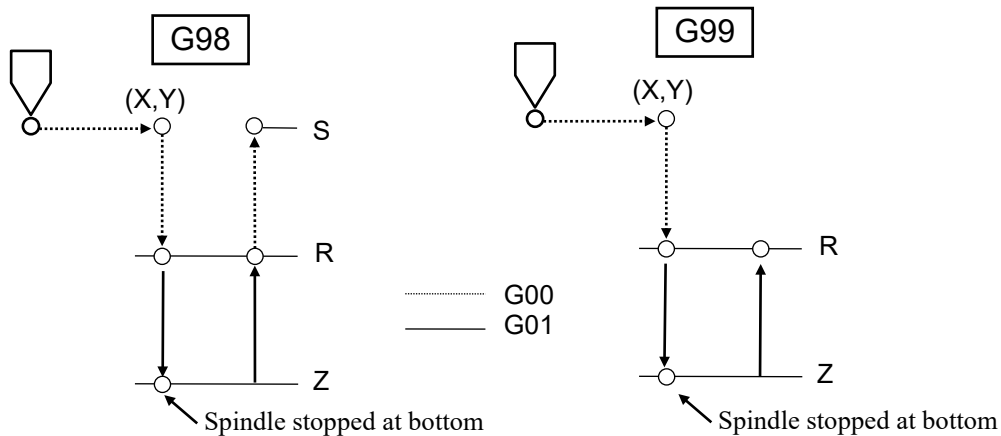


Fig 3-31 G86 Boring Canned Cycle with Spindle Stopped at Hole Bottom

The difference between G85 and G86 is that the spindle stops at hole bottom before the drill bit retracts.

G89 Boring Canned Cycle with Dwell at Hole Bottom

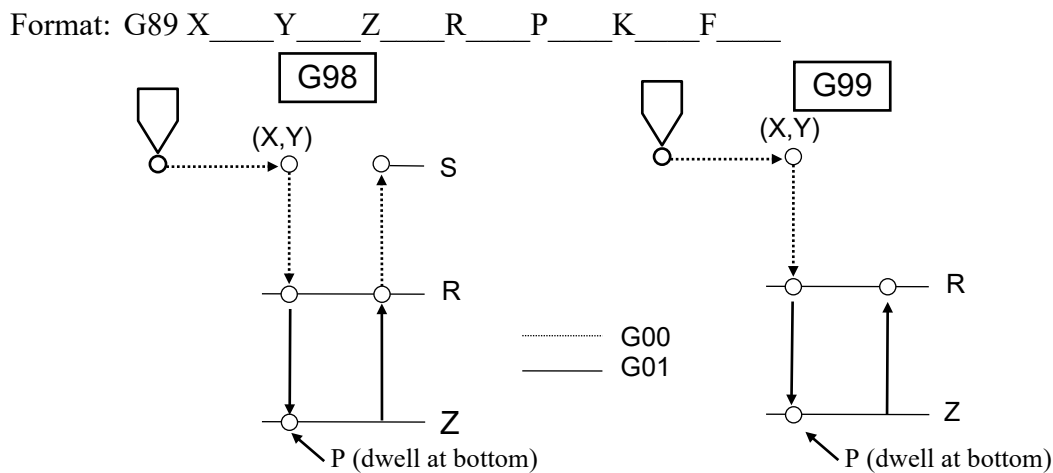


Fig 3-32 G89 Boring Canned Cycle with Spindle Dwell at Hole Bottom

The dwell time (P) is input as an integer in 1/1000 seconds.

### 3.19 Auxiliary Functions, M-codes, S-codes

HUST 11-series controller provides M-code functions for users to program certain mechanical actions outside the CNC controller. M-code function consists of a capital letter M followed by a 2-digit number, 00~99. Different M-code represents different action. The following M-codes are the ones used by HUST M-11 system and customers should not attempt to change them.

- M00 Program stop.  
When program execution comes to this point, all actions stop, including spindle and the coolant. Press "CYCST" to re-start from where the program was stopped.
- M01 Option stop.  
The program will stop only when the C-bit, C026=1. See Sec. 8.5 of Chap 8.
- M02 Program end.
- M30 Program end.  
M02 and M30 are identical.
- M98 Sub-program call.
- M99 Sub-program end.

Except those mentioned above, the remaining M-codes can be defined by users. The following M-codes come with the standard HUST M-11 PLC, but users can redefine these M-codes in the PLC if desired.

- M03 Spindle rotation in normal direction.
- M04 Spindle rotation in reversed direction.
- M05 Spindle rotation stops.
- M07 Coolant ON.
- M09 Coolant OFF.

The S-code is used to control the rpm of the spindle rotation. The max. setting is S999999.

Example: S1000 ..... The spindle rpm is 1000 rev/min

For method of decoding the number that associates with M-code and S-code, please refer to Chap 6 of Connecting Manual.

### 3.20 Sub-program

When a group of program steps will be used repeatedly, these program steps can be grouped in a sub-program that can be called out for execution whenever is required from the main program. In doing this way, the structure of the program can be greatly simplified. The structure of the sub-program is pretty much the same as the main program except that the sub-program is ended with a M99 as follows:

```
O005      Program number (No 5 in this case)
.....    Program steps
M99      Program end
```

The sub-program can be independently executed by pressing the "Auto" and "CYCST" button. However, the execution will go round and round to a max. of 8000 times because the sub-program is ended with a M99 function.

#### Execution of a sub-program from a main program

Format: M98 P\_\_\_ L\_\_\_

P : Sub-program number  
 L : Number of execution. If not specified, execute once.

Example: M98 P05 ..... Execute sub-program No 5 once.  
 M98 P05 L3 ..... Execute sub-program No 5 three times.

The M98 block can not contain any position code, such as X or Z except those shown in the format. A sub-program can call another sub-program. This stepwise sub-program call can go up to a max. of 5-level for HUST 11-series controller as below:

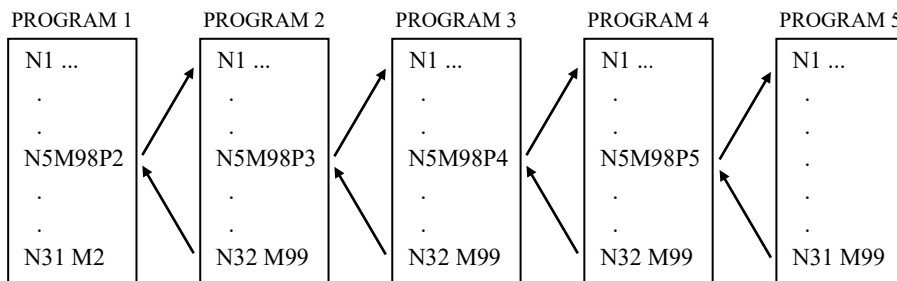


Fig 3-33 Sub-program Call

## 4 TOOL COMPENSATION FUNCTIONS

In HUST M-11/I-11 system, the tool compensation is divided into three categories and the data are stored in MCM parameters #20, #21, and #22. Each item can store up to 16 sets of data. They may be called out for compensation in the program by G41~G44.

### 1. Tool radius compensation -- by G41, G42

The tool-tip is normally made in a circular shape and the radius compensation function is used to minimize the cutting error due to the radius of the tool-tip. The compensation is normally made in the X-Y plane and the compensation data are stored in MCM #20.

### 2. Tool wear compensation -- by G41, G42

This function is used for the cutting error due to the wear on the tool-tip and is used in combination with tool radius compensation in the X-Y plane. The compensation data are stored in MCM #21.

### 3. Tool offset (length) compensation -- by G43, G44

If more than one cutting tool is used for a work-piece, the length difference (Z-axis) from the standard tool can be compensated using offset compensation function. The compensation data are stored in MCM #22.

For compensation data input and storage in MCM parameters, please refer to Sec. 7.1 of Chap 7 and Sec. 8.2.3 of Chap 8.

## 4.1 Tool-tip Radius and Wear Compensation, G40, G41, G42

Format: G41 D      X      Y      ..... Compensation to the left of the tool path  
 G42 D      X      Y      ..... Compensation to the right of the tool path  
 G40 ..... Compensation cancel

D : Tool group number (1~16) in MCM #20, #21. D0=D1.

X, Y : The coordinate where the radius compensation starts

The tool number or group number must be declared before setting compensation function in effect. The direction of compensation is specified by the command codes, G41 or G42. G41 causes the tool to compensate (move) to the left of the program path and G42 does to the right. The direction of compensation (right or left of the tool path) is shown in Fig 4-1. Use G41 if the compensation tool path falls to the left of the program path and use G42 if falls to the right. Since the arrow can point to any direction, be careful about the compensation direction when applying. G41 and G42 are modal codes and must be cancelled by G40.

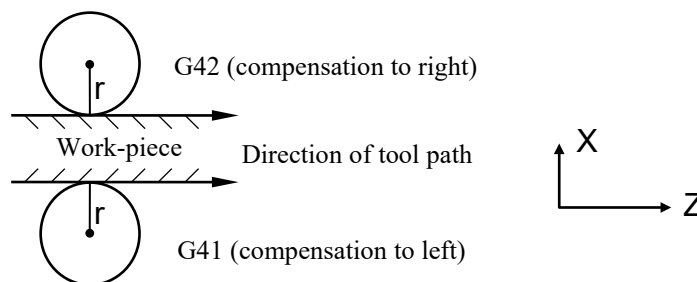


Fig 4-1 Application for G41 and G42

### Tool-tip Wear Compensation

The application of tool-tip wear compensation is identical to that of radius compensation. When you use G41 or G42 with "D" code to call for radius compensation, HUST controller will select both the radius (MCM #20) and the wear (MCM #21) compensation and add them together before making any compensation.

Example: Assuming that the compensation data for tool number 3 are  
 Radius comp. = 2.000 mm, Radius wear comp. = -0.010 mm  
 The combined radius comp. = 2.000 - 0.010 = 1.990 mm

Note that the radius wear compensation is a negative number in MCM #21. Both the radius and the wear compensations are valid only in the X-Y plane, NOT in the Z-axis.

### 4.1.1 Start of Tool-tip Radius Compensation

When executing a program block with G41 or G42 command, the tool will move with a G01 speed to the location specified by X, Y. When the tool reaches X, Y, the tool-tip will be offset by an amount equal to the tool radius (or the combined radius). From this point on, the tool radius compensation is in effect. Therefore, the tool-tip radius compensation must be started in a block with G00 or G01 linear motion. An error will result if user attempts to start radius compensation in a block with G02/G03 command. Followings are some examples showing the start of tool-tip radius compensation.

Example 1 -- Inception of radius comp. during A~B and full comp. starting at B.

```
N1 G01 X___ Y___ F___      -- Point A
N2 G41 (G42) D___ X___ Y___ -- Point B
N3 X___
```

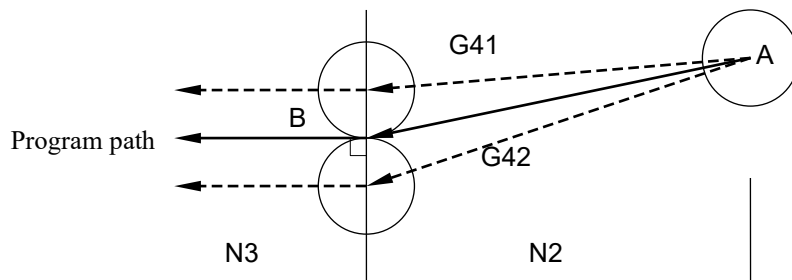


Fig. 4-2 Start of Tool-tip Radius Compensation - 1

Example 2 -- Inception of radius comp. during A~B and full comp. starting at B.

```
N1 G01 X___ Y___ F___      -- Point A
N2 G41 (G42) D___ X___ Y___ -- Point B
N3 G02 X___ Y___ J___
```

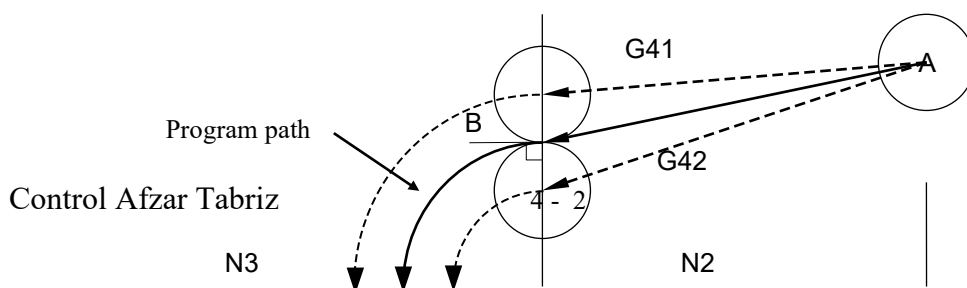




Fig. 4-3 Start of Tool-tip Radius Compensation - 2

### 4.1.2 The Relationship Between The Radius Compensation and The Tool Path

As mentioned before, when G41 or G42 is in effect, the center of the tool will shift by an amount equal to the combined radius compensation. This will be all right if the cutting path is a straight line or a continuous smooth curve. If the tool path is formed by two lines with a sharp angle, an un-cut portion may result. Followings are some examples.

1. Internal angle --

A small portion at the intersecting point P cannot be reached by the cutting tool.

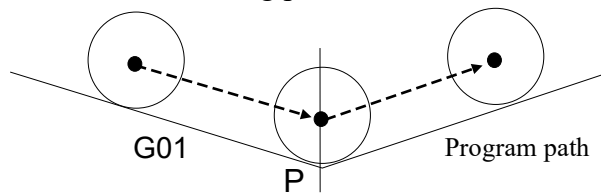


Fig 4-4

2. External angle --

When the tool comes to the intersection P of an external angle, the tool will make a circular movement (A~B) to rearrange its new cutting direction. In this case, the cutting is correct.

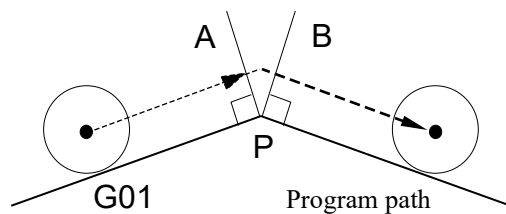


Fig 4-5

If the angle is a wedge-shaped sharp angle, the cutting on the external side will be correct. The cutting on the internal side, however, will depend on the size of the opening "C". If "C" is smaller than the tool radius, the tool will not be able to get in. Even "C" is greater than the tool radius, a big portion of this internal angle will remain un-cut.

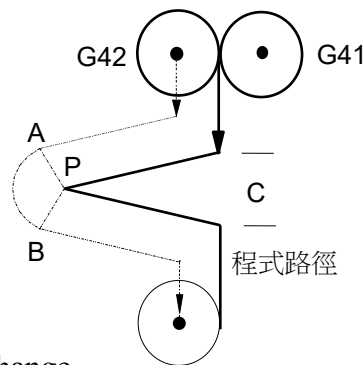


Fig 4-6

3. Compensation direction change --

HUST M-11/I-11 will NOT accept the compensation direction change directly from G41 to G42 or from G42 to G41. If you have to change the compensation direction in the program, be sure to use G40 to cancel it first then change the direction.

4. Compensation radius (tool number) change --  
 HUST M-11/I-11 will NOT accept the compensation radius (i.e. tool number) change while another tool number is in effect. If you have to change the tool number, you have to use G40 to cancel the first tool radius compensation then change the tool number.

**4.1.3 Cancel of Tool-tip Radius Compensation**

Once G41 or G42 has been set in effect, G40 command must be used to cancel the tool radius compensation. The cancellation of G40 has to be done in a linear motion mode, such as G00 or G01, but NOT in the block with G02/G03. G40, G41, and G42 are all modal G-codes. Followings are some examples illustrating the application of G40 command.

```

1. N20 G41 (G42) D ____ .....
   .....
   N31 G01 Z ____ F ____
   N32 G40 X ____ Y ____
    
```

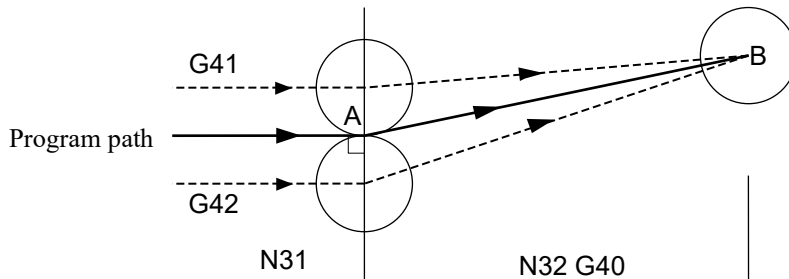


Fig 4-7 Cancel of Tool-tip Radius Compensation - 1

```

2. N10 G41 (G42) D ____ .....
   .....
   N15 G02 X ____ Y ____ I ____ J ____ F ____
   N20 G01
   N25 G40 X ____ Y ____
    
```

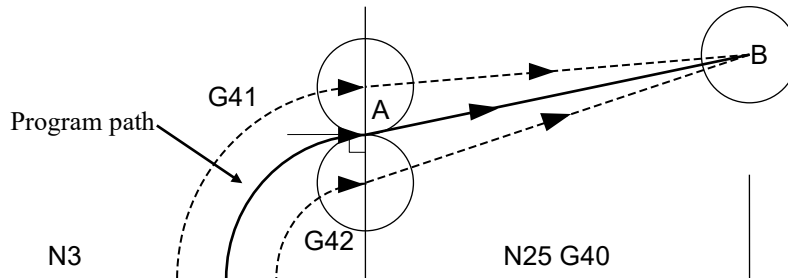


Fig 4-8 Cancel of Tool-tip Radius Compensation - 2

#### 4.1.4 Notes On Tool Radius Compensation

1. When cutting around an inside corner (arc or circle), the arc radius of inside corner must be equal to or greater than the tool-tip radius. Otherwise, an error will result. Cutting around an outside corner is, however, NOT subject to this type of restriction.
2. The insertion (start) of the tool-tip radius compensation should be done in a block with G00 or G01 command.
3. Do not use tool-tip radius compensation in an MDI mode or in G80~G89 canned cycle function.
4. The use of radius "R" method in an arc cutting is permissible when applying G41 or G42 command.
5. The radius compensation is only effective in the X-Y plane, NOT in the Z-axis.
6. When machining a step-wise work-piece with step size smaller than the tool radius, over-cutting will be produced as shown in Figure 4-15.

