

SYNTEC

Mill Controller Manual

By: SYNTEC

Date: 2010/4/20

Version: 8.3

Version edit record

No.	Modify Content	date	editor	New Vision
01				
02	Modify teach-in operation	2007/03/20	Jerry_Lai	V8.1
03	Modify auto center and auto tool	2008/08/12	Samuel _Chen	V8.2
04	Modify graph and sync. with Chinese version	2010/04/20	Samuel _Chen	V8.3

Contents

1	CNC MILLING CONTROLLER INTERFACE	1
1.1	CNC System Configuration.....	1
1.2	Screen Sections.....	2
1.3	Main Menu Selections	3
1.3.1	F1 : Position.....	4
1.3.1.1	F1 : Coord. Switch	5
1.3.1.2	F2 : 1/2 Coord.	5
1.3.1.3	F3 : Clear Coord.....	5
1.3.1.4	F4 : Clear All Relative.....	6
1.3.1.5	F5 : Workpiece Coord.	6
1.3.1.5.1	F1 : Coord. Latched	7
1.3.1.5.2	F2 : Rel. Latched	7
1.3.1.5.3	F3 : Aux. Latched	7
1.3.1.5.4	F5 : Middle Point Func.	7
1.3.1.5.5	F6 : Tool Tip Measure	7
1.3.2	F2 : Program	8
1.3.2.1	F1 : Insert Cycle	9
1.3.2.2	F2 : Delete Line.....	9
1.3.2.3	F3 : Edit Cycle	10
1.3.2.4	F5 : EDIT	11
1.3.2.4.1	EDIT sub function "F1 : Search"	12
1.3.2.4.2	EDIT sub function "F2 : Replace"	13
1.3.2.4.3	EDIT sub function "F3" : Go To Line"	14
1.3.2.4.4	EDIT sub function "F4" : copy line"	14
1.3.2.4.5	EDIT sub function "F5" : Insert line"	14
1.3.2.5	F6 : Teach	15
1.3.2.5.1	F1 : Rapid Teach	15
1.3.2.5.2	F2 : Line Teach.....	15
1.3.2.5.3	F3 : Arc Teach.....	15
1.3.2.5.4	F4 : Cancel Middle	16
1.3.2.5.5	F5 : Radius Teach.....	16
1.3.2.5.6	F6 : Delete Line	16
1.3.2.5.7	F7 : Coord Switch.....	16
1.3.2.5.8	F8 : Point Teach	16