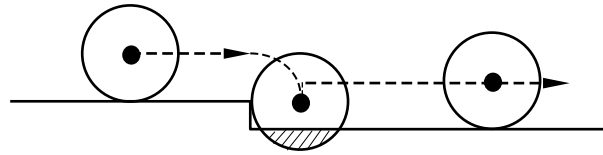


Fig 4-9 Over-cutting (Shaded area)

#### Example -- Tool Radius Compensation

N1 G91	... Incremental coord. setting
N2 G01 Z-2.500 F150.	... Z-axis, depth of cut = 2.5 mm
N3 G17 F300.	... X-Y cutting plane setting
N4 G41 D10 Y30.000	... A, tool radius compensation start
N5 Y100.000	... A~B linear cutting
N6 X30.000 Y40.000	... B~C linear cutting
N7 G02 X100.000 I50.000	... C~D circular cutting
N8 G01 X30.000 Y-40.000	... D~E linear cutting
N9 Y-100.000	... E~F linear cutting
N10 X-40.000	... F~G linear cutting
N11 G03 X-80.000 R50.000	... G~H circular cutting
N12 G01 X-70.000	... H~I linear cutting
N14 Z2.500	... Z-axis, tool raised by 2.5 mm
N15 G40	... Compensation cancelled for comp. direction change
N16 M01	... Option stop
17 G0 X130. Y90. F200.	... Move to point N
N18 G01 Z-2.500 F150.	... Z-axis, depth of cut = 2.5 mm
N19 G42 Y-40.000 F300.	... N~O linear cutting
N20 X-60.000	... O~J linear cutting
N21 Y30.000	... J~K linear cutting
N22 G02 X80.000 I40.000	... K~L circular cutting

```

N23 G01 Y-30.000    ... L~M linear cutting
N24 X-60.000       ... M~P linear cutting
N25 Z2.500         ... Z-axis, tool raised by 2.5 mm
N26 G40 X-60.00 Y-80.00 ... Radius comp. cancel, move to start point S
N27 M02            ... Program end
    
```

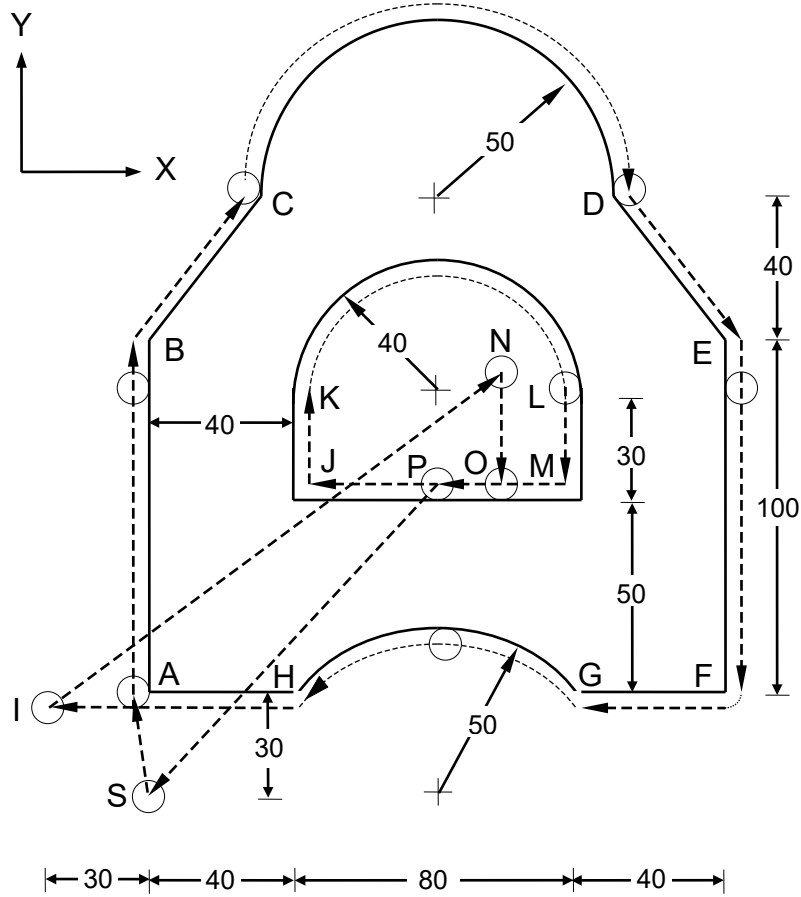


Fig 4-10 Radius Compensation Example

### 4.2 Tool Offset Compensation, G43, G44, G49

If more than one cutting tool is used for a work-piece, the length difference in the direction of Z-axis can be compensated using offset compensation function. The compensation data are stored in MCM #22 for 16 sets of tool.

```

Format: G43 (or G44) Z____ H____    -- Compensation set
        or G43 (or G44) H____        -- Compensation set
        G49                          -- Compensation cancel
    
```

Z : Coordinate of offset compensation start  
H : Tool number (1~16), H0 = H1

Use G43 if the compensation data in MCM #22 is to be added directly to the Z-axis.  
Use G44 if the sign (+/-) of the compensation data in MCM #22 is to be reversed, then added to the Z-axis.

The direction (+/-) of offset compensation depends on the direction of tool movement in the Z-axis. If the tool moves in the +Z direction after compensation, it's a positive compensation. If the tool moves in the -Z direction after compensation, it's a negative compensation.

	MCM #22 data, "+"	MCM #22 data, "-"
G43	Positive comp.	Negative comp.
G44	Negative comp.	Positive comp.

Example 1:

```

N1 G00 Z0.000
N2 G0 X1.000 Y2.000
N3 G43 Z-20.000 H10 (MCM #22, Tool #10=-3.000)
N4 G01 Z-30.000 F200
N5 G49 Z0.000
    
```

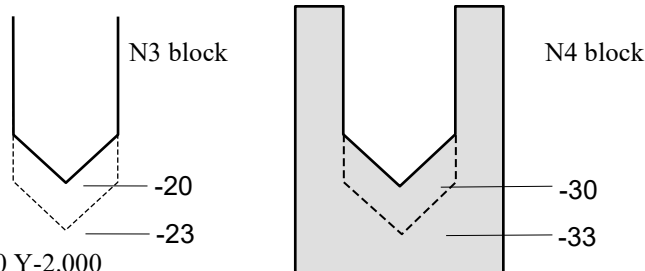


Fig 4-11

Example 2:

```

N1 G00 X-2.000 Y-2.000
N2 G44 Z-30.000 H1 (MCM #22, Tool #1=4.000) (after comp.)
N3 G01 Z-40.000
N4 G49 Z0.000
    
```

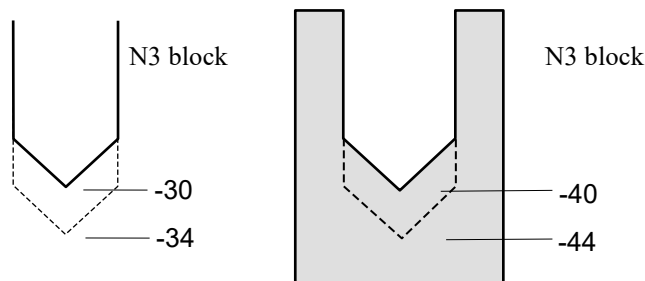


Fig 4-12

Example 3:

```

N0 G91
N1 G00 X120.000 Y80.000 (after comp.)
N2 G43 Z-32.000 H01
N3 G01 Z-21.000 F100.
N4 G04 X2.000
N5 G00 Z21.000
N6 X30.000 Y-50.000
N7 G01 Z-41.000
N8 G00 Z41.000
N9 X50.000 Y30.000
N10 G01 Z-25.000
N11 G04 X2.000
    
```

N12 G00 Z57.000  
N13 G49 X-200.000 Y-60.000  
N14 M02

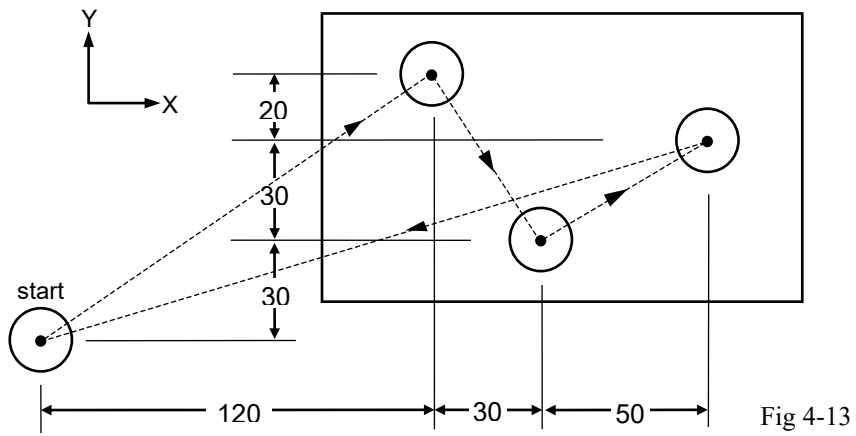


Fig 4-13

### 5 CONTROL PANEL KEYS AND SCREEN DISPLAY

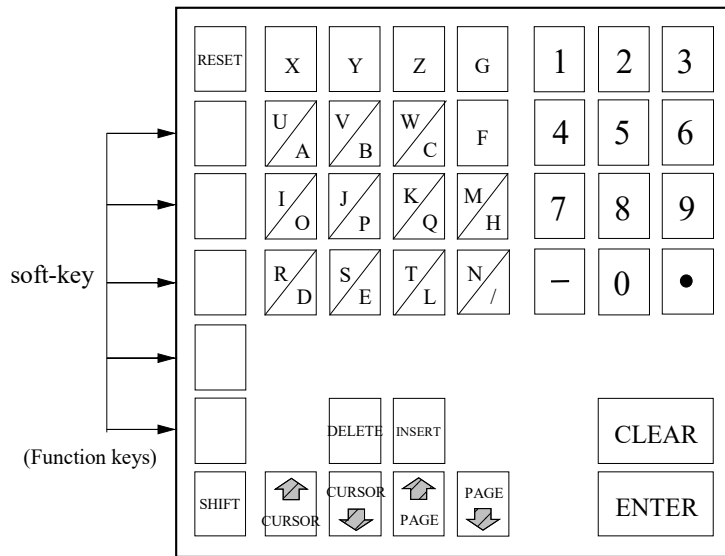


Fig 5-1 HUST 11-series Panel Keys

Fig 5-1 shows HUST 11-series panel keys arrangement. Except those on the left-hand most column, all the keys are for data input, program edit, MCM parameters input, etc. The keys on the left-hand most column are function keys (soft-keys) whose functions are displayed on the CRT screen as shown in Fig 5-2. HUST 11-series has a total of 34 soft-key functions which are spread over nine (9) pages.

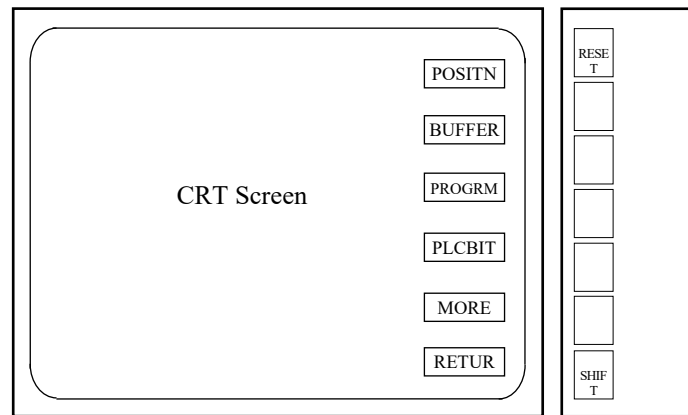


Fig 5-2 Function Keys (Soft-keys)

In order for the function on CRT screen to be effective, press the corresponding soft-key on the right. "More" key is for paging through the soft-keys and "Return" key is to return to the first page. These soft-keys are controlled by O112~O119 of O-bit in HUST standard PLC. Customer can choose what page of soft-key to be open, simply by setting the corresponding O-bit = 1. Otherwise, only the soft-keys for page 1 and 2 will be available.

Page 1 (0112)	Page 2 (0113)	Page 3 (0114)	Page 4 (0115)	Page 5 (0116)	Page 6 (0117)	Page 7 (0118)	Page 8 (0119)	Page 9 (0120)
POSITN	MCM	EDIT	SINGLE	HOME-X	JOG-X	CLR-XR	FIND-N	TEST
BUFFER	GRAPH	AUTO	OPST	HOME-Y	JOG-Y	CLR-YR	AUTO-N	F-HOLD
PROGRM	TAPE	MDI	SKIP	HOME-Z	JOG-Z	CLR-ZR	RE-STA	

Fig 5-3 Function and Execution Mode Keys

**5.1 Panel Key and Function Key Descriptions**

POSITM	The work coordinate of current tool position, in large size.
BUFFER	The work coordinate of current tool position and its relative
PROGRAM	Display the current program selected.
PRNO	Display and select the program stored in the memory.(See Sec.6.1)
MORE	Press once to page through the function keys.
RETURN	Press once to return to the first page.
MCM	Display or set MCM parameters. (See Chap 7)
GRAPH	Display the tool path during program execution. (See Sec. 5.2.7)
TAPE	Program or MCM data input/output by RS232C interface. (Chap
PLCBIT	Display the status of I/O/C/S/A-Bit in the PLC diagram. (See
EDIT	Program edit mode. (See Sec. 6.2)
AUTO	Program auto execution mode. (See Sec. 8.3)
MDI	Manual data input or manual single block execution. (See Sec. 8.2)
TEACH	Program teach mode. (See Sec. 6.4)



- SINGLE Single block execution. (See Sec. 8.4)
- OPST Option stop. (See Sec. 8.5)
- SKIP Single block SKIP function. (See Sec. 8.6)
- DRYRUN Program dryrun at high speed. (See Sec. 8.7)
- HOME-X HOME-Y Tool returns to HOME position on X/Y-axis.(See Sec. 8.1.1)
- HOME-Z HOME-B Tool returns to HOME position on Z/B-axis.(See Sec. 8.1.1)
- JOG-X JOG-Y JOG move for X-axis. (See Sec. 8.1.2)
- JOG-Z JOG-B JOG move for Z-axis. (See Sec. 8.1.2)
- CLR-XR CLR-YR Clear the relative coordinate on X-axis. (See Sec.
- CLR-ZR CLR-XR Clear the relative coordinate on Z-axis. (See Sec. 8.12)
- FIND-N Search for the program block (sequence) number. (See Sec. 6.3)
- AUTD-N Generate automatically the program block number.(See
- RE-STA Program re-start. (See Sec. 8.9)
- LAST-N Search for the block number where the program was interrupted. (Sec.6.3)
- TEST Program test by MPG hand-wheel. (See Sec. 8.8)
- F-HOLD Temporary stop of program execution. (See Sec. 8.11)

Followings are the keys for data input or program editing. To access the letters on the lower-right corner, press the corresponding key twice within 0.75 seconds. Press "Shift" key is NOT going to work.

- RESET Reset and clear all CNC functions to power-on status.  
Without changing screen display.
- DELETE Delete program block.
- INSERT Insert program block.
- CURSOR CURSOR Move the data display up/down one item or one block.  
(i.e. MCM, ,I/O/C/S/A)
- PAGE PAGE

Move the data display up or down one page or 10 items.  
(i.e. MCM, ,I/O/C/S/A)

## 5.2 Screen Display Modes

HUST 11-series provides eight (8) types of screen display, which are controlled by seven (7) function keys on Page 1 and 2. Only one display can exist at a time. The "Tape" key on page 2 and all other keys on page 3~9 will be discussed in Chap 6~9.

### 5.2.1 Power-on Display

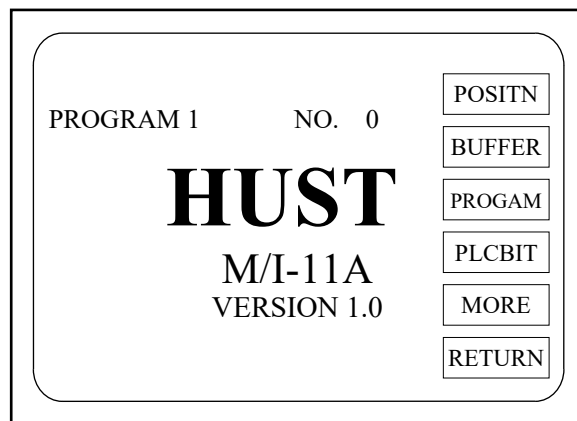


Fig 5-4 Power-on Display

The power-on display includes:

- Name brand of "HUST" and its model number T(M,I)-11.
- Current program number selected -- Program 1
- MCM counter number -- NO. 0
- Function keys -- Page 1

If "Error 22 Em-stop, Home again" is displayed when power-on, press "Reset" to clear it. For power-on default G-codes, please refer to Chap 3.

## 5.2.2 Coordinate Display (POSITN)

Press POSITN once to display the coordinate of current tool position and the following information.

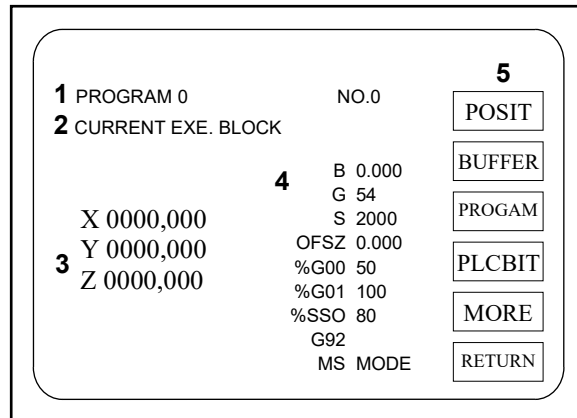


Fig 5-5 Tool Position Display

1. Current program number selected and the number of times the current program having been executed (counter number).
2. Display three blocks of the program being executed.
3. Work coordinate of the current tool position.
4. B 0.000 : Tool coordinate of B-axis.  
G 54 : The current coordinate system being used.  
S 2000 : The current spindle rpm.  
OFSX 0.000 : The current offset compensation for X-axis.  
OFSZ 0.000 : The current offset compensation for Z-axis.  
%G00 50 : G00 feed-rate override (MFO) in %. 0, 25, 50, or 100.  
%G01 100 : G01 (G02) feed-rate override in %. Normal, 0, 10, 20, 30 .....150.  
0~250% for MPG test.  
%SSO 80 : Spindle speed override in %. Normal, 0, 10, 20, 30 .....150.  
0~250% for MPG test.  
G99 : Display feed-rate mode, G98 or G99.  
MS mode : Master/slave mode in effect
5. Function keys

### 5.2.3 Buffer Display

Press BUFFER to display the data in the buffer as in Fig 5-6.

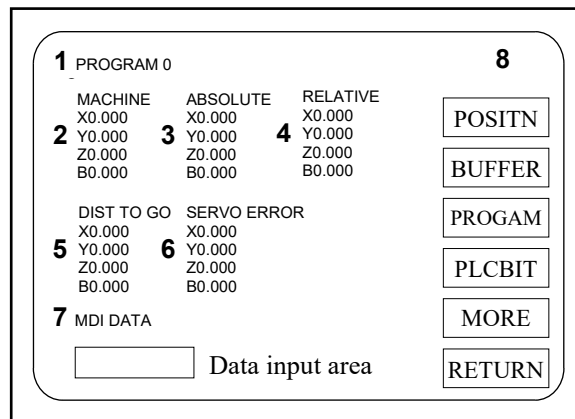


Fig 5-6 Buffer Display

1. Program number and counter number display.
2. Display the machine coordinate of the current tool position.
3. Display the current tool position in absolute coordinate.
4. Display the current tool position in machine unit (i.e. pulses).
5. Display the distance between the current tool position and the end point of the block.
6. Display the difference between the servo command and the encoder feed-back.
7. Display the MDI input data.
8. Function keys

### 5.2.4 Program Display

Press PROGRAM once to display the program content.

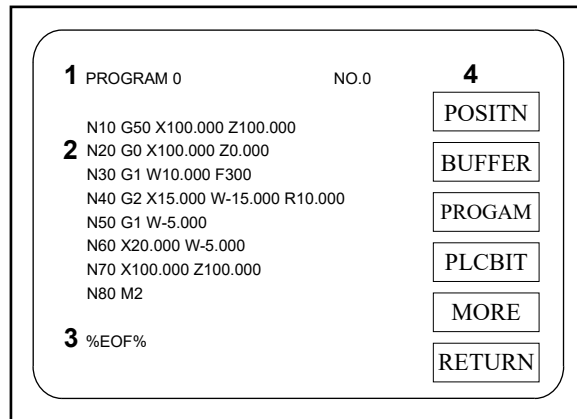


Fig 5-7 Program Display

1. Display current program number and the number in the counter.
2. Display program content. If the content is more than one page, use PAGE↑ or PAGE↓ to see the program content on other pages.
3. Display the end of the program as %EOF%. (End of File)
4. Function keys.

### 5.2.5 Program Directory Display

Press PRNO once to display program directory as in Fig 5-8.

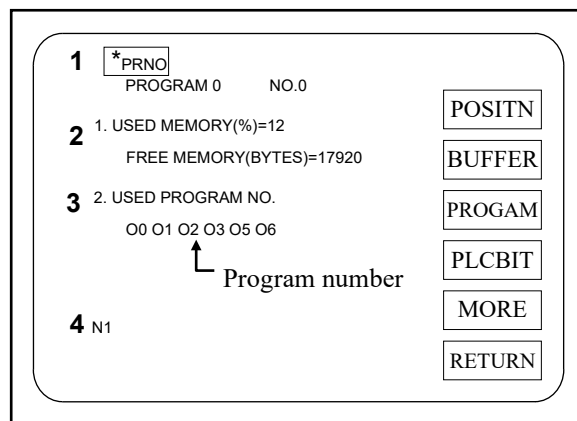


Fig 5-8 Program Directory Display

1. "\*\*PRNO" means the CNC is in program selection mode.

2. Display the current program memory status
  - Used-Memory      The memory that has been used in %.
  - Free-Memory      The memory that is free for more program storage in Bytes.
3. The current numbers that have been assigned.
4. The program number input data.

To select a program, simply key-in "N", followed by a "program number", then key "ENTER" to complete the selection process.

### 5.2.6 MCM Parameter Display

Press MCM on page 2 to display the first page of MCM parameters as in Fig 5-9.

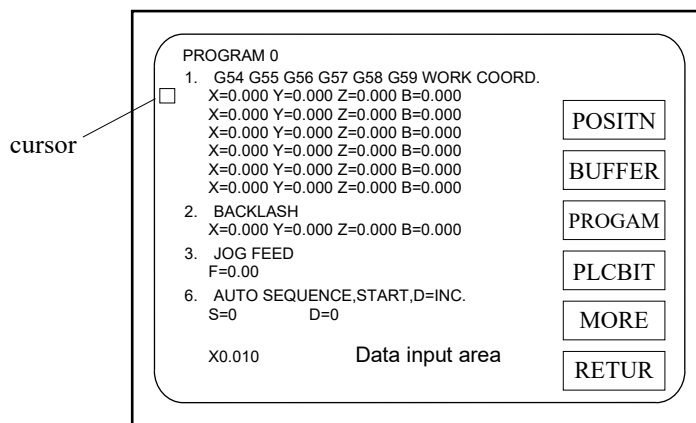


Fig 5-9 MCM Parameter Display

This screen displays the setting values for all MCM parameters. These parameters are divided into two groups -- the protected and the un-protected. Parameters #1~#22 are the un-protected and can be viewed and revised with direct input. Parameters #31 and above are protected and are required a proper procedures (see Chap 7) to un-lock the protection codes in order to viewing and revising data.

Use PAGE↑ or PAGE↓ to page through the parameters and CURSOR↑ or CURSOR↓ to move cursor to the specific item for parameter revision. For method of parameter revision, please see Chap 7.

## 5.2.7 Graphics Display

Press GRAPH on page 2 to display the tool's cutting path during program execution.

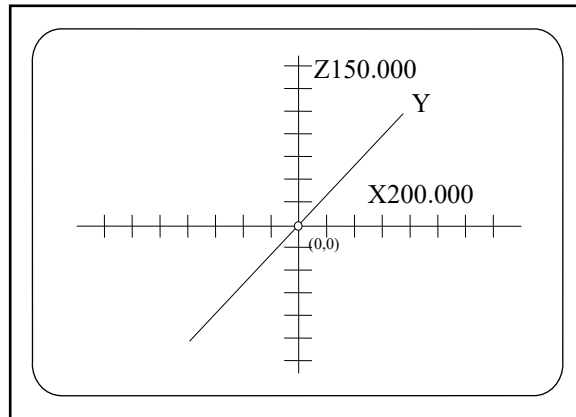


Fig 5-10 Graphic Display - 1

The graph size and the start point are determined by the settings of MCM parameter #71, #72 and #73.

MCM #71 Graphic Horizontal Length  
X=100.000

This parameter sets the length of the horizontal X-axis of the graph on the CRT screen. The vertical length will be equal to 3/4 of the horizontal length. The intersection of X, Y, Z-axis is the origin of the graph.

MCM #72 Graphic Start Point From Zero  
X=60.000, Y=45.000, Z=0.000

This parameter is used to set the location for the graphic start point. The graphic start point is the work origin of the work-piece coordinate system. Therefore, if the settings are X=0.0, Y=0.0, Z=0.0, the work origin and the graph origin are overlapped. If the settings change to X=60.0, Y=45.0, Z=0.0, the work origin (or graphics start point) will be moved to the location of X=60.0, Y=45.0, Z=0.0 of the graph coordinate.

MCM #73 Graphic Plane N=0 XYZ, N=1 XY, N=2 YZ, N=3 ZX.

- N=0, Display graph in 3-dimension X, Y, Z-axis.
- N=1, Display graph in X-Y plane with X-axis as horizontal axis.
- N=2, Display graph in Y-Z plane with Y-axis as horizontal axis.
- N=3, Display graph in Z-X plane with Z-axis as horizontal axis.

Example: MCM #71, X=200.0  
MCM #72, X=60.0, Y=45.0, Z=0.0  
MCM #73, N=1

The graphic start point (or work origin) will be as shown in Fig 5-11.

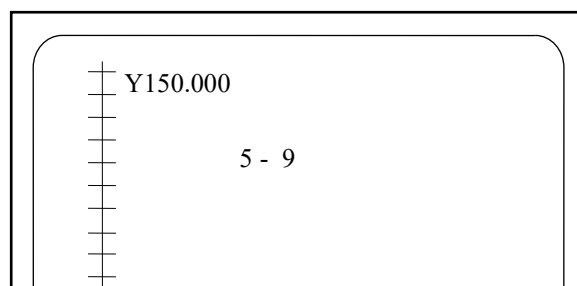


Fig 5-11 Graphic Display - 2

**5.2.8 I/O/C/S/A-bit Status Display (PLCBIT)**

Press PLCBIT once to display the current status of I-bit. Use PAGE↑ or PAGE↓ to view the status of O/C/S/A-bit. For details of I/O/C/S/A-bit, please refer to "HUST Connecting Manual". The meanings of these bits are briefly explained below:

- IBIT : The status of input bit from machine to PLC.
- OBIT: The status of output bit from PLC to machine.
- CBIT: The status of command-bit from PLC to CNC controller.
- SBIT : The status of CNC status-bit from CNC to PLC.
- ABIT: The status of the auxiliary-bit for PLC operation.

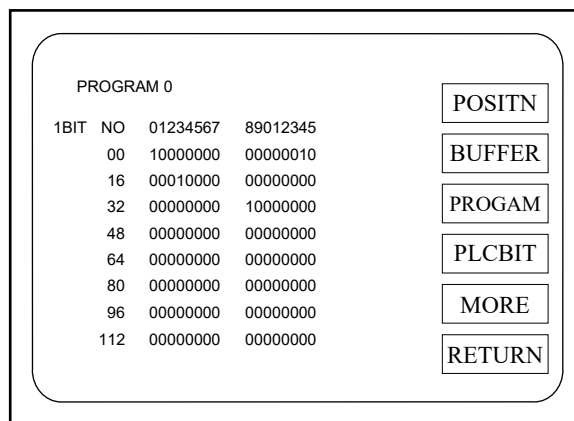
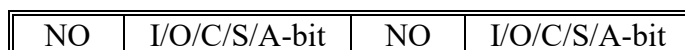


Fig 5-12 I-bit Status Display

HUST 11-series provides 128 bits for each I/O/C/S/A-bit. Fig 5-12 shows 128 I-bit status. They are arranged in 8 rows and each row has 16 bits. The status of each bit is shown by either 0 or 1. The I/O/C/S/A-bit status display is very useful for PLC de-bugging or error checking process.

- 0 = The corresponding bit is NON-Active or OFF.
- 1 = The corresponding bit is ACTIVE or ON.

The numbering sequence is as follows:





00	00~15	64	64~79
16	16~31	80	80~95
32	32~47	96	96~111
48	48~63	112	112~127

### 5.3 Descriptions of Messages in Reversed Cursor Display

During program execution, the current active functions will be displayed either at the top or at the bottom of the screen separated by an "\*" in reversed background color. In the case of CNC error, an error message will also be displayed.(see Chap 10 for error messages). The displayed messages are summarized as follows:

Message	C/O-Bit	Meanings
LOCK	C000	Machine lock or temporary stop
FEHD	C003	Feed hold
PRNO	C005	Program selection by keyboard
SINGLE	C006	Single block program execution
DRYRUN	C015	Program dry-run at high speed without cutting action
OPST	C026	Option stop
SKIP	C027	Option SKIP function
AUTO	C008~C010	Auto execution mode
HOME	C008~C010	Tool moves to HOME location (machine origin)
EDIT	C008~C010	Program edit mode
JOG	C008~C010	JOG mode
TAPE	C008~C010	PC on-line operation
TEACH	C008~C010	Program teach mode
MDI	C008~C010	Manual single block operation
IN PROCESS	S080	Program being executed
XAXIS	C060~C062	X-axis being selected
YAXIS	C060~C062	Y-axis being selected
ZAXIS	C060~C062	Z-axis being selected
BAXIS	C060~C062	B-axis being selected
RE-START	C011	Program re-start
X MIRROR		Mirror cutting along X-axis in effect
Y MIRROR		Mirror cutting along Y-axis in effect
TEST	C056	Program test by MPG hand-wheel
N-STOP	C036	Round corner non-stop operation
S-PLC	C031	PLC ladder simulation
G91		The position coordinate in incremental mode
READER IN	O096~O099	Program is transferred from PC to CNC controller
PUNCH OUT	O096~O099	Program is transferred from CNC to PC
TAPE EXE.	O096~O099	The controller is reading the program from PC & execute
LADDER IN	O096~O099	PLC ladder is transferred from PC to CNC controller



## 6 PROGRAM EDITING

Four topics are to be discussed in this chapter:

1. Selecting a program from memory (PRNO)
2. Editing a new program
3. Revising an old program
4. Editing a program in TEACH mode

### 6.1 Program Selection

HUST 11-series controller has a memory capacity of storing up to 100 programs. You can choose one of them for execution. The procedure is as follows:

1. Press PRNO (page 1 of Function keys) to bring up program directory.

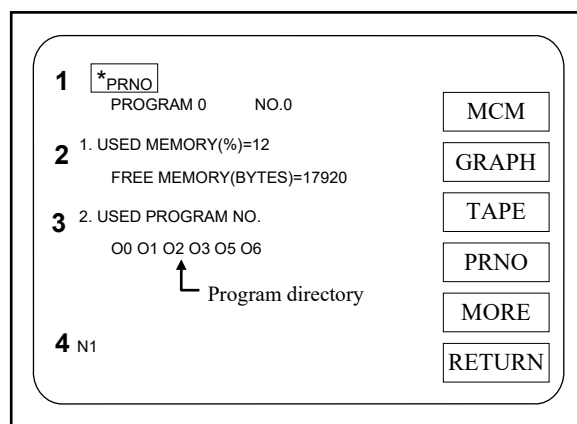


Fig 6-1 Program Selection - 1

2. Press "N" and "program number" (N1 for example). If the number is not in the directory, this becomes a new program number.

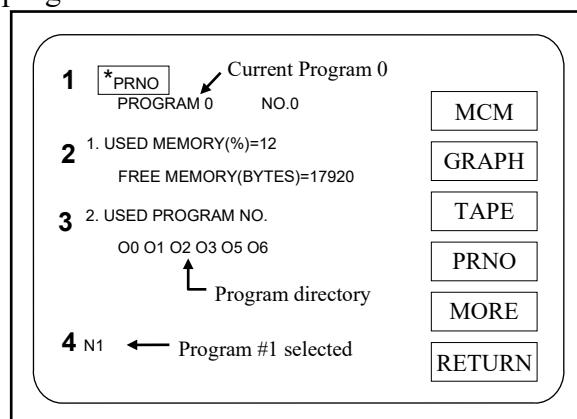


Fig 6-2 Program Selection - 2

3. Press "ENTER" key to complete the selection process and the display becomes:

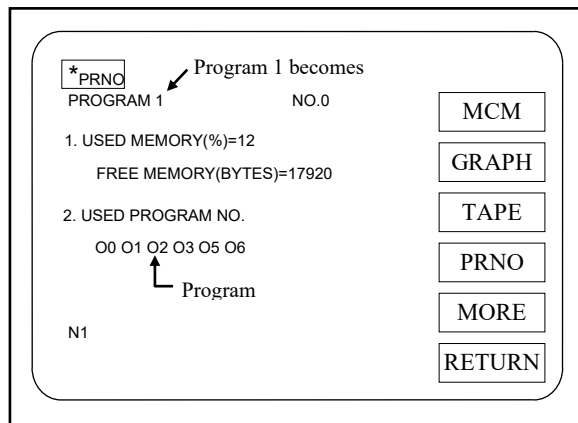


Fig Program Selection - 3

Program No 90~99 are protected. To access these programs, you have to un-lock the protection code as explained in Chap 7.

## 6.2 Program Editing

Once the program number has been selected, press MORE key to turn function-key to page 3. Press EDIT key to bring up the edit mode screen. There will be no program content if this is a new program.

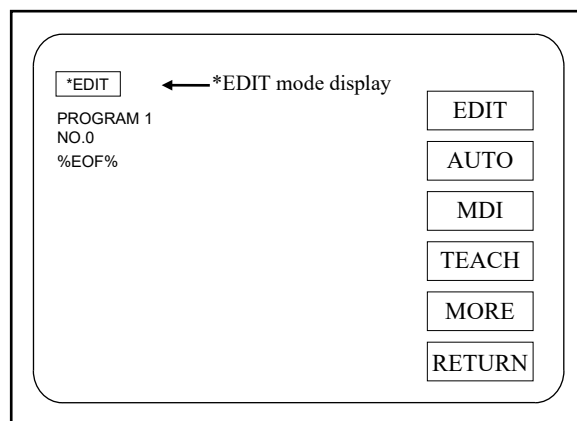


Fig 6-4 Edit A New Program

### Program Block (sequence) Number Generation

When editing a program, the program block number can be omitted. In this case, please set "D" value of MCM #6 equal to 0. Otherwise, the program block number will be generated automatically every time you press the INSERT key. The number generated will be based on the parameter settings of MCM #6. To revise the block number, press N and the number, then ENTER.

### Program Editing Example:

```

Program 1:  N10 G0 X0. Z0.
            N20 G1 X50. Z50.
            N30 U30. W-30.
            N40 G0 X0. Z0.
            N50 M2
    
```

Auto generation of block number with MCM #6 settings, S=0, D=10. Make sure the controller is in EDIT mode.

- The first block -- N10 G0. X0. Z0.

Press [G] - [0], then INSERT key to establish a new block. The screen displays as:

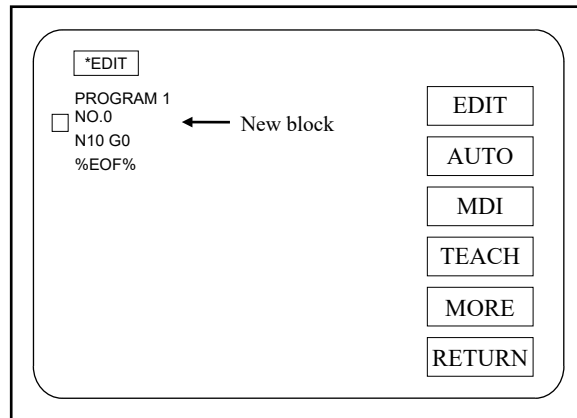


Fig 6-5 Establishing A New Block - Step 1

Press [X] - [0] - [.] , then ENTER. The screen displays as:

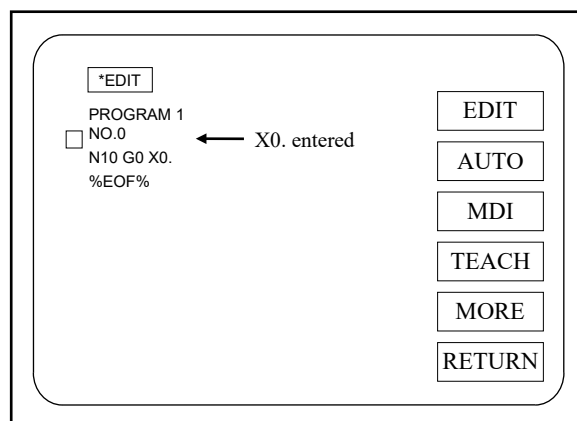


Fig 6-6 Establishing A New Block - Step 2

Press [Z] - [0] - [.] , then ENTER. The screen displays as:

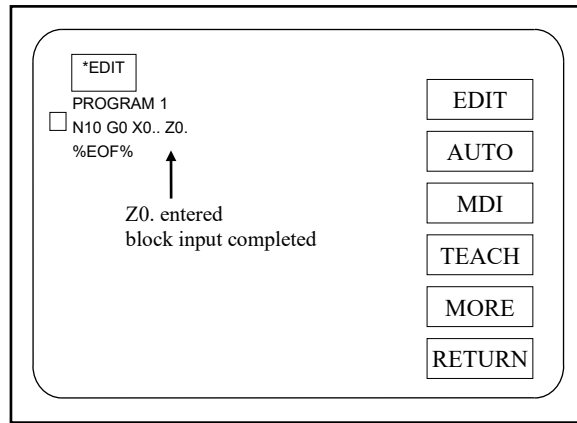


Fig 6-7 Establishing A New Block - Step 3

Use the same method to edit Blocks 2~5.

- The second block -- N20 G1 X50. Z50.

[G] - [1] - INSERT  
 [X] - [5] - [0] - [.] - ENTER  
 [Z] - [5] - [0] - [.] - ENTER

- The third block -- N30 U30. W-30.

[U] - [3] - [0] - [.] - INSERT  
 [W] - [-] - [3] - [0] - [.] - ENTER

Note: You can input [-] sign any time before pressing ENTER.

- The fourth block -- N40 G0 X0. Z0.

[G] - [0] - INSERT  
 [X] - [0] - [.] - ENTER  
 [Z] - [0] - [.] - ENTER

- The fifth block -- N50 M2

[M] - [2] - INSERT

### 6.3 Revision of An Old Program

To revise an old program in the memory, select the program and put the CNC controller in EDIT mode as explained before. The program will be displayed on the screen. Let's use Program 1 in the last section for explanations.

- Add and revise a command code to the third block

Before revision : N30 U30. W-30.  
 After revision : N30 U35. W-30. F300.

1. Use CURSOR↑, CURSOR↓ key to bring cursor to N30 block.
2. Then input as follows:  
 [U] - [3] - [5] - [.] - ENTER  
 [F] - [3] - [0] - [0] - [.] - ENTER

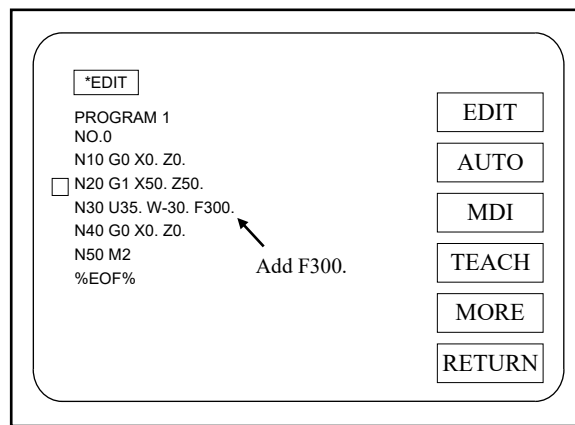


Fig 6-8 Program Revision

- Delete a command code in the third block

Before revision : N30 U30. W-30. F300.

After revision : N30 U30. W-30.

1. Use CURSOR $\uparrow$ , CURSOR $\downarrow$  key to bring cursor to N30 block.
2. Input the corresponding capital letter without a numerical number and press ENTER.  
[F] - ENTER

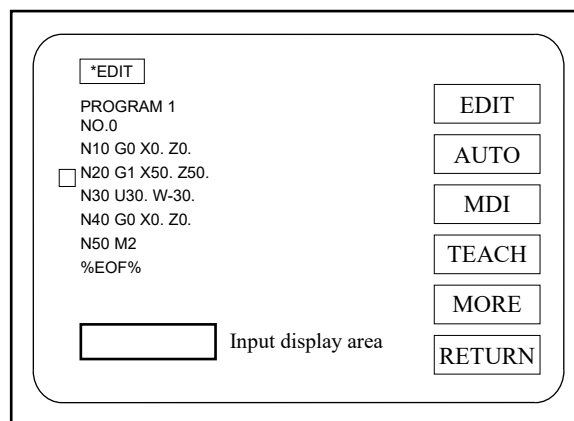


Fig 6-9 Delete a Command Code

Note: Press CLEAR key to clear data in the input display area.

- Insert a block of program

Let's insert a block N35 U20. W-20. between N30 and N40.

1. Use CURSOR $\uparrow$ , CURSOR $\downarrow$  key to bring cursor to N30 block.
2. Input the followings:

[U] - [2] - [0] - [.] - INSERT

[W] - [-] - [2] - [0] - [.] - ENTER

Block number will be generated automatically, but not 35 as expected. You can press N35 and ENTER to revise it if desired.

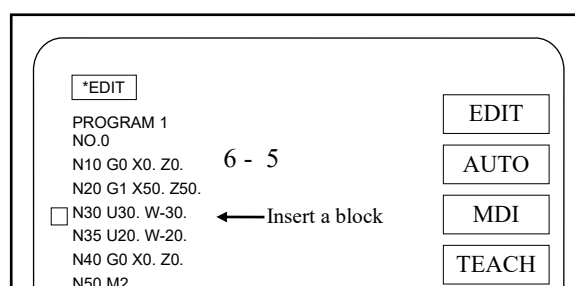


Fig 6-10 Insert A Block of Program

- Delete a block of program

Let's delete block N35 U20. W-20.

1. Use CURSOR↑, CURSOR↓ key to bring cursor to N35 block.
2. Then press DELETE key.

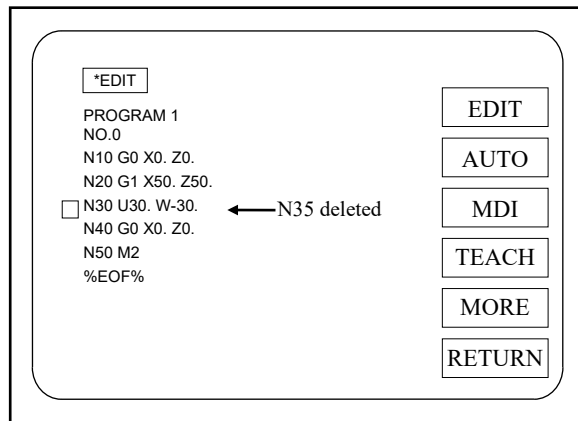


Fig 6-11 Delete A Block of Program

- Delete a program

1. Use PRNO to select the program to be deleted.
2. Press MDI key. (must be done under MDI mode)
3. Input G10 P2000 command as follows:  
 [G] - [1] - [0] - ENTER  
 [P] - [2] - [0] - [0] - [0] - ENTER
4. Press [CYCST] key. (Note: CYCST is an external key)

Once you have pressed [CYCST] key, the whole program will be deleted and can not be recovered. Be careful when using this command.

- Delete all programs (0~99) in the memory

1. Press MDI key. (must be done under MDI mode)
2. Input G10 P2001 command as follows:

[G] - [1] - [0] - ENTER  
 [P] - [2] - [0] - [0] - [1] - ENTER



3. Press [CYCST] key.

Once you have pressed [CYCST] key, the entire programs in the memory will be deleted and can not be recovered. Be careful when using this command.

- Search for specific block number (FIND-N)
  1. Key in the desired block number, such as N150.
  2. Press FIND-N on page 8 of function key. The cursor will move to the first N150 encountered if there are more than one N150. If the desired block number doesn't exist, a message "Error 27 not found" will be displayed.

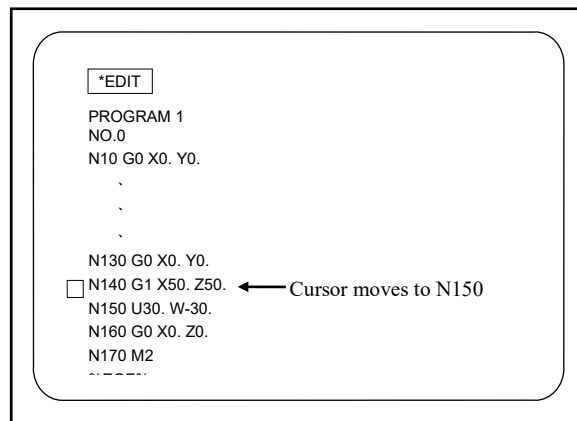


Fig 6-12 Search For Specific Block Number

- Re-arrange the block number (AUTO-N)

When you have finished editing a program and decided to re-arrange the block number based on the parameter settings of MCM #6, you can follow the steps below:

1. Make sure the CNC is in EDIT mode.
2. Press AUTO-N on page 8 of the function keys.

- Search for the block number where the program was interrupted (LAST-N)

If the program execution was interrupted due to any reason and when the interruption was resolved, you can press this function key to bring the execution back to the block where it was interrupted.

1. Make sure the CNC is in EDIT mode and the C-bit ,C057 = 1.
2. Press LAST-N on page 8 of the function keys.

## 6.4 Editing A Program in Teach Mode

Occasionally in program editing, particularly for a 3-D work-piece, it becomes very difficult to obtain a correct coordinate. To overcome this problem, HUST 11-series provides a TEACH function. All you have to do is to use a MPG hand-wheel to move the tool to the desired location and press ENTER. The desired coordinate will be transferred to the program.

The TEACH mode is identical to EDIT mode. The only difference is that the TEACH mode uses a MPG hand-wheel to find the coordinate by moving the tool to the desired location. Therefore, TEACH mode shares the same function keys that are used for EDIT mode, such as INSERT, DELETE, ENTER, etc.

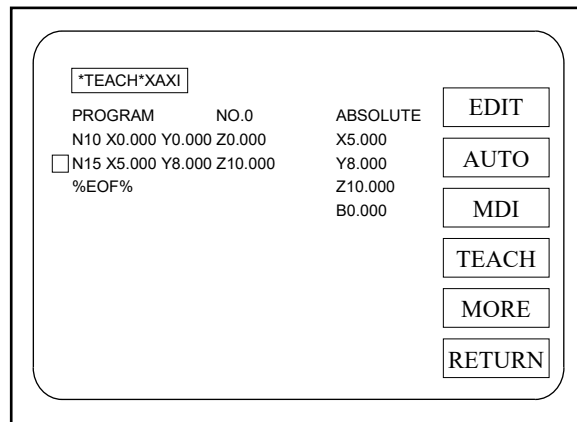


Fig 6-13 Program Edit in Teach Mode

Fig 6-13 is a screen display for TEACH mode. Please note the three things on the screen:

- XAXIS on the first line in reversed background. -- X-axis is active
- The coordinate under "ABSOLUTE" -- Current tool position
- Cursor location -- Working on the coordinate of the cursor block

When you rotate the MPG hand-wheel, the X-axis coordinate under ABSOLUTE will change accordingly. When you press ENTER key, the coordinate under ABSOLUTE will be transferred to the block indicated by the cursor. The procedure for using TEACH function is as follows:

1. Use PRNO to select the program.
2. Press TEACH (page 3) to activate teach function.
3. If editing a new program, key in G/M-code, then press INSERT to establish a new block. If editing an old program, use CURSOR key to move the cursor to the desired block.
4. Use external input key to select X- or Z-axis. (see C060~C062 bit of Connecting Manual)
5. Rotate MPG hand-wheel to move the tool to the desired location (watch the coordinate change under ABSOLUTE). Repeat step's 4~5 for other axis. Press ENTER to transfer the coordinates under ABSOLUTE to the block indicated by the cursor.
6. Repeat steps 3~5 until finishing the entire editing process.
7. Complete the program with M02, M30, or M99.

Note that every time you press ENTER key, the coordinate under ABSOLUTE will be transferred to the cursor block. If you like to key-in the coordinate, please do it in EDIT mode.

## 6.5 Rules For Decimal Input

The numerical input that is associated with the function (or position) code is either integer or decimal. In HUST 11-series system, the numerical input is divided into three categories.

- G, M, N, S, T-code ..... Integer
- X, Y, Z, B, U, V, W, I, J, K-code ..... Three (3) decimal places
- F-code ..... Two (2) decimal places

If you input the number according to the format mentioned above, it will not present any problem. The only time that might cause a problem is when you enter an integer that requires a decimal input. The reason is that HUST controller will internally count backwards 3 decimal places (2 for F-code) and add a decimal point at that position. Followings are some example inputs and the resulting numbers after internal process.

Input	Resulting Number
X2	X0.002 mm
Z250	Z0.250 mm
U2500	U2.500 mm
W25.0	W25.000 mm
F25	F0.25 mm/min
F25.	F25.00 mm/min

To avoid any possible problem, please use integer for G, M, N, S, and T-code and use decimal input for X, Y, Z, B, U, V, W, I, J, K, and F-code.

## 6.6 Notes on Program Editing

### Program block number

1. Block number can be omitted, but you can not use SEARCH-N function to search for the program block without block number.
2. The ranking of the block number has nothing to do with the program execution sequence. The program is executed in a top-down fashion.

### Program Block

1. Do not use two G-codes in the same block. If more than one G-code exists in a block, only the last one is effective.
2. If you specify absolute coordinate and incremental coordinate for the same axis in a block, only the incremental coordinate will be executed.  
Example: G1 X100. U50. will execute U50.
3. Do not exceed 48 bytes of data input for a single block. Otherwise, the CNC controller will show an error message as:  
"ERROR 8 EXCEED 48 CHARACTERS FOR ONE BLOCK"



## 7 MCM (Machine Constant) PARAMETERS

### 7.1 MCM Parameter Setting, G10

The MCM parameter allows the user to define certain machine constants that match to the mechanical specifications of the equipment and the machining requirements. The correct and proper setting of these constants enable the user to smoothly operate the equipment. Once they have been set, press RESET key to restart the system.

Some of the constants requires only initial setting during installation. Once they have been properly set, they stay there forever unless the machine or the CNC system is altered or disturbed. Except for parameters #1~#22, all other parameters are electronically protected to prevent from being accidentally changed or erased. To view or revise these parameters, You have to follow the unlocking procedure described below. Program No 90~99 have also been protected in the same way.

To read or view MCM parameters:

1. Press MCM key once to display the first page of MCM parameters.
2. Use PAGE↑ or PAGE↓ key to scroll through pages.

The unlocking procedure:

Format: G10 L9999

1. Press MDI key.
2. Key in G10 and press ENTER.
3. Key in L9999 and press ENTER.
4. Press CYCST key to complete the unlocking procedure.

The locking procedure:

Format: G10 L1

1. Press MDI key.
2. Key in G10 and press ENTER.
3. Key in L1 and press ENTER.
4. Press CYCST key to complete the locking procedure.

When the CNC controller is powered on, the MCM parameters are in protected mode.

To clear all MCM settings by G10 to default values:

Format: G10 P1000

1. Press MDI key.
2. Key in G10 and press ENTER.
3. Key in P1000 and press ENTER.
4. Press CYCST key to complete the procedure.

To clear the value in a MCM Counter by G10: (MCM parameter #74)

Format: G10 P201

1. Press MDI key.
2. Key in G10 and press ENTER.
3. Key in P201 and press ENTER.
4. Press CYCST key to complete the procedure.

To set a limit for a MCM Counter by G10: (Counter Limit, MCM parameter #122)

Format: G10 P200 L\_\_\_\_ (L\_\_\_\_=Counter limit)

1. Press MDI key.
2. Key in G10 and press ENTER.
3. Key in P200 and press ENTER.
4. Key in L\_\_\_\_ and press ENTER.
5. Press CYCST key to complete the procedure.

To set graphic parameters by G10: (MCM parameter #71~#73)

Format: G10 P300 X\_\_\_\_ I\_\_\_\_ J\_\_\_\_ K\_\_\_\_ Q\_\_\_\_

X : Horizontal length

I, J, K : X, Y, Z-coordinate of graphic start point with respect to the graphic origin

Q : Graphic plane setting (0, 1, 2, or 3)

1. Press MDI key.
2. Key in G10 and press ENTER
3. Key in P300 and press ENTER.
4. Key in Xxxx and press ENTER
5. Key in Ixxx and press ENTER. (xxx = X-coordinate)
6. Key in Jyyy and press ENTER. (yyy = Y-coordinate)
7. Key in Kzzz and press ENTER. (zzz = Z-coordinate)
8. Key in Qx and press ENTER. (x = 0 ~ 3)
9. Press CYCST key to complete the procedure.

To clear graphic display by G10:

Format: G10 P301

1. Press MDI key.
2. Key in G10 and press ENTER.
3. Key in P301 and press ENTER.
4. Press CYCST key to complete the procedure.

To revise or modify MCM parameters:

1. Press MCM key once to bring MCM parameters onto the screen.
2. Use CURSOR key to move cursor to the desired parameter.
3. Type in the corresponding letter and the correct number.
4. Press ENTER key to complete the modification.

Notes on decimal input for MCM parameters:

(The decimal input here has nothing to do with those in Sec. 6-5)

When you enter a MCM parameter with a decimal number, the CNC controller will process internally by adding zero (0), if necessary, to have three digits following the decimal point. For example:

X1.2 → X1.200 → X1200

Based on the decimal format for each individual parameter, the CNC controller will then place a decimal point at the proper position. The number so obtained will be the final number that is recognized by the CNC controller. The table below summarizes the recognized numbers with different input.

Input Number	MCM Parameter Format		
	Integer, ie MCM #65	2-decimal places, ie MCM #31	3-decimal places, ie MCM #2
Integer 1234	1234	12.34	1.234
2-decimal 12.34	12340	123.40	12.340
3-decimal 1.234	1234	12.34	1.234

When entering a MCM parameter, you have to key in a capital letter followed by a number. This letter can be any letter (A~Z).

For example: MCM #70 RS232C BAUD RATE,  
the input N4800, Y4800, or X4.8 will give the same result.

## 7.2 Machine Constant Description

Unless where is specified, the default settings for the parameters are zero (0).

1. G54, G55, G56, G57, G58, G59 Work Coordinate, in mm

Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€ B=□€.€€€

This MCM is to specify the location of the work origin with respect to the machine origin. Total 6 sets are available. (See Chap 3)

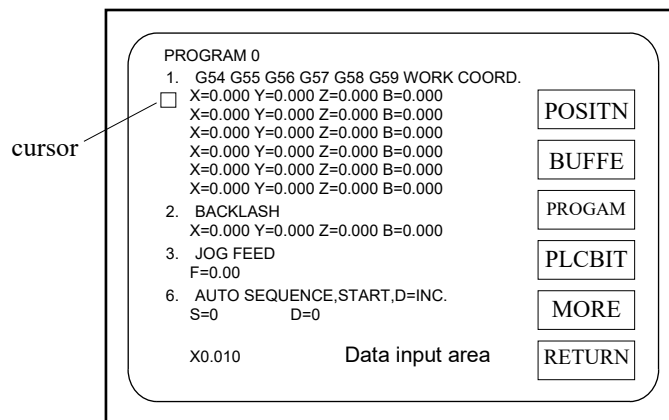


Fig 7-1

2. Backlash, in mm

Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€ B=□€.€€€

Setting range = 0~255 pulses or

Setting range = 0~0.255 mm if the resolution =0.001 mm/pulse.

Setting range = 0~0.510 mm if the resolution =0.002 mm/pulse.

(See MCM #31 for resolution)

Set the backlash compensation of the lead-screw for X, Y, Z and B-axis.

3. JOG Feed, in mm/min

Format: F=□€€€

Set the feed-rate when in JOG mode.

6. Auto Sequence, S=Start, D=Increment

Format: S=□€ The starting number for the first block

D=□€ The numerical increment between blocks

When pressing the INSERT key during editing or revising a program or when pressing function key AUTO-N, these MCM parameters allow you to generate automatically the program block numbers. When D=0, auto sequence function is OFF.

20. Tool Radius Compensation Data (Total 16 groups)

Format: Radius =□€€€€ Unit in mm.

Column 1, 3 (NO) : Tool number

Column 2, 4 (Radius) : The corresponding radius compensation data

PROGRAM 0		NO.0	
20. TOOL RADIUS .DATA			
NO	RADIUS	NO	RADIUS
01	0.000	09	0.000
02	0.000	10	0.000
03	0.000	11	0.000
04	0.000	12	0.000
05	0.000	13	0.000
06	0.000	14	0.000
07	0.000	15	0.000
08	0.000	16	0.000

MCM

GRAPH

TAPE

PRNO

MORE

RETURN

Fig 7-2 Tool Radius Compensation

21. Tool Wear Compensation Data (Total 16 groups)

Format: Radius Wear =□€€€€ Unit in mm.

Column 1, 3 (NO) : Tool number

Column 2, 4 (Radius Wear) : The corresponding radius wear compensation data



PROGRAM 0			NO.0		
21. TOOL RADIUS WEARING .DATA					
NO	RADIUS	WEAR	NO	RADIUS	WEAR
01	0.000		09	0.000	
02	0.000		10	0.000	
03	0.000		11	0.000	
04	0.000		12	0.000	
05	0.000		13	0.000	
06	0.000		14	0.000	
07	0.000		15	0.000	
08	0.000		16	0.000	

POSITN

BUFFER

PROGAM

PLCBIT

MORE

RETURN

Fig 7-3 Tool Wear Compensation

## 22. Tool Offset Compensation Data (Total 16 groups)

Format: Length =□€.€€€ Unit in mm.

Column 1, 3 (NO) : Tool number

Column 2, 4 (Radius Wear) : The corresponding tool offset compensation data

PROGRAM 0			NO.0		
22. TOOL LENGTH .DATA					
NO	LENGTH		NO	LENGTH	
01	0.000		09	0.000	
02	0.000		10	0.000	
03	0.000		11	0.000	
04	0.000		12	0.000	
05	0.000		13	0.000	
06	0.000		14	0.000	
07	0.000		15	0.000	
08	0.000		16	0.000	

POSIT

BUFFER

PROGAM

PLCBIT

MORE

RETURN

Fig 7-4 Tool Offset Compensation

MCM Parameter No 1~22 can be directly accessed by pressing MCM key. The following items, however, require to unlock the "protection code" before viewing or revising.

## 31. Machine Unit in Micro-meter, in 痠/pulse (=0.001 mm/pulse)

Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€ B=□€.€€€

This MCM is for setting the resolution of the machine system, which is based on the width of the lead-screw's pitch and the encoder feed-back resolution (see MCM #64).

The equation for finding the setting value is as follow:

$$\text{Setting value} = P \div N$$

P : The pitch of the lead-screw, in mm/revolution

N : The encoder feed-back resolution, in pulses/revolution

Example: P = 5 mm/rev for X-axis, and N = 4000 pulses/rev

$$\begin{aligned} \text{The setting value for X-axis} &= 5 \div 4000 \\ &= 0.00125 \text{ mm/pulse} = 1.25 \text{ 痠/pulse} \end{aligned}$$

**IMPORTANT -- This parameter is determined by the drive mechanism and the servo-motor encoder. It should not be changed once it has been determined, unless the machine system is changed. Otherwise, serious error may result.**

32. Home Direction, N=0 +, Else -  
 Format: X=□€ Y=□€ Z=□€B=□€

Setting the direction of the Home location (Machine zero) for each axis.  
 Setting = 0, Tool moves to the Home location in positive (+) direction.  
 Setting = 1, Tool moves to the Home location in negative (-) direction.

33. G83 CLEARANCE, in mm (For M-11 controller only)  
 Format: Z=□€€€ (Default = 0.100)

When applying G83 deep hole canned cycle, set the distance (or coordinate) where the feed-rate for the drill bit changes from G00 to G01. (See G83 in Chap 3)

34. Traverse Speed Limit (G00), in mm/min  
 Format: X=□€€€ Y=□€€€ Z=□€€€ B=□€€€  
 (Default X=5000.00, Y=5000.00, Z=5000.00, B=5000.00)

Setting the max feed-rate limit for rapid-move command G00 in each axis. This feed-rate limit can be calculated from the following equation:

$$F_{max} = 0.95 * \text{RPM} * \text{Pitch} \div \text{GR}$$

RPM : The max rpm of servo motor  
 Pitch : The pitch of the lead-screw  
 GR : Gear ratio of Lead-screw/motor

Example: Max rpm = 2000 rpm, Pitch = 5 mm/rev, Gear Ratio = 2/1  
 $F_{max} = 0.95 * 2000 * 5 / 2 = 4750 \text{ mm/min}$

35. Home Speed Limit, in mm/min  
 Format: X=□€€€€€ Z=□€€€€€  
 (Default X=2500.00, Y=2500.00, Z=2500.00, B=2500.00)

Set the moving speed in each axis when the tool is moving to the HOME limit switch.

36. Spindle Speed At 10 volts, in revolutions/min  
 Format: S=□€€€€ (Default S=2000)

Set the max spindle RPM when the CNC controller sends out 10 Volts of command.  
 S=3600 means that the spindle rpm is 3600 when the controller sends out 10 volts.

37. The 4th Axis Condition, N=0 Axis, N=1 Spindle  
 Format: N=□€ (Default N=1)

Set the 4th axis as spindle axis or linear axis (B-axis).

N=0 Set the 4th axis as a linear axis (B-axis).

N=1 Set the 4th axis as a spindle axis.

### 38. Home Grid Speed, in mm/min

Format: F=□□□□□.□□

Set the moving speed when the tool, after having touched the HOME limit switch, is searching for the encoder grid signal during HOME execution. HUST 11-series CNC has three (3) different speeds when you execute HOME function.

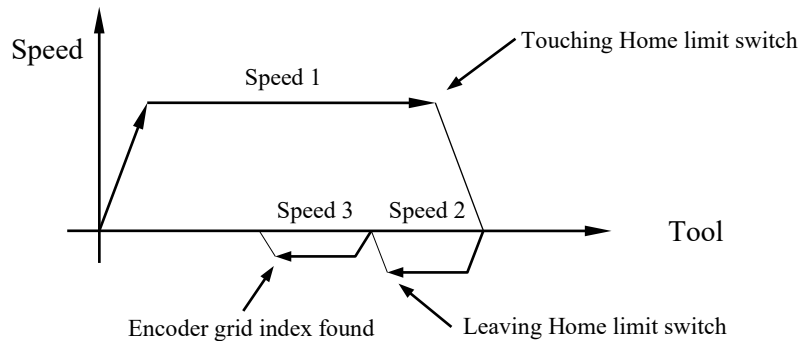


Fig 7-5 Tool's moving speed when searching for HOME location

Speed 1: Determined by the settings of MCM #35.

Speed 2: Equal to 1/10 of Speed 1.

Speed 3: Determined by the settings of MCM #38.

Note that the length of the Home limit switch should be longer than the distance (AB in Fig 7-5) for the deceleration of Speed 1. Otherwise, serious error may result. The equation to calculate the length of the Home limit switch is

$$\text{Length of Home Limit Switch} \geq \frac{\text{FDCOM} * \text{ACC}}{60000}$$

FDCOM = Speed 1, in mm/min. (MCM #35)

ACC = Time for acceleration/deceleration, in ms. (MCM #44)

Example: FDCOM = 3000.00 mm/min, and ACC = 100 ms

$$\text{Length of Home Limit Switch} = 3000 * 100 / 60000 = 5 \text{ mm}$$

### 39. Position Gain, Normal N = 64

Format: X=□□□□ Range = 32 ~ 256

Set the position gain for encoder feed-back. Recommended setting value is 64. The proper setting of this MCM is very important in assuring the smooth operation of the servo motor. Do not change once the correct setting is determined.

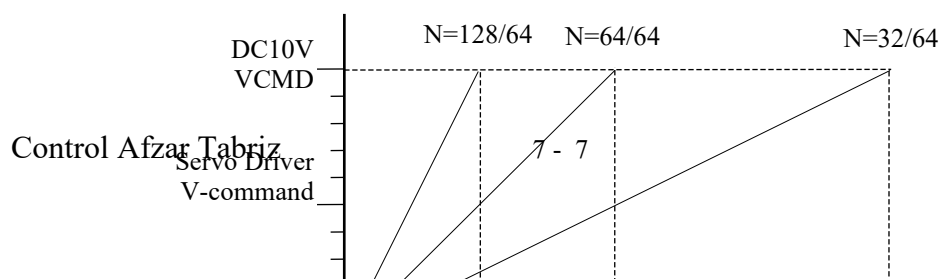


Fig 7-6 The relationship of Driver V-command and Servo Error

Recommended servo motor adjusting procedure:

1. Adjust the servo driver based on its operation manual.
2. Adjust the encoder multiplication factor (MCM #43). Under the normal condition, the servo error will be blinking between 0~1 if the motor is properly locked. If it's blinking between 4~5, adjust MCM #43 by lowering its multiplication factor.
3. Adjust the position gain of encoder feed-back. (MCM #39)

$$\text{CNC V-command} = (\text{Gain}/64) * \text{Servo Error} * (10 \text{ V}/2048)$$

HUST controller adopts a closed-circuit system. The servo-error is the difference in pulses between the CNC V-command and the motor encoder feed-back. Then, the next V-command from CNC will be properly adjusted based on this error.

If Servo-error > 4096, a message of "Error 2" will be displayed.

In this case, please increase the setting value for MCM #39 and press RESET. If the problem still exists, please check the motor and the driver for proper connection.

42. Motor Direction, N=0 CW, N=1 CCW.

Format: X=□€ Z=□€

Set the direction of motor rotation.

N=0, Motor rotates in the positive direction. (CW)

N=1, Motor rotates in the negative direction. (CCW)

This MCM can be used to reverse the direction of motor rotation if desired. The setting of this MCM may affect the direction of machine origin.

**IMPORTANT: Motor Divergence**

Due to the variations in circuit design of the servo drivers that are available from the market, the proper electrical connections from servo encoder to the driver, then to the CNC controller may vary. If the connections do not match properly, the motor RPM may become divergent (Rotate @ HIGH RPM) and damage to the machine may result. For this reason, HUST strongly suggest separate the servo motor and the machine before you are 100% sure the direction of the motor rotation. If a motor divergence occurs, please inter-change the connections of (A and B phase) and (A- and B- phase) on the driver side.

## 43. Encoder Multiplication Factor, Normal 65536 (Factory Default Setting)

Format: X=□€ Y=□€ Z=□€ B=□€

Range: 1 ~ 65536

Set the multiplication factor for the servo motor's encoder resolution. The resulting feed-back signals (pulses/rev) will be used in machine resolution calculation of MCM #31. The setting of 65536 is the max value, which represents a multiplication factor of 4. The setting value for a factor less than 4 can be obtained by the following equation.

$$\text{Setting value} = 65536 * \text{Factor} / 4$$

Example: If factor = 2, setting value = 32768.

If the encoder resolution is 2000 pulses/revolution,  
then the feed-back signals = 2000 \* 2 = 4000 pulses/revolution.

## 44. Acceleration/Deceleration Time, in milli-second

Format: T=□€€€ G00 mode (Default 300 ms), Range 10 ~ 1000 ms.

T=□€€€ G01 mode (Default 30 ms), Range 10 ~ 500 ms.

Set the time constant T for motor acceleration and deceleration. If the setting of MCM #123=0, the motor accel./decel. takes one of the forms as follow:

For G00 mode --- LinearFor G01, G02, G03 mode --- Exponential

The motor speed during exponential accel./decel. is calculated below:

$$\text{Motor Speed} = (1 - e^{-t/T})$$

where t = Time variable, T = Time constant setting

If T = 300 ms, the motor speed will approach to 95% of the program feed-rate when t = 900 ms. When the speed is up to 99% of the program feed-rate, HUST CNC will treat it as 100%.

45. Spindle Encoder Pulse Per Revolution  
Format: N=□€ (Default 4096)

Set the spindle encoder feedback in pulses per revolution. This setting is effective only if the setting of MCM #37 N=1.

60. G28 Reference Point, in mm  
Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€ B=□€.€€€

Set the coordinate w.r.t the machine origin for the first reference point.

61. G30 Reference Point, in mm  
Format: X=□€.€€€ Z=□€.€€€

Set the coordinate w.r.t the machine origin for the second reference point. Total 10 sets of data are available for setting.

70. RS232C Baud Rate  
Format: N=□€€€€ (Default N=4800)

Set RS232C communication speed. Choose from 9600, 4800, 2400, and 1200.

Please use the following settings for your PC:

Parity -- Even  
Stop Bits -- 2 bits

71. Graphic Horizontal Length, in mm  
Format: X=□€.€€€ (Default=300.000)

This parameter sets the length of the horizontal X-axis of the graph on the CRT screen. The vertical length will be equal to 3/4 of the horizontal length.

72. Graphic Start Point From Zero, in mm  
Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€ B=□€.€€€

This parameter is used to set the location for the graphic start point. The graphic start point is the work origin of the work-piece coordinate system. See Sec. 5.2.7 or Chap 5 for details.

73. Graphic Plane N=0 XYZ, N=1 XY, N=2 YZ, N=3 ZX.  
Format: N=□€ (Default N=0)

N=0, Display graph in 3-dimension X, Y, Z-axis.  
N=1, Display graph in X-Y plane with X-axis as horizontal axis.  
N=2, Display graph in Y-Z plane with Y-axis as horizontal axis.  
N=3, Display graph in Z-X plane with Z-axis as horizontal axis.

## 74. Current Counter

Format: N=□€

This parameter is effective only if C030=1 in the PLC. (See Connecting Manual)

Record the number of times that the current program has been executed. Every time the program execution encounters M02 and M30, it will add one (1) to the counter. For HUST I-11 controller, when the program execution encounters M99, it will add one (1) to the counter also. When the number is equal to the number as set in MCM #122 (Counter limit), the program execution will stop.

## 80. Software OT Limit +, in mm

Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€ B=□€.€€€  
(Default X=10.000, Y=10.000, Z=10.000, B=10.000)

Set the software over-travel limit in the positive (+) direction, with X, Y, Z and B coordinate being with respect to the machine origin.

## 81. Software OT Limit -, in mm

Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€ B=□€.€€€  
(Default X=-300.000, Y=-300.000, Z=-300.000, B=-300.000)

Set the software over-travel limit in the negative (-) direction, with X, Y, Z and B coordinate being with respect to the machine origin.

## 82. Home Shift Data, in mm

Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€ B=□€.€€€

Set the amount of coordinate shift for Home location (or machine origin). With these settings, the machine coordinate will be shifted by the same amount when you execute "Home". If home shift data are zero for all axes, the machine coordinate after executing "Home" will be zero also. Note that the work coordinate will be shifted by the same amount.

## 93. N=0, Offset Clear On RESET

Format: N=□€

N=0, The tool compensations will be cleared when RESET key is pressed.

N=1, The tool compensations will NOT be cleared when RESET key is pressed.

**MCM parameters 100~112 are for Lead-screw's pitch error compensation**

## 100. Pitch Error Compensation Mode Setting

Format: X=□€ Y=□€ Z=□€(see table below)

X-axis	Y-axis	Z-axis	Explanation
0	0	0	Compensation cancel
-1	-1	-1	Do compensation when the tool is on the (-) side of the reference point
1	1	1	Do compensation when the tool is on the (+) side of the reference point

The reference point for pitch error compensation is the machine origin.

Example : X=1, Y=0, Z=0

No compensation will be done in Y and Z-axis direction.

Do compensation on X-axis when the tool is on the (+) side of the machine origin, but no compensation if on the (-) side.

101. Segment Length for Pitch Error Compensation, in mm

Format: X=□□□.□□□ Y=□□□.□□□ Z=□□□.□□□

The pitch error compensation is done by dividing the total length of the lead-screw into segments and the guideline is as follows:

	Range for Segment length	Max Number of Segment
X-axis	20 ~ 480 mm	40
Y-axis	20 ~ 480 mm	40
Z-axis	20 ~ 480 mm	40

1. Segment length is the lead-screw length divided by the number of segment.  
For example, divide a lead-screw with 1 meter in length into 10 segments, the segment length is  $1000.00/10 = 100.00$  mm.
2. If the average segment length is less than 20 mm, use 20 mm. The amount of compensation for each segment is specified in MCM #110, #111 and #112.
3. The max number of segment over entire lead-screw is 40.  
0=1st segment, 1=2nd segment, 2=3rd segment, 3=4th segment, .....
4. When doing compensation, HUST controller will further divide each segment into 8 sections. The amount of compensation for each section is equal to the whole number, in 痠, of 1/8 of the amount in MCM #110, #111 and #112. The remainder of the whole number will be added to the next section.

Example: Segment length = 100.00 mm and the amount of compensation is 0.026 mm as set in MCM #110. Then, the compensation for each section is  $0.026/8 = 0.00325$  mm. The detailed compensation sequence for this segment is tabulated below:

Section	Tool Position	Avg. comp. for each section	Actual comp. at each section	Accumulated compensation
1	12.5	0.00325	0.003	0.003
2	25	0.00325	0.003	0.006
3	37.5	0.00325	0.003	0.009
4	50	0.00325	0.004	0.013
5	62.5	0.00325	0.003	0.016
6	75	0.00325	0.003	0.019
7	87.5	0.00325	0.003	0.022
8	100	0.00325	0.004	0.026

110. Amount Of Compensation For Each Segment On X-axis, in mm
111. Amount Of Compensation For Each Segment On Y-axis, in mm
112. Amount Of Compensation For Each Segment On Z-axis, in mm



Format: X=□€.€€€ Y=□€.€€€ Z=□€.€€€

The table below is for user's convenience. This table is identical to the one you see on the CRT screen for MCM #112~#112. Note that the compensation value is in incremental mode. If the number of segment is less than 40, please fill the un-compensated segments with zero (0) to avoid any potential errors.

Pitch Error Compensation For Each Segment

Segment Number	MCM #110, X-axis	MCM #111, Y-axis	MCM #112, Z-axis
0			
1			
2			
3			
4			
5			
6			
7			
8			
9			
0			
1			
2			
3			
4			
5			
6			
7			
8			
9			
0			
1			
2			
3			
4			
5			
6			
7			
8			
9			
0			
1			
2			
3			
4			
5			
6			
7			
8			
9			

115. PLC Timer, in seconds

Format: 1=□€€.€€€ 2=□€€.€€€ 3=□€€.€€€ 4=□€€.€€€

The timer here is identical to those for PLC, but the time base can be any number as set by the format. Total 4 timers are available, as designated as 1, 2, 3, 4. For details, please refer to the Connecting Manual, C120~C123 (Timer on) and S120~S123 (Timer off).

116. Metric/Inch Mode, in mm/inch

Format: N=□€

N=0 The length measurement in Metric mode, mm.

N=1 The length measurement in Inch mode, inch.

**MCM parameters 120~123 are for HUST I-11 Controller Only**

120. Cycle Clearing (Machine Position)

Format: X=□€ Y=□€ Z=□€ B=□€

Setting = 0, NO, do not clear the coordinate of the machine position.

Setting = 1, YES, clear the coordinate of the machine position.

When the setting=1, the coordinate of the machine position in the CNC controller will be cleared to zero (0) when the program execution encounters M99 block. In other words, the current tool position is the machine origin after cycle clearing.

121. Master/Slave Mode Setting, N=0 None, X=1, Y=2, Z=3, B=4

Format: N=□€

N=0, CNC mode

N=1, Master/slave mode with X-axis as the master axis, the rest as slave axes.

N=2, Master/slave mode with Y-axis as the master axis, the rest as slave axes.

N=3, Master/slave mode with Z-axis as the master axis, the rest as slave axes.

N=4, Master/slave mode with B-axis as the master axis, the rest as slave axes.

122. Counter Limit

Format: N=□€€€€□€€€

Set the max. number of times for program execution. Also see MCM #74, Current Counter. The max. setting value is 9,999,999.

123. S Curve Acc. Dec. Mode and S Curve Acc. Dec. Time

Format: N=□€□ (Mode), T=€€€ (Time in 1/1000 seconds)

N=0, Linear or Exponential type accel./decel. for motor. (See MCM #44)

N=1, "S" curve type accel./decel. for motor.

T=4~1024, Set acceleration/deceleration time for "S" curve. The actual time for acceleration or deceleration will be twice the setting value.

**MCM Parameter Setting Table (HUST M/I-11 CNC Controller)**

MCM No.	Axis or Parameter	Unit	Description	Setting
1	X, Y, Z, B	mm	Work coordinate (origin), G54~G59	
2	X, Y, Z, B	pulse	Backlash, 0~255	
3	F	mm/min	JOG Feed-rate	
6	S, D		Program block number, initial & increment	
20	R	mm	Radius compensation	
21	R	mm	Tool radius wear compensation	
22		mm	Tool offset compensation	

Following parameters are protected.

MCM No.	Axis or Parameter	Unit	Description	Setting
31	X, Y, Z, B	μm	Machine system resolution	
32	X, Y, Z, B	0/1	HOME direction (+/-)	
33	Z	mm	G83 Clearance (For M-11 only)	
34	X, Y, Z, B	mm/min	Traverse speed limit	
35	X, Y, Z, B	mm/min	HOME speed limit	
36	S	rpm	Max. spindle rpm @ 10 volts	
37	B	0/1	The 4th axis condition	
38	F	mm/min	Home grid speed during HOME execution	
39	N	--	Position gain, 32~256	
42	X, Y, Z, B	0/1	Direction of motor rotation, 0=CW, 1=CCW	
43	X, Y, Z, B	1~65536	Encoder multiplication factor	
44	G00	msec	G00 accel./decel. time, 1~1000	
	G01	msec	G01 accel./decel. time, 1~500	
45	N	pulse/rev	Spindle encoder pulse per revolution	
60	X, Y, Z, B	mm	G28 reference point coordinate	
61	X, Y, Z, B	mm	G30 reference point coordinate	
70	N	bits/sec	RS232C Baud rate	
71	X (Y, Z)	mm	CRT screen horizontal length	
72	X, Y, Z	mm	Starting point (origin) for CRT graph	
73	X-Y-Z	0~4	Graphic plane	
74	N	--	Current counter	
80	X, Y, Z, B	mm	Software OT limit, (+) direction	
81	X, Y, Z, B	mm	Software OT limit, (-) direction	
82	X, Y, Z, B	mm	HOME shift data	
93	N	0/1	Offset compensation clear on RESET key, 0=Y, 1=N	
100	X, Y, Z	0/-1/1	Pitch error compensation mode setting	
101	X, Y, Z	mm	Segment length for pitch error compensation	
110	X	mm	X-axis, compensation amount each segment	
111	Y	mm	Y-axis, compensation amount each segment	
112	Z	mm	Z-axis, compensation amount each segment	
115	1, 2, 3, 4	sec	PLC Timer	
116	N	0/1	Metric (0)/Inch (1) mode	

Parameters 120~123 are for HUST I-11 controller only.

MCM No.	Axis or Parameter	Unit	Description	Setting
120	X, Y, Z, B	0/1	Cycle clearing (machine position)	
121	X, Y, Z, B	0~4	Master/Slave mode setting	
122	N		Counter limit	
123	N, T		Type & time setting for motor accel./decel. N=0 linear, N=1 "S" curve, T=4~1024 ms	

## 8 MANUAL OPERATION

This Chapter is to discuss the application and the usage of the function keys (soft-key) on Pages 3~9 of Fig 5-3.

### 8.1 Manual operation

#### 8.1.1 Homing to Machine Origin (HOME)

1. Press RESET key to put the controller in a power-on status.
2. Press POSITN key to be in coordinate display mode. (for visual inspection)
3. Press HOME-X (HOME-Y, HOME-Z, HOME-B) key to enable the function. Notice the reversed display "**\*HOME\*XAXIS**" on the first line.
4. Press the external key CYCST (use C063 to install) to execute. Notice the reversed display "**\*HOME\*XAXIS\*IN PROCESS**" on the first line.

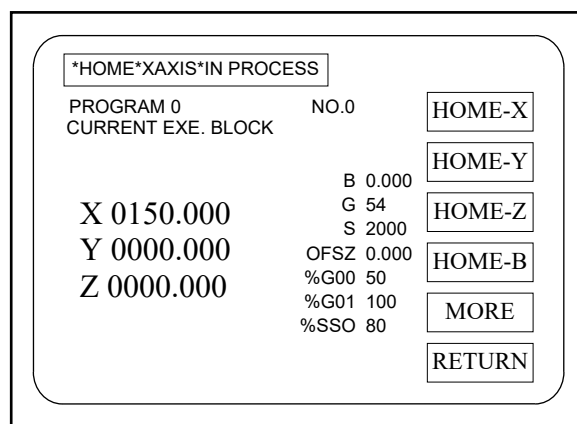


Fig 8-1 Coordinate Display (HOME-X in Process)

1. Perform "Homing to machine origin" when the controller is powered-on.
2. "Homing to machine origin" is performed one axis at a time.
3. If the tool exceeds the Home-limit switch, move the tool manually, using JOG function, inside the Home-limit switch.

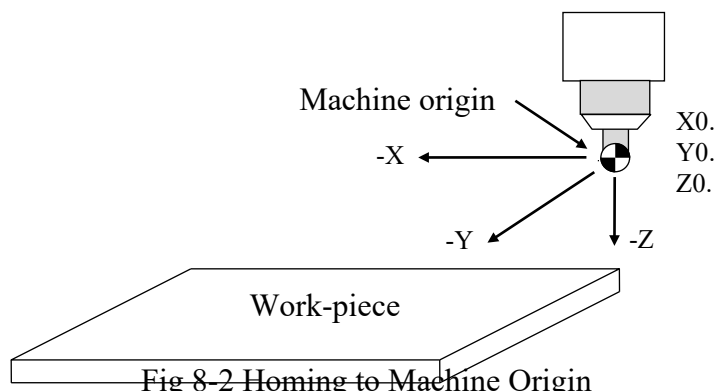
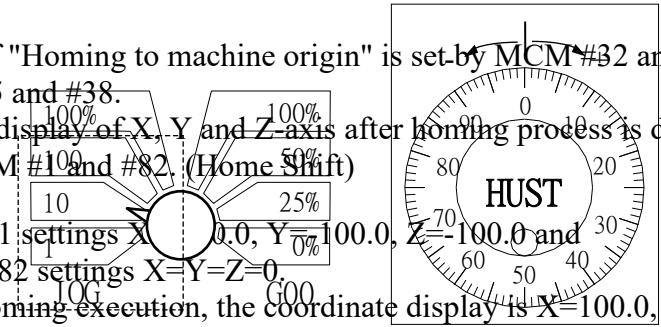


Fig 8-2 Homing to Machine Origin

4. The direction of "Homing to machine origin" is set by MCM #32 and the homing speed is by MCM #35 and #38.
5. The coordinate display of X, Y and Z axis after homing process is determined by the settings of MCM #81 and #82. (Home Shift)

Example: MCM #1 settings X=0.0, Y=100.0, Z=-100.0 and  
 MCM #82 settings X=Y=Z=0.  
 After homing execution, the coordinate display is X=100.0, Y=100.0, Z=100.0



If MCM #82 settings change to X=10.0, Y=10.0, Z=20.0. Then,  
 After homing execution, the coordinate display is X=110.0, Y=110.0, Z=120.0.

### 8.1.2 Manual JOG Feed Operation

There are two ways for manual JOG feed operation.

1. Use the external switch and the signal processed by PLC ladder.
2. Use a MPG hand-wheel.

Operation steps:

1. Press JOG-X (JOG-Y, JOG-Z, JOG-B) key to enable the JOG feed function.
2. Press POSITN key to be in coordinate display mode. (for visual inspection)
3. Press [+] or [-] key (external keys using C016~C019) to start the feeding. Release [+] or [-] key, feeding stops. Notice the reversed display "\*JOG\*XAXIS\*IN PROCESS".

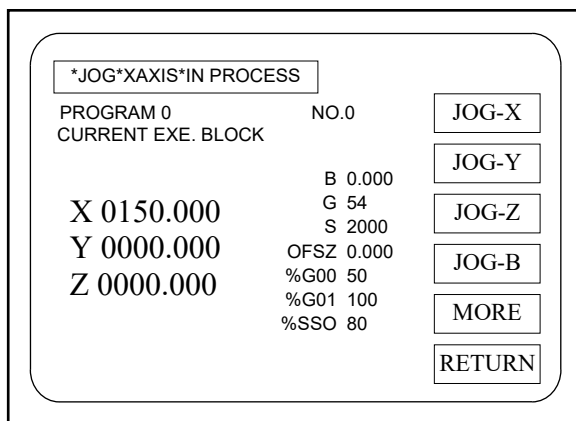


Fig 8-3 Coordinate Display (JOG)

- 3A. If use a MPG hand-wheel, turn the hand-wheel and the tool will move in the direction as indicated on the hand-wheel. When stop turning, motion stops also. Note that, in Fig 8-4, the JOG multiplication factors of 1, 10, 100 are processed by C024~C025 bits in the PLC and the switch is externally installed. The G00 factors are by C012~C013 bits.

Fig 8-4 MPG Hand-wheel

### 8.1.3 G01 Manual Feed-rate Override (MFO %)

HUST provides two ways for G01 manual feed-rate override, which is controlled by C104 bit in the PLC as:

1. When C104=0, MFO adjustment is controlled by C040~C043 in the PLC ladder.
2. When C104=1, MFO adjustment is controlled by MPG hand-wheel.

#### Adjustment by C040~C043 when C104 = 0.

The adjustment knob is installed by the user, with the input signals to be processed by C040 ~ C043 in the PLC. User can adjust the feed-rate any time during the machining process. The adjustment range is 0 ~ 150% with 10% increment and the standard PLC is fixed at 100%. Please see "Connecting Manual" for details.

Example: Programmed feed-rate  $F = 500.00$  mm/min  
 If MFO% = 120%, the actual feed-rate  $F = 500.0 * 120\% = 600.0$   
 If MFO% = 80%, the actual feed-rate  $F = 500.0 * 80\% = 400.0$

#### Adjustment by MPG hand-wheel when C104 = 1.

When C104 = 1, the override adjustment is internally controlled and the controllable range is 0, 1, 2, 3, 4, ..... 249, 250%. If

- C105=1, Use MPG hand-wheel to adjust the speed to any number between 0~250%.
- C105=0, the override adjustment is no longer controlled by MPG.

### 8.1.4 G00 Manual Feed-rate Override

The adjustment knob is installed by the user, with the input signals to be processed by C012 ~ C013 in the PLC. User can adjust the feed-rate any time during the machining process. The adjustment range is 0 ~ 100% with 25% increment and the standard PLC is fixed at 100%. Please see "Connecting Manual" for details.

### 8.1.5 Manual Spindle Speed Override (SSO, %) and Rotating Direction

HUST 11-series controller provides two ways for spindle speed manual override, which is controlled by C106 bit in the PLC as:

- When C106=0, spindle adjustment is controlled by C108~C111 in the PLC ladder.
- When C106=1, spindle adjustment is controlled by MPG hand-wheel.

Adjustment by C108~C111 when C106 = 0

The SSO adjustment knob is installed by the user, with the input signals to be processed by C108 ~ C111 in the PLC. User can adjust the spindle speed any time during the machining process. The adjustment range is 0 ~ 150% with 10% increment and the standard PLC is fixed at 100%. Please see "Connecting Manual" for details.

Example: Programmed spindle speed S = 1000 rpm  
 If SSO = 120%, the actual spindle rpm S = 1000\*120% = 1200  
 If SSO = 80%, the actual spindle rpm S = 1000\*80% = 800

Adjustment by MPG hand-wheel when C106 = 1

When C106 = 1, the override adjustment is internally controlled and the controllable range is 0, 1, 2, 3, 4, ..... 249, 250%. If

- C107=1, use MPG hand-wheel to adjust the speed to any number between 0~250%.
- C107=0, the override adjustment is no longer controlled by MPG.

Direction of spindle rotation

The direction of spindle rotation can be controlled by the C103 bit in the PLC . In most cases, the direction of spindle rotation can be achieved by a simple connection on the CNC output when C103 = 0. In the cases that this is not possible, use C103 bit in the PLC and set C103 = 1 to produce the reversed rotation.

- C103 = 0, CNC output 0~10 volts, direction controlled by CNC output connection.
- C103 = 1, CNC output 0~-10 volts, spindle rotates in reversed direction.

**8.2 MDI Single Block Operation, MDI**

MDI function enables the user to enter a single block of program and execute it, or enter data into the controller by G10. Once the operation is finished, MDI command disappears. Followings are the situations when you can use MDI operation.

1. Single program block execution
2. Locking and un-locking of MCM parameters by G10 -- see Sec 7.1
3. Work coordinate origin setting by G10
4. Tool offset compensation input by G10

**8.2.1 Single Program Block Input and Execution**

Execution steps:

1. Press MDI key to enable MDI function. Notice "\*MDI" in reversed display.

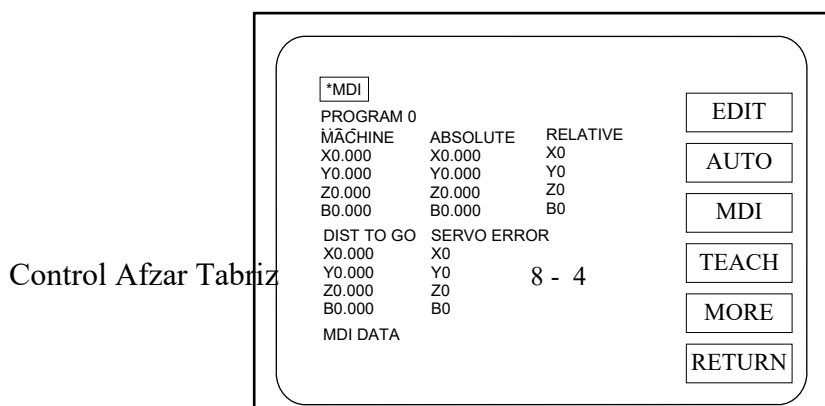




Fig 8-5 MDI Display - 1

2. Key in the desired single block of program as follow:

Example: [G] - [0] - ENTER  
 [X] - [0] - [.] - ENTER  
 [Y] - [0] - [.] - ENTER  
 [Z] - [1] - [0] - [.] - ENTER

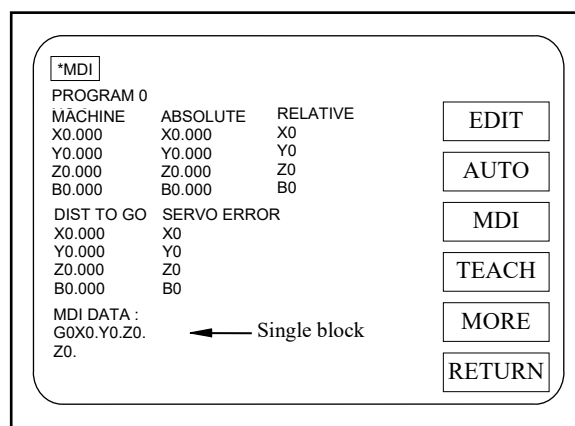


Fig 8-6 MDI Display - 2

3. Press CYCST to start execution. When the execution is finished, the single block program is disappeared.

## 8.2.2 G54~G59 Work Coordinate System Setting, G10

### The setting steps of G54 work coordinate origin (G10 method)

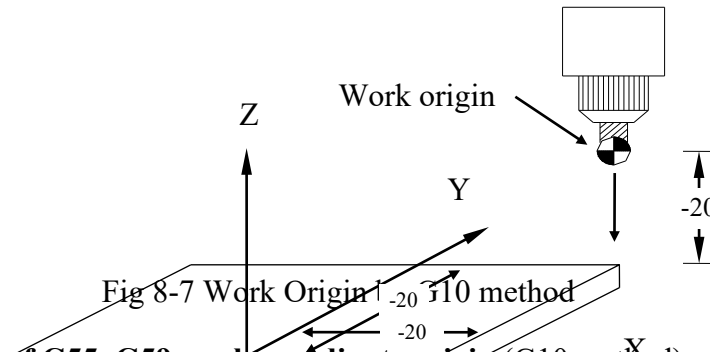
Format: G10 X\_\_\_Y\_\_\_Z\_\_\_B\_\_\_

1. Execute HOME-X (HOME-Y, HOME-Z, HOME-B) to move the tool to home position.
2. Press JOG-X (JOG-Y, JOG-Z, JOG-B) key to enable the JOG feed function. Then, use the MPG hand-wheel to move the tool or the work-table to the desired coordinate origin position.
3. Press MDI key to enable the MDI function.
4. Use G10 to input the coordinate of the current tool position with respect to the desired origin position in step 2. Since the current tool position is the desired origin position in step 2, the coordinate is X0.0, Y0.0, Z0.0. Input the single block data as follows:

[G] - [1] - [0] - ENTER  
 [X] - [0] - [.] - ENTER  
 [Y] - [0] - [.] - ENTER  
 [Z] - [0] - [.] - ENTER

5. Press CYCST to complete the setting procedure. The coordinate of the first line in MCM #1 is adjusted automatically.

In the case that the tool is unable to be moved to the desired origin location in step 2, obtain the coordinate of the tool position with respect to the desired origin position, then input the data as in step 4. For example, if the current tool position is X-30.0, Y-20.0, Z-20.0 away from the desired origin, input the data as: G10 X-30.0 Y-20.0 Z-20.0.



**The setting steps of G55~G59 work coordinate origin (G10 method)**

The setting steps of G55~G59 work coordinate origin are identical to those for G54 except that the data input format in step 4 is added one more term to specify the coordinate system number.

Format: G10 X\_\_\_ Y\_\_\_ Z\_\_\_ B\_\_\_ P?

- where P?=
- P01, For G55 work coordinate origin setting.
  - P02, For G56 work coordinate origin setting.
  - P03, For G57 work coordinate origin setting.
  - P04, For G58 work coordinate origin setting.
  - P05, For G59 work coordinate origin setting.

### 8.2.3 Tool Offset Compensation Setting, G10

Format: G10 Z\_\_\_ P1? (To place data Z\_\_\_ into Tool #? of MCM #22)  
 G10 W\_\_\_ P1? (To add data W\_\_\_ into Tool #? of MCM #22)

P1?: 1, Indicate that the data input is for tool offset compensation.  
 ?, The tool number (1~16) of MCM #22.

1. Press MDI key to enable the MDI function.
2. Input the compensation data as in the format: G10 Z\_\_\_ P1?.  
 G10 Z3.000 P102 → to place comp. data 3.000 into tool #2 of MCM #22.  
 G10 W0.01 P102 → to add comp. data 0.01 into tool #2 of MCM #22.  
 If Z=3.00, add 0.01, result Z=3.01.
5. Press CYCST to complete the setting procedure.

### 8.3 Auto Execution, AUTO

The "AUTO" function enables the user to execute a program continuously until the end. When executing this function, be sure that the tool is within the hardware and software over-travel limit and that there is no obstruction along the tool path.

Execution steps:

1. Press PRNO to select the desired program.
2. Press AUTO key to enable the function.
3. Press CYCST key to start execution. The whole program will be executed until the end. Notice the "**\*AUTO\*IN PROCESS**" in the reversed display.

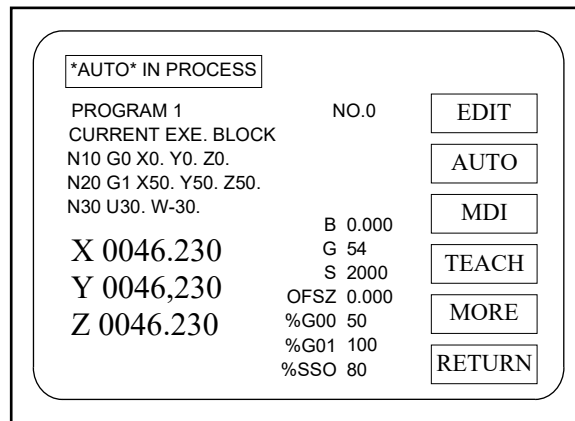


Fig 8-8 AUTO Execution Display

#### 8.4 Single Block Execution in AUTO Mode, SINGLE

This function allows the user to execute a program one block at a time until the end if desired. The SINGLE function can not be used in a program with canned cycle cutting.

Execution steps:

1. Press PRNO to select the desired program.
2. Press AUTO and SINGLE key to enable the function.
3. Press CYCST key to start the execution of the first block. When the execution stops, press CYCST again to execute the next block. Repeat the same procedure until the end. Notice the "**\*SINGLE\*IN PROCESS**" in the reversed display.

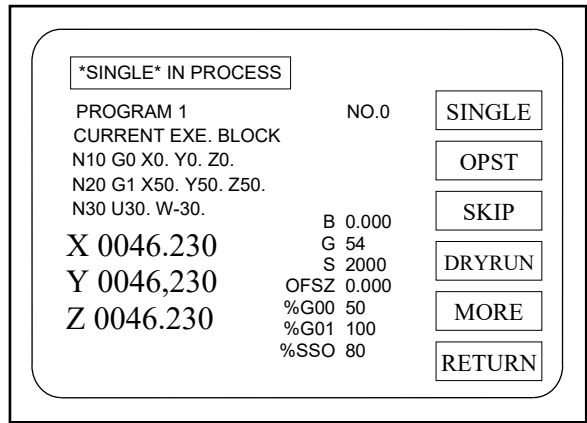


Fig 8-9 SINGLE Execution Display

To cancel SINGLE mode execution, simply press SINGLE key one more time. If you want to continue execution from where the SINGLE execution was stopped, press CYCST.

### 8.5 Option Stop (OPST)

This function is valid only when M1 command code is present in the program. When OPST function is enabled, the program execution will stop at the block with M1 command. When the CYCST key is pressed again, the program execution will resume from the M1 block.

Execution steps:

1. Enter M1 code in the program where you want the program execution to stop.
2. Press OPST key to enable the option stop function.
3. Press AUTO and CYCST key to execute the program.
4. When the execution runs into M1 block, the execution stops.
5. Press CYCST key to resume program execution.

Notice that the OPST function can not be used in the program with canned cycle cutting. If the OPST function is NOT enabled, the execution will ignore and skip the M1 block and continue executing the next block.

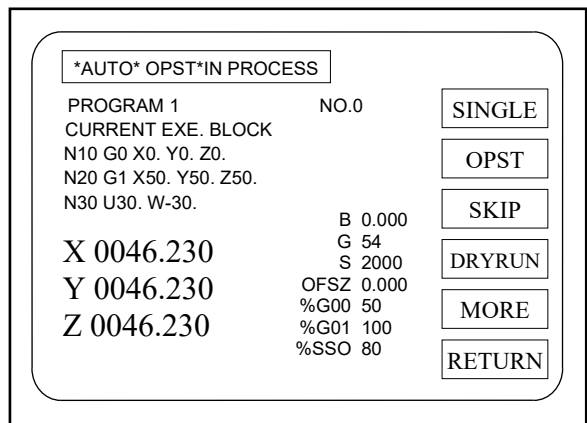


Fig 8-10 OPST Execution Display

Example:

```

N10 G0 X20.0 Y20.0 Z0.0
N20 G1 W-20.0 F200.
N30 M1          ..... If OPST function is enabled, the execution will stop at this
                  block. Press CYCST key will resume execution from
N40
N40 G1 X30.0 F300.
N50 X50.0 Y0.0
N60 G0 X0.0 Y0.0 Z0.0
M70 M2

```

## 8.6 Skip Function, SKIP

When SKIP function is effective, the program execution will ignore and skip the block containing the "/" code and continue execution from the next block. Otherwise, the block with "/" code will be treated as normal block.

Execution steps:

1. Add the "/" code in the program block to be skipped.
2. Press SKIP key to enable the skip function.
3. Press AUTO and CYCST key to execute the program.
4. The execution will skip the block containing the "/" code and continue execution from the next block.

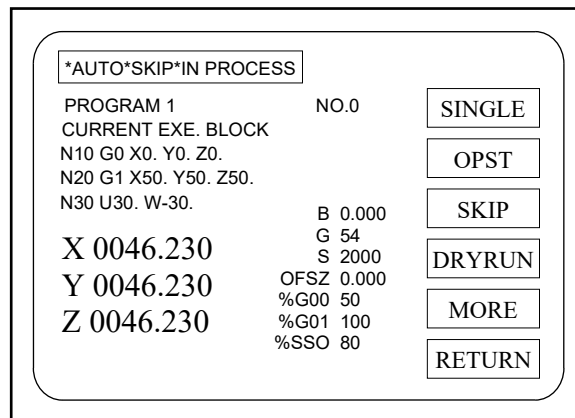


Fig 8-11 SKIP Function Display

Notice that the SKIP function can not be used in a program with canned cutting cycle.

Example:

```

N10 G0 X0.0 Y0.0 Z0.0
N20 G1 X50.0 Y50.0 Z50.0 F200.
N30 U30.0 W-30.0 /1 ..... If the SKIP function is enabled, execution will skip this
                  block and continue from the next block, N40.
N40 G1 X30.0 F300.
N50 G0 X0.0 Y0.0
N60 M2

```

### 8.7 Program DRYRUN

DRYRUN function is used to test a program with G00 high speed. If DRYRUN key is pressed during program execution, the controller will ignore all programmed feed-rates (F-values) and execute at G00 high speed until the program end. Before activating this function, be sure that there is no any obstruction along the tool path in order to avoiding any potential damages to the equipment. HUST M-11/I-11 provides two types of DRYRUN based on the status of C000-bit.

1. C000 = 0      DRYRUN with servo motor action.
2. C000 = 1      DRYRUN without servo motor action.

Execution steps:

1. Press PRNO key to select the program for dryrun.
2. Press AUTO key to enable the auto execution function.
3. Press DRYRUN key to enable the dryrun function.
4. Press GRAPH key for visual inspection of the tool path.
5. Press CYCST to start dryrun. If you use coordinate display for visual inspection, the screen display is as follow:

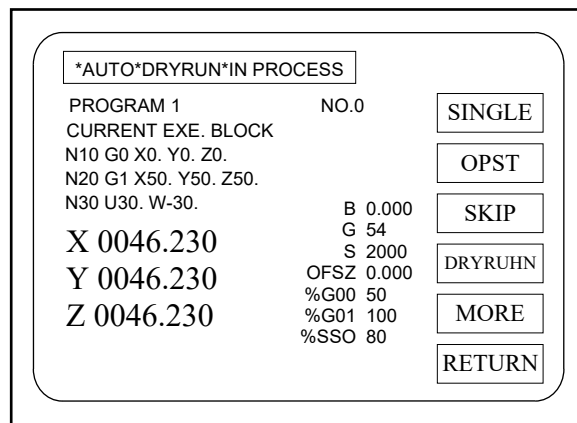


Fig 8-12 DRYRUN Function Display

### 8.8 MPG Hand-wheel Testing

In addition to the DRYRUN function for program testing, HUST 11-series also provides another testing function by a MPG hand-wheel. The advantage of the MPG testing is that the user can do actual cutting using a MPG hand-wheel. Any errors in the program can be detected and the product inspected before making mass production.

Customer can press the function key on HUST controller to activate the MPG testing function, press one more time to cancel it. Or, you can use C056 bit in the PLC for external control. Please see "Connecting Manual" for details. During testing, the feed-rate by MPG hand-wheel is determined from the MFO% switch.

MPG testing function steps:

1. Select the program and the feed-rate (MFO% switch).
2. Press AUTO and GRAPH for visual inspection.

3. Press TEST to enable MPG testing function.
4. Press CYCST.
5. Rotate MPG to start testing. When you stop MPG, the testing also stops. If you press TEST key one more time to cancel the MPG testing mode, the controller will execute the program with the normal feed-rate.

## 8.9 Program Re-start, RE-STA

The program restart function allows the user to restart the execution from where the program was interrupted. User must know the exact location of program interruption when applying the RE-STA function, which is processed by C011 bit in the PLC. When C011=1, RE-STA function is enabled.

Execution steps:

1. Press RESET key and use MPG to move the tool away from the work-piece. If the interruption is due to EM-stop or servo error (Error 2), execute "HOME" prior to pressing RESET.
2. Press RE-STA key to enable the restart function.  
If S028=1 or I032=1, the bit C011=1 as shown below:



3. Press AUTO key, then PROGRAM key to display the interrupted program.
4. Use Cursor $\uparrow$ , Cursor $\downarrow$ , Page $\uparrow$ , or Page $\downarrow$  key to move the cursor to the block where the program was interrupted or use LAST-N function. (see Sec 6.3)
5. Press CYCST to start the RE-STA function.

Notes: M02, M30, or M99 in the program must be processed through PLC in order to cancel the RE-STA function. During RE-STA execution, the M-, T-, S-code in the program before the interrupted block will be executed again.

Example: Program 2 (Fig 8-13)

Work origin at X=-150.0, Z=-250.0.

Execute "HOME" to move the tool to the machine origin.

N10 S2000

N15 G0 X100. Z0.

N20 G1 W50. F200.

N30 G1 U-20.

N40 U-20. W60. .... program was interrupted at this block, restart from here.

N50 U-20.

N60 U-20. W80.

N65 U-20.

N70 G0 X150. Z250.

N80 M2

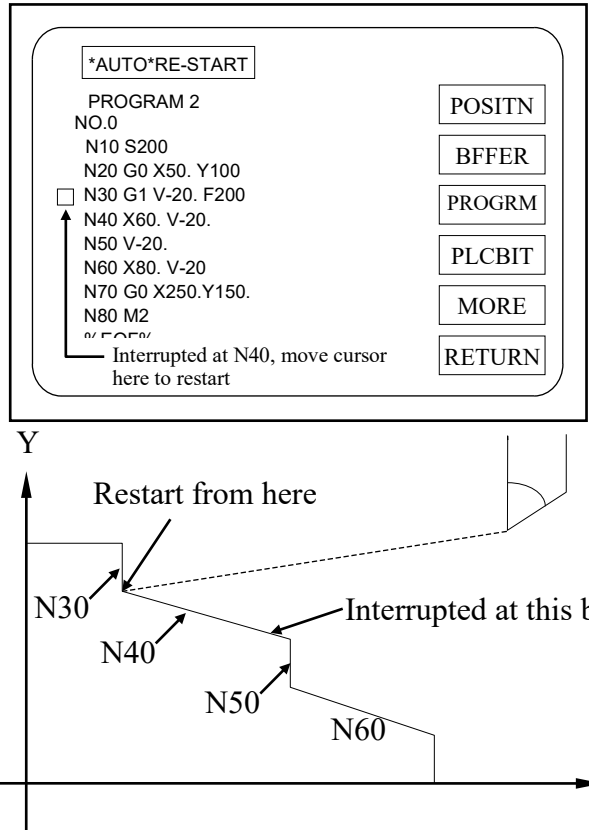


Fig 8-13 Program Restart (RE-STA)

Move cursor to N40 and activate the restart function. The controller will calculate the coordinate change from N10 to N30, then move the tool to the end of N30 and continue the program execution from there.

### 8.10 Round Corner Non-stop Operation

When executing two program blocks with the tool going in the different direction, the intersect normally forms a sharp angle and the motor will go through deceleration and acceleration. With this operating condition, some machine such as glue machine, flame or laser cutting machine can not obtain a satisfactory result. To overcome this problem, HUST 11-series provides a round corner non-stop operation.

There is no function key available on the HUST keyboard. However, customer can use the input point (I033) in the standard PLC and the bit C036. When I033=1 and C036=1, the round corner non-stop operation is enabled. As in Fig 8-14, the distance  $d$  ( $SP = PE$ ) can be calculated from the equation below:

$$d = 0.5 * \text{feed-rate, } F * \text{acceleration/deceleration time}$$

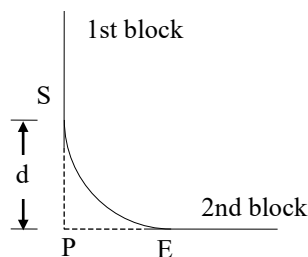




Fig 8-14 Round Corner Non-stop Operation

Example:  $F = 500. \text{ mm/min.}$  Time for acceleration/Deceleration = 300 ms  
 $d = 0.5 * 500/60 * 300/1000 = 1.25 \text{ mm}$

### 8.11 Feed Hold (F-HOLD)

When F-HOLD key is pressed during automatic cutting operation, the program execution will be temporarily put on HOLD.

Execution steps:

1. The program is in AUTO execution mode.
2. Press F-HOLD key, the program execution will be temporarily put on HOLD.
3. Press CYCST key, the program execution will resume from the point where the program was stopped in step 2.

### 8.12 CLR-XR, CLR-YR, CLR-ZR, CLR-BR Function

These keys are used to clear the relative coordinates when using G10 command to set the work origin as shown in Fig 8-15.

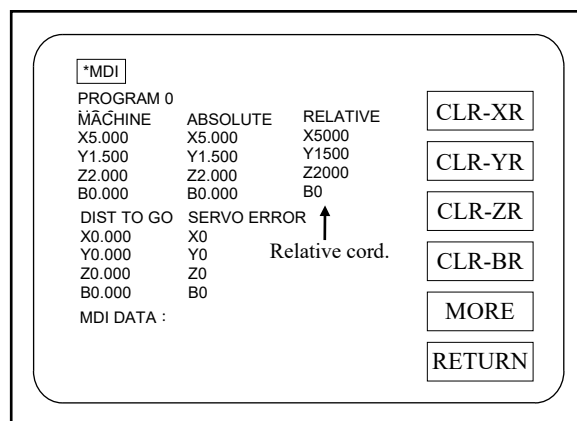


Fig 8-15 Relative Coordinate



## 9 PC ON-LINE OPERATION - RS232C

Through the TAPE function key (see Fig 5-3), HUST 11-series can do the following PC (personal computer) on-line operations via RS232C interface.

1. Transfer part program from PC (personal computer) to CNC controller.
2. Transfer part program from CNC controller to PC.
3. Transfer part program from PC to CNC controller and execute the program.
4. Transfer PLC ladder program from PC to CNC controller and test the ladder program.

The TAPE function is processed by C008~C010 and O096~O099 in the PLC. Please refer to "Connecting Manual" for details.

### 9.1 Program Transfer From PC To CNC Controller (Reader In)

1. On the PC side, execute "DNC" to bring up the main menu and do Steps 1, 2, 3 on the main menu as explained in Sec. 9.6-A. Make sure communication protocol is correct.
2. On the CNC controller side, select the program number (PRNO) that is to be transferred. Then, press RESET key.
3. Use cursor up/down to select "Reader In". Notice the display of READER IN at the bottom of the screen.

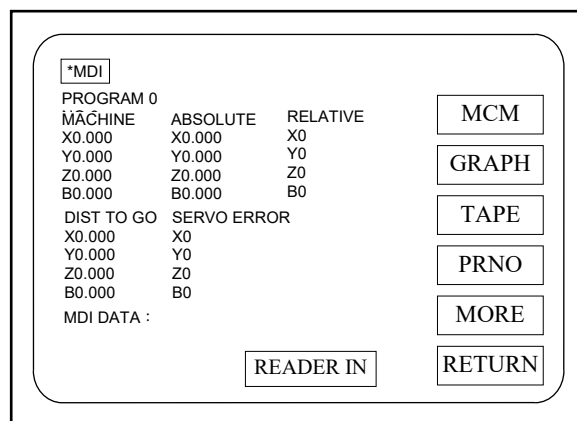


Fig 9-1 TAPE function - READER IN

4. Press CYCST to start program transmission to the controller. When the transmission is completed, a message of "Tape Found" will be displayed at the bottom of the screen.

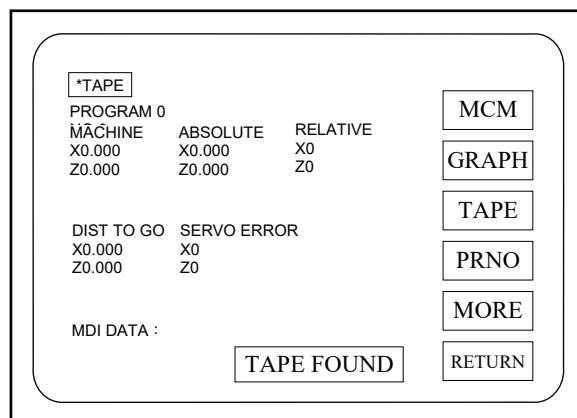


Fig 9-2 TAPE function - TAPE FOUND

### 9.2 Program Transfer From CNC Controller To PC (Punch Out)

1. On the PC side, execute "DNC" to bring up the main menu and do Steps 1, 2, 3 on the main menu as explained in Sec. 9.6-B. Make sure communication protocol is correct.
2. On the CNC controller side, select the program number (PRNO) that is to be transferred. Then, press RESET key.
3. Use cursor up/down to select "Punch Out". Notice the display of PUNCH OUT at the bottom of the screen.

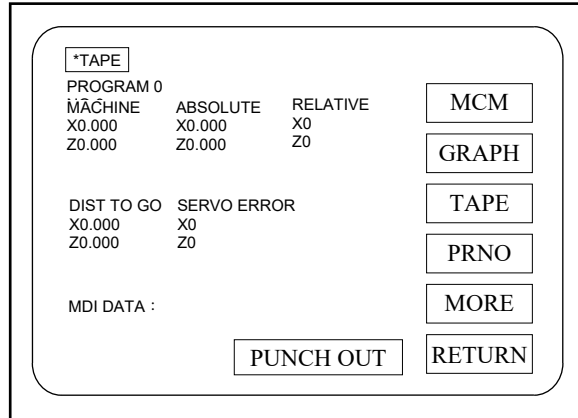


Fig 9-3 TAPE Function - PUNCH OUT

4. Press CYCST to start program transmission to the PC. When the transmission is completed, a message of "Tape End" will be displayed at the bottom of the screen.

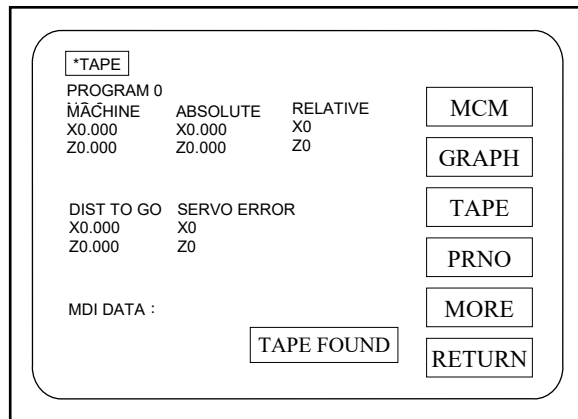


Fig 9-4 TAPE Function - TAPE END

Notes: When you press TAPE key, READER IN, PUNCH OUT, TAPE EXE, LADDER IN will appear consecutively.

### 9.3 Transfer part program from PC to controller and execute the program (Tape Exe.)

1. On the PC side, execute "DNC" to bring up the main menu and do Steps 1, 2, 3 on the main menu as explained in Sec. 9.6-A. Make sure communication protocol is correct.
2. On the CNC controller side, select the program number (PRNO) that is to be transferred. Then, press RESET key.
3. Use cursor up/down to select "Tape exe". Notice the display of TAPE EXE at the bottom of the screen.

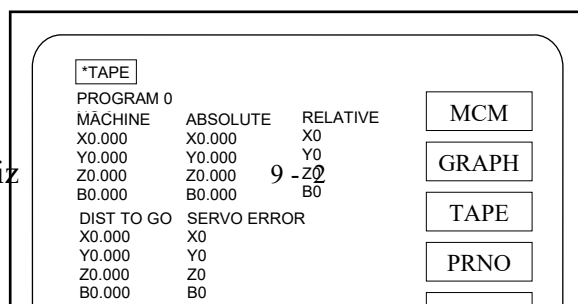


Fig 9-5 TAPE function - TAPE EXE

4. Press CYCST to start program transmission to the controller which will execute the program block by block.

#### 9.4 PLC Ladder Transfer From PC To CNC And Ladder Test (Ladder In and Ladder Simulation)

1. On the PC side, execute "DNC" to bring up the main menu and do Steps 1, 2, 3 on the main menu as explained in Sec. 9.6A. Make sure communication protocol is correct.
2. On the CNC side, press RESET key.
3. Use cursor up/down to select "Ladder In". Notice the display LADDER IN at the bottom of the screen.

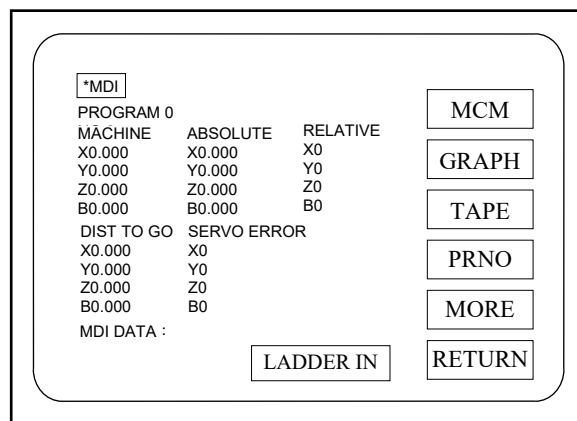


Fig 9-6 TAPE Function - LADDER IN

4. Press CYCST to start PLC program transmission to the CNC. When the transmission is completed, a message of "%" will be displayed at the bottom of the screen.

Now you can do ladder simulation if the bit C031=1. Notice the reversed display of "S-PLC" at the top of the screen. When C031=0, ladder simulation is OFF. The max. PLC ladder capacity for simulation is 16 K.

Note: LADDER IN function is designed for machine manufacturer, NOT for end users.

## 9.5 Program Format for PC On-line Transfer

The format rules for the program to be transfer between PC and HUST CNC controller are as follows:

1. The program must be started with and ended with a symbol of "%".
2. All letters must be in upper case including the program number "O".
3. One program block occupies one line of space. Do not put more than one block on the same line as follows:  
N10 G0 X0. Z0. N20 G1 X10. Z10. N30 X0. Z0. N40 M2
4. When transferring program from PC to HUST controller, the currently selected program will be written over if the program being transferred does not contain a program number "Oxxxx".
5. The zeros after decimal point may be omitted, but the decimal point must be retained.

Example:

```
%  
O0001  
N10 G0 X100.0 Z100.0  
N20 G1 X30. Z40. F.20  
N30 G2 X40. Z50. R10.  
N40 G1 X60.000 Z70.  
N50 G0 X100. Z100.  
N60 M05  
N70 M02  
%
```

## 9.6 RS232C Interface -- HUST's DNC.EXE Software

HUST provides "DNC.EXE" software for program/data communication between PC and HUST controller. This software is of menu driven and easy to use.

### DNC.EXE installation

- Personal Computer : IBM PC or compatible with DOS 3.30 or above
- Software : HUST's "DNC.EXE"
- Copy DNC.EXE to your PC as follows:  
C:\>md dnc <ENTER>  
C:\>cd dnc <ENTER>  
C:\DNC>copy a:\dnc.exe <ENTER>

**DNC.EXE program execution**

Be sure you are in the DNC directory. Key in DNC and press ENTER. The main menu will be on the screen as below:

```

DNC MODULE

  1. COMMUNICATION PROTOCOL
  2. NC FILE NAME
  3. COMPUTER ---> CONTROLLER
  4. CONTROLLER ---> COMPUTER
  5. LIST NC FILE NAMES
  6. LIST NC FILE CONTENT
  9. QUIT

PLEASE CHOOSE .....

```

Main menu description:

1. Communication protocol -- For setting port, baud rate, parity, etc.
2. NC file name -- Program name to be transmitted with directory path.
3. Computer → Controller -- Transfer from PC to CNC
4. Controller → Computer -- Transfer from CNC to PC
5. List NC file names -- List all NC files in current directory on screen
6. List NC file content -- List the content of the file selected in step 2
9. Quit -- Quit DNC operation

**A. Procedures for program transfer from PC to CNC controller**

1. On the main menu, select "1" and the screen display is shown below. Check all the settings for communication are the same as those in MCM parameter #70.

```

PROTOCOL

  1. RS232C PORT (COM1)
  2. BAUD RATE (4800 BPS)
  3. PARITY CHECK (EVEN)
  4. WORD LENGTH (7 BITS)
  5. STOP BITS (2 BITS)

PLEASE CHOOSE

(PRESS RETURN TO QUIT)

```

2. On the main menu, select "2" and the screen display is shown below. Type in the program file name (TEST for example) with an appropriate directory path if not in the current directory. Press ENTER and return to main menu.

```

INPUT NC FILE NAME

NO CURRENT NC FILE!

KEY IN FILE NAME (EXTENSION NAME WILL BE .NCD)

PRESS [ESC] TO QUIT

```

3. On the main menu, select "3" and the screen display is shown below.

```
DNC PUNCH MODULE  
NC FILE IS TEST.ncd!  
  
PRESS [READER IN] BUTTON ON CNC CONTROLLER
```

4. Go to HUST controller and continue from step #2 of Section 9.1 to complete the program transfer. Note that DNC.EXE software accepts the file with an extension of ".ncd" only.

### **B. Procedures for program transfer from CNC controller to PC**

1. On the main menu, select "1". Same as Procedure A-1.
2. On the main menu, select "2". Same as Procedure A-2.
3. On the main menu, select "4" and the screen display is shown below.

```
DNC READ MODULE  
NC FILE IS TEST.ncd!  
  
READER IS WORKING NOW ..... [PRESS [ESC] TO ABORT]
```

4. Go to HUST controller and continue from step #2 of Section 9.2 to complete the program transfer.

## **9.7 RS232C Connection**

A proper connection between PC and HUST controller is shown in Fig 9-7. Please refer to Connecting Manual for more information. When making connection, please be aware of the followings:

1. The connecting cable should not exceed 15 meters to minimize the potential noise interference. The voltage at the PC interface should be in the range of 10~15 volts.
2. Avoid working in an environment which is under the direct noise interference from the machines such as EDM, electric welder, etc. Do not use the same power outlet as for EDM and electric welder. Twisting the cable may help in noise reduction.



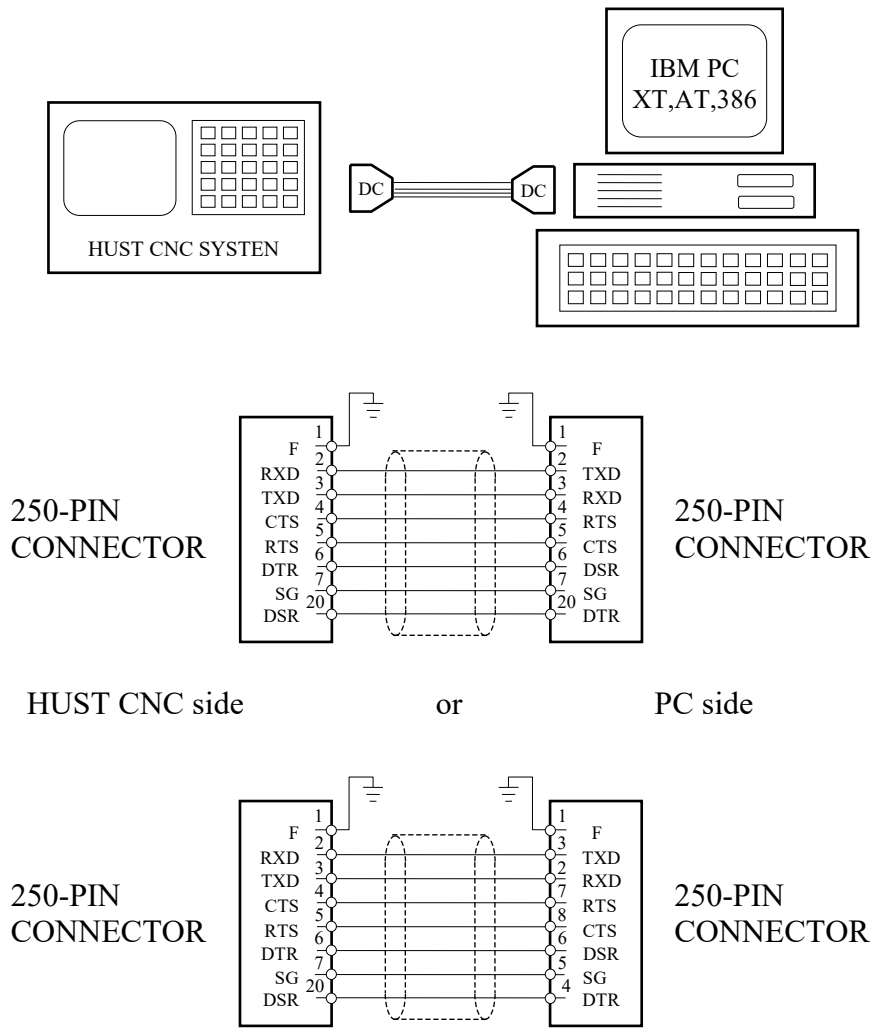


Fig 9-7 RS232C Connection



## 10 ERROR MESSAGES

When an error occurs during operation, HUST controller will be stopped and an error message displayed at the top of the screen as shown in Fig 10-1. If there are more than one error, only one error will be displayed at a time. The second error will be displayed when the first problem is resolved. This chapter is to explain the error messages and the method to solve them.

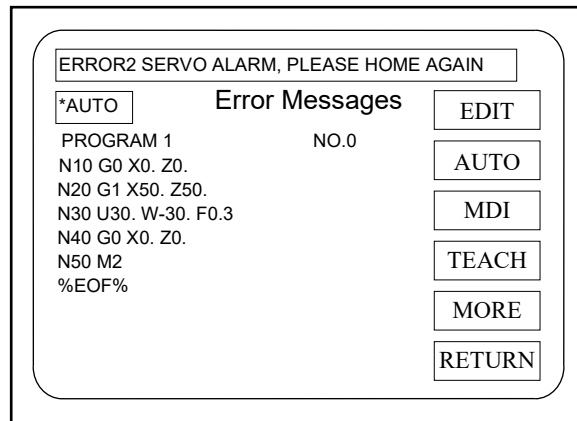


Fig 10-1 Error Message Display

### **ERROR 1** -- MCM Data Error Or Battery Fail

Message:

MCM parameter setting is incorrect or the backup battery has failed.

Recommended Remedy:

1. Check if MCM parameter setting data are correct. Otherwise, use "G10 P1000" to clear all MCM data and re-enter all MCM data. Press RESET.
2. If the problem is on the battery, turn on the controller and leave it "on" for 4 hours to recharge the backup battery. If the error persists, please contact the maintenance engineer to change out the battery. If the controller has not been turned on for a month, the data in the memory will be lost.

### **ERROR 2** -- X(Y,Z,B) Servo Alarm, Please Home Again

Message:

Servo position control system (servo feedback) error. Possible causes are:

1. The voltage command from the controller is too fast for the motor to response.
2. The controller does not receive any feedback signal from the servo motor.

Recommended Remedy:

1. Check if the feed-rate F in the part program is too fast.
2. Check if the resolution setting of MCM parameter #31 is correct.
3. Check the acceleration/deceleration time setting. (MCM #44)
4. Check if the work table being overloaded, or any obstruction in the motor. Also check the servo system including the connections.

### **ERROR 6** -- Memory Full

Message:

The RAM memory capacity in the CPU is already full.

Recommended Remedy:

Remove or delete some of the part programs in the RAM memory.

**ERROR 8** -- Exceed 48 Characters For One Block

Message:

A block of the part program exceeds 48 characters.

Recommended Remedy:

Check the part program and make sure each single block of program is less than 48 characters.

**ERROR 9** -- Too Large Ladder Program For Simulation

Message:

The PLC ladder program for simulation is too large for the memory.

Recommended Remedy:

Revise the PLC ladder program and limit its memory capacity to be less than 16 K.

**ERROR 11** -- Program Memory Error

Message:

Program memory error. Possible causes are:

1. Battery failure.
2. Part program is too large and exceeds memory capacity.

Recommended Remedy:

1. Check and revise the program, then press RESET.
2. If the program is in serious error, use G10 P2001 to clear the program.
3. If problem is on the battery, turn on the controller and leave it "on" for at least 4 hours.  
If the error persists, change out the battery.

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

Message:

Incorrect G-code in the program.

Recommended Remedy:

Check the part program for the G-code that is not acceptable by HUST controller.

**ERROR 14** -- X-axis Over-travel

**ERROR 15** -- Y-axis Over-travel

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

**ERROR 17** -- B-axis Over-travel

Message:

The cutting tool travels beyond the hardware limit in X-axis. (Error 14)

The cutting tool travels beyond the hardware limit in Y-axis. (Error 15)

The cutting tool travels beyond the hardware limit in Z-axis. (Error 16)

The cutting tool travels beyond the hardware limit in B-axis. (Error 17)

Recommended Remedy:

Use the MPG hand-wheel to manually move the tool within the operating range in X-axis.  
(or Y-, or Z-, or B-axis)

**ERROR 18** -- End Of File

Message:

Ending error in the part program.

Recommended Remedy:

Check the ending statement of the part program, such as M02, M30, M99.

**ERROR 20** -- X+ or X- Software Over-travel

Y+ or Y- Software Over-travel

Z+ or Z- Software Over-travel

B+ or B- Software Over-travel

Message:

The cutting tool travel has reached the bounding limit as set by the software.

Recommended Remedy:

Check the part program or reset the MCM #80, #81 for software travel limit.

**ERROR 22** -- Em-Stop, Home Again

Message:

Controller is in emergency stop state.

Recommended Remedy:

Resolve the cause for emergency stop. Restore Emergency-STOP button and press RESET or HOME key before normal operation.

**ERROR 24** -- M98 Exceed 5 Level

Message:

The sub-program calls exceed 5 levels.

Recommended Remedy:

Revise the part program and make sure the sub-program calls do not exceed 5 levels.

**ERROR 25** -- Wrong Circle Format

Message:

The circular cutting command (G02,G03) or the command format is in error.

Recommended Remedy:

Check the part program and recalculate the coordinate of the center of the arc/circle.

**ERROR 27** -- Not Found

Message:

Can not find the program block number as specified.

Recommended Remedy:

Check if the specified program block number is correct.

**ERROR 28** -- Exceed Counter Limit

Message:

The number in the Counter (MCM #74) exceeds the one specified by MCM #122.

Recommended Remedy:

Clear the number in the Counter or reset the number in MCM #122 to larger one.

**ERROR 31** -- None PLC

Message:

There is no PLC ladder program in the CNC's EPROM memory.

Recommended Remedy:

Re-write the PLC ladder program into the EPROM'S and insert these EPROM'S into the appropriate locations on the CPU main board .

**ERROR 33** -- Over-cutting Alarm In Radius Comp. Linear

Message:

Tool radius compensation path for linear cutting portion is in error.

Recommended Remedy:

Check the tool path in the part program while applying the tool radius compensation. If the tool paths form a sharp angle or the radius is too large, an error will occur.

**ERROR 34** -- Over-cutting Alarm In Radius Comp. Circular

Message:

Tool radius compensation path for circular cutting portion is in error.

Recommended Remedy:

Check the tool path in the part program while applying the tool radius compensation.

**ERROR 35** -- No Intersection Point In Radius Comp. Mode

Message:

The CNC controller can not resolve the intersecting point between the two consecutive program blocks while in radius compensation mode.

Recommended Remedy:

Check the part program if the tool paths of the two consecutive program blocks are in near parallel. If this is the case, please revise the part program.

**ERROR 37** -- NC Alarm

Message:

There is a mechanical problem from the machine tool side.

Recommended Remedy:

Check the machine tool for proper operation.

**ERROR 38** -- 68 K Error

Message:

MCM parameter settings in error; or HUST system software or PLC ladder in error.

Recommended Remedy:

Check the MCM parameter setting. Re-write the HUST system software or PLC ladder into the EPROM'S and change out.

**ERROR 39** -- X (or Y, or Z, or B) Feedback Error

Message:

There is a servo motor connection error for X (or Y, Z, B) axis. The servo motor tends to diverge or the servo error count exceeds 200 for the first 1 second.

Recommended Remedy:

1. Check the servo motor connection. 2. Adjust the servo motor for proper operation.





## CHAPTER 12

### MAINTENANCE

#### 12.1 MAINTENANCE OF CNC CONTROLLER

The maintenance of CNC controller starts once the controller is installed, adjusted, and handed over to the customers. The sole purpose of the maintenance is to keep the CNC controller running normally and smoothly throughout its life. In order to achieve this purpose, each CNC manufacturer requires some installation conditions that need to be satisfied, such as proper ambient temperature, voltage surge protector, sufficient distance of unit to the wall, and proper grounding, etc. The target of CNC running rate is to keep it functioning at all times.

HUST has been striving to give you the best and most durable controller. However, the end users are responsible for the proper maintenance in order to achieve the highest running rate. The followings are the two types of maintenance which will be briefly discussed:

- (1) Preventive maintenance\_\_\_prevent CNC from getting into trouble.
- (2) Trouble shooting\_\_\_ try to correct the problem as soon as possible.

#### 12.2 PREVENTIVE MAINTENANCE

The best way to keep CNC operating properly at all times is to do preventive maintenance regularly. Some of the preventive maintenance can be done by the users on routine basis, while the others must be periodically carried out by persons trained by HUST specialists. The followings are brief discussions of preventive maintenance.

##### 1. Routine Maintenance by Users

The CNC controller is normally installed with the machine in a machine shop where the metal cuttings and the oiled dusts are the main pollutants. For this reason, every efforts should be taken to prevent these pollutants from getting into the controller. Dust cover should be put over the controller when not in use. Air filter must be checked daily and cleaned regularly to prevent filter from possible clogging, which can cause damages due to insufficient ventilation.

The items of routine check, as listed in Table 12-1, must be done regularly. The cleaning period in the table is only a guide, it may be changed according to the actual environment.

Table 12-1 ROUTINE CHECKS BY USERS

NO	ITEM	PERIOD	EXPLANATION
1	Clean around CNC	Every week	Surface and the surroundings of the CNC cabinet.
2	Clean the filter	Every week	Clean the filter with compressed air

3 Clean cooling fan Every month or wash it with neutral cleaner  
 opeates Check if the cooling fan motor  
 motor properly

2. PERIODICAL CHECK BY PERSONS TRAINED BY HUST SPECIALISTS

The items of periodical check by persons specially trained by HUST are listed in Table 12-2. HUST or its agent provides training service to the maintenance personnel of our customers.

Table 12-2 PERIODICAL CHECK BY PERSONS SPECIALLY TRAINED BY HUST

NO	ITEM	EXPLANATION
1	Clean the device	Clean the filter and NC device.
2	Check wires and connections proper inside the cabinet	Check for loosen screws & nuts, contact resistance of terminal and switch, Check contacts for burns.
3	Confirm the setting voltage	
4	Confirm feed speed frequency	
5	Confirm servo motor drive current circuit waveform and electric current value	Make sure the value of electric within the allowable range.
6	Confirm output waveform of pulse pulse encoder	Make sure the output waveform of encoder within the specs.
7	Check servo motor	
8	Confirm movement in MAMUAL mode	
9	Confirm movement in AUTO mode	
10	Over-voltage test voltage to	Intentionally raise the power the limits and operate the machine. check various functions and electronic parts for proper working order.
11	Vibration test contact and	When CNC is working, check poor or welding of printed circuit board terminals thru vibration.
12	Temperature test CNC limit.	Check the semiconductor by raising cabinet temperature to the upper
13	Continuous operation test working	Confirm that the CNC controller is normally when subject to continuous operation using real commands.

NOTE: The items to be checked vary with the controller, parts and servo

motor used. This table is only a guide.

### 12.3 TIMELY COMMUNICATION AND PROPER DESCRIPTION

When CNC fails, the user (maintenance personnel) should attempt to repair it in order to save the precious down time. If it's beyond his ability to correct the problem, please call HUST (or NC manufacturer) as soon as possible with detailed descriptions of the problem.

The CNC problems are normally in related to mechanical aspects, such as incorrect tool movement, cutting speed, cutting length, auxiliary functions, etc.. The proper description of the problem as to how, what and when it occurred will, most of the times, help HUST (or CNC manufacturer) get the problem resolved over the phone. If shop visit is required, HUST maintenance specialist will bring the appropriate parts to fix the problem on the first visit. The sole purpose of the detailed problem description is nothing but minimizing the machine down time.

### TROUBLE SHOOTING

#### 1. GENERAL

##### (1) Check the type of problem:

- \* What model is the controller?
- \* What error message is displayed on the MDI/CRT?
- \* Is there a positioning error? If so, on which axis and by what amount?
- \* Is there a tool path error? If so, by what amount?
- \* Is the speed abnormal?
- \* Is the problem occurred in an auxiliary function?

##### (2) Check the frequency of occurrence:

- \* When did the problem occur? What is its frequency?  
(Was another machine also being operated when it occurred?)
- \* What is the occurring frequency on the same workpieces?
- \* Which program is it and what is the block number with the problem?
- \* Is problem related to a specific mode?
- \* Is the problem related to tool replacement?
- \* Is the problem related to the feedrate?

##### (3) For recurrent problems:

- \* Go through and check the program tape where the error occurs repeatedly.
- \* Check the numerical value stored in the CNC memory with the one from program display.
- \* Is the problem due to an external cause?
- \* Check the reponse to override (decrease or increase the override amount)  
after distribution.
- \* Obtain the details of the problem from machine operator.

#### 2. Checking input voltage, peripheral conditions, operation, programming, drivers, machine and interface control.

##### (1) Check the input voltage:

- \* Are there fluctuations in the input voltage?
- \* Is front or rear door left open (door interlock)?

- \* Is there any other driver using large amounts of current?
  - \* Is there an electric discharge machine or welding machine nearby?
- (2) Check the peripheral conditions:
- \* What is the temperature of the controller?
  - \* Did the temperature change? Was it excessive?
  - \* Is the filter dirty?
  - \* Is the tape reader dirty?
  - \* Is there any oil or cutting fluid getting inside?
  - \* Are there any vibrations?
  - \* Is the unit under the direct sunshine?
- (3) Check for any external causes:
- \* Has the machine recently been repaired or adjusted?
  - \* Has the magnetic cabinet recently been repaired or adjusted?
  - \* Has the CNC unit recently been repaired or adjusted?
  - \* Is there a source of noise nearby?
- (Example: Cranes, High-frequency machines, Electric discharge machines)
- \* Has a new machine been mounted nearby?
  - \* Is there any other CNC with the same problem?
  - \* Has the user adjusted the CNC himself?
  - \* Has the same problem occurred before?
- (4) Check operation procedures and its related
- \* Is the operator familiar with the program?
  - \* Did the program finish too early or was it interrupted?
  - \* Is the tool compensation value correctly set?
  - \* Is the tool compensation value being changed, Is it done correctly?
  - \* Does the machine change to another mode of operation?
  - \* Is the optional block skip function used correctly?
- (5) Check the program
- \* Is the program new?
  - \* Was the program created according to the OPERATOR'S MANUAL?
  - \* Are address codes correct?
  - \* Does the problem occur in any specific block?
  - \* Are the speed and lead screw values correctly set for thread cutting?
- \* Is there enough space at the beginning and end of thread cutting?
  - \* Does the problem occur in the subprogram?
- (6) Check for changes in operation
- \* Has any change or adjustment been made in the operation procedure?
  - \* Has a fuse been blown?
  - \* Is the CNC in the emergency stop status?
  - \* Is the machine tool ready?
  - \* Is the CNC in the alarm status?
  - \* Is the MODE SELECT switch set correctly?
  - \* Is the override switch set to the zero position?
  - \* Is the CNC in the machine lock status?
  - \* Is the feed hold switch pushed?
- (7) Check the machine itself?
- \* Is the machine properly installed?
  - \* Does vibration occur during operation?
  - \* Is the tool tip normal?
  - \* Is there any offset due to tool exchange?
  - \* Is there sufficient backlash compensation?
  - \* Are there distortions in any part of the machine due to

- temperature changes?
- \* Is the workpiece measurement made at a constant temperature?(1 meter of steel changes 10u in length for a temperature change of 1 degree)
- \* Are the cables normal (bent, broken or damaged)?
- \* Are the signal lines and power lines separated?
- (8) Check the interface control
  - \* Are power lines and CNC cables mounted separately?
  - \* Is the shield normal? Is a spark suppressor attached to the relay, solenoid, and motor?

### 3. CNC SYSTEM CHECK

- (1) Check control unit external conditions
  - \* Is there any damage to the cabinet?
  - \* Is the MDI/CRT unit normal?
  - \* Is the filter clean?
  - \* Is the machine being operated with the cabinet door open?
- (2) Check inside the control unit
  - \* Is there any metal accumulation inside the cabinet?
  - \* Is the cooling fan motor running normally?
  - \* Is there any sign of corrosion?
- (3) Check the power unit
  - \* Is the unit correctly connected?
  - \* Are all fuses OK?
  - \* Is the circuit breaker normal?
  - \* Is the voltage within the allowable range?
  - \* Are the shield and cable grounded correctly?
  - \* Is the wiring path OK?
  - \* Are all terminals fully tightened?
- (4) The grounding
  - \* Is the grounding connection OK?
  - \* Is the shield ground OK?
- (5) Check all cables
  - \* Are cable connectors fully pushed in?
  - \* Are there any abnormalities inside and outside the cables?
  - \* Are there any scratches, bends or breaks?
- (6) Check printed circuit boards
  - \* Are all PCBs mounted properly?
  - \* Is the plug connector OK?
  - \* Are physical conditions normal (no distortions, etc.)?
- (7) Check the MDI/CRT unit:
  - \* Do the push buttons operate normally?
  - \* Is the tape cable normal?

When you have problem with your CNC controller, the first thing you do is to try to fully understand the problem. Once you have fully analyzed the problem, you can narrow it to a small area. If the problem is related to the part program, you should be able to get it fixed very easily. If it's related to the hardware, the best way is to replace the printed circuit board. Trying to pin-point the worn part by using oscilloscope or multimeter is time consuming and no longer practical. If the problem is so complicate that it's beyond the ability of your maintenance personnel, then fill in all facts as much detailed as you can in the PROBLEM SHEET

(next page) and send it to us by FAX, we will try to response to you as soon as possible.