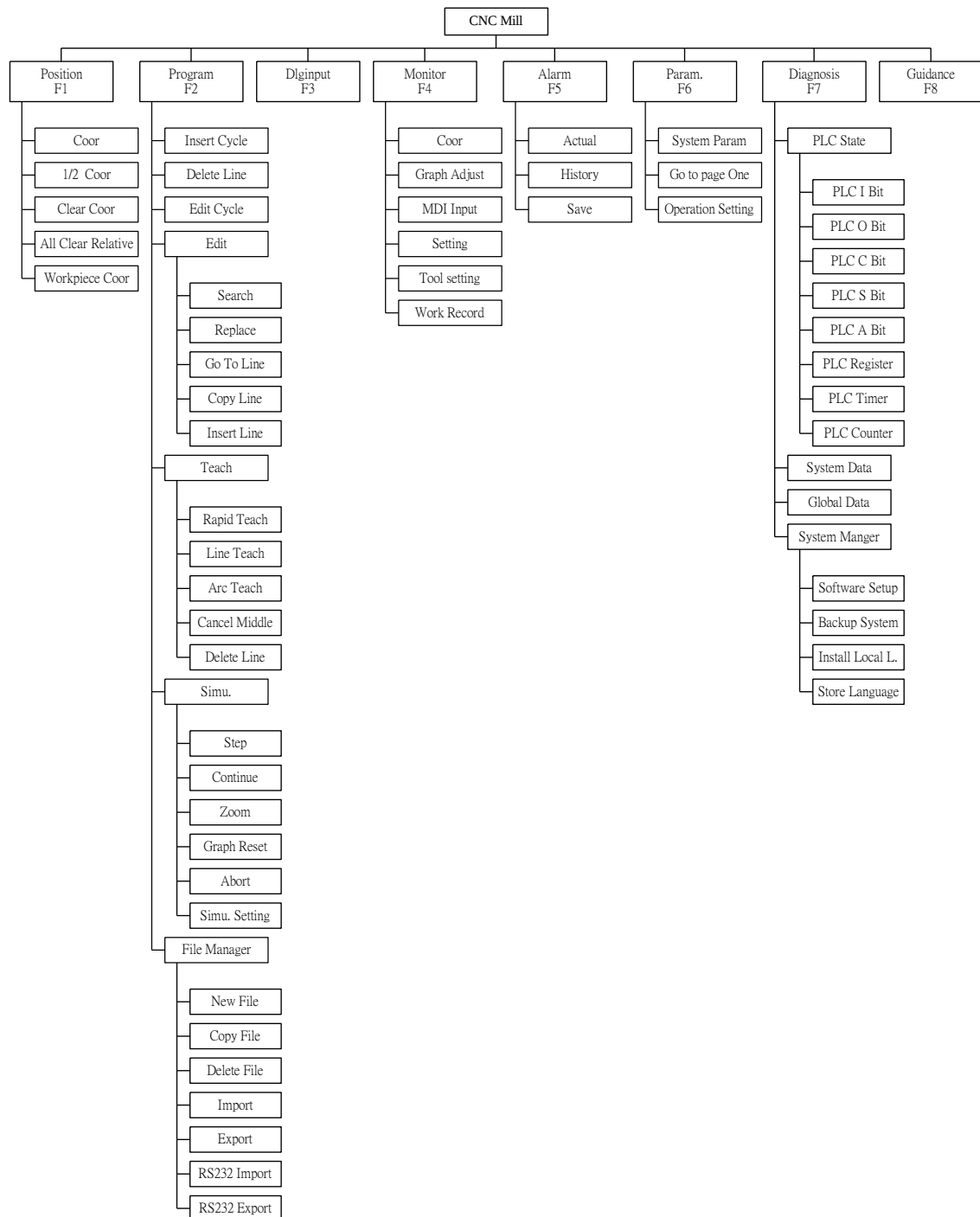
1.3.2.6	F7 : Simulation.....	17
1.3.2.6.1	F1 : STEP	17
1.3.2.6.2	F2 : Continue	17
1.3.2.6.3	F3 : Zoom.....	18
1.3.2.6.4	F5 : Graph reset	18
1.3.2.6.5	F6 : Abort	18
1.3.2.6.6	F8 : Simu. Setting	18
1.3.2.7	F8 : File Manager	20
1.3.2.7.1	F1 : New File	21
1.3.2.7.2	F2 : Copy File.....	21
1.3.2.7.3	F3 : Delete File	21
1.3.2.7.4	F4 : Import	22
1.3.2.7.5	F5 : Export	23
1.3.2.7.6	F6 : RS232 Import	24
1.3.2.7.7	F7 RS232 Export.....	25
1.3.2.7.8	RS232 communication parameter setting	26
1.3.3	F3 : DlgInput (ONLY for 940M).....	27
1.3.4	F4 : Monitor.....	27
1.3.4.1	F1 : Coord.....	27
1.3.4.2	F2 : Graph Adjust.....	28
1.3.4.2.1	F1 : Zoom.....	28
1.3.4.2.2	F2 : Graph Reset.....	28
1.3.4.2.3	F5 : Simu Setting.....	28
1.3.4.3	F3 : MDI Input	29
1.3.4.4	F4 : SETTING.....	30
1.3.4.5	F5 : Tool Setting.....	31
1.3.4.6	F8 : Work Record.....	32
1.3.5	F5 : Alarm.....	33
1.3.5.1	F1 : Actual.....	34
1.3.5.2	F2 : History	34
1.3.5.3	F5: Save	34
1.3.6	F6 : Parameter.....	35
1.3.7	F7 : Diagnosis	36
1.3.7.1	F1 : PLC State	37
1.3.7.2	F2 : System Data	38
1.3.7.3	F3 : Global Variable	39
1.3.7.4	F4 : Coord Variable	40
1.3.7.5	F8 : System Manager.....	41

1.3.8	F8 : Guidance	42
2	MACHINE OPERATION PANEL	43
2.1	2 nd machine operation panel	43
2.1.1	POWER ON	43
2.1.2	POWER OFF	43
2.1.3	Emergency STOP.....	43
2.1.4	Home mode and Home function.....	43
2.1.5	Continues JOG (Rapid JOG)	43
2.1.6	Incremental JOG	44
2.1.7	MPG JOG	44
2.1.8	AUTO mode NC file execute	44
2.1.9	MDI mode single block execute	44
2.1.10	MPG Simulation	45
2.1.11	Dry Run	45
2.1.12	Single block	45
2.1.13	Option Stop.....	46
2.1.14	Option Skip.....	46
2.1.15	Spindle control.....	46
2.1.16	Working led	47
2.1.17	Working Liquid.....	47
2.1.18	Aux table backward.....	47
2.1.19	Aux table forward	47
2.2	Text key description:.....	48
3	HOW TO OPERATE SYNTEC 900ME	50
3.1	Manual function (JOG, INC_JOG, MPG)	51
3.2	HOME	52
3.3	Open a file (EDIT / FLOPPY / RS232)	53
3.4	Tool setting (G40/G41/G42, G43/G44/G49)	54
3.5	Tool Length measurement (G43/G44/G49)	55
3.6	Setting the Workpiece origin offset value (G54...G59)	56

3.7	Manual Data Input (MDI)	57
3.8	Assigned an executing NC file (AUTO).....	58
3.9	Graphic Simulation	59
3.10	How to check NC file in SYNTEC controller.....	60
3.11	Auto Center	61
3.11.1	Manual Center.....	61
3.11.2	Auto center operation	63
3.12	Z-axis Auto Tool.....	65
3.13	RS232 FUNCTION	70
3.14	DNC Function	72
3.15	The software of SYNTEC controller replace the software of DNC to execute RS232 function.....	74

1 CNC Milling Controller Interface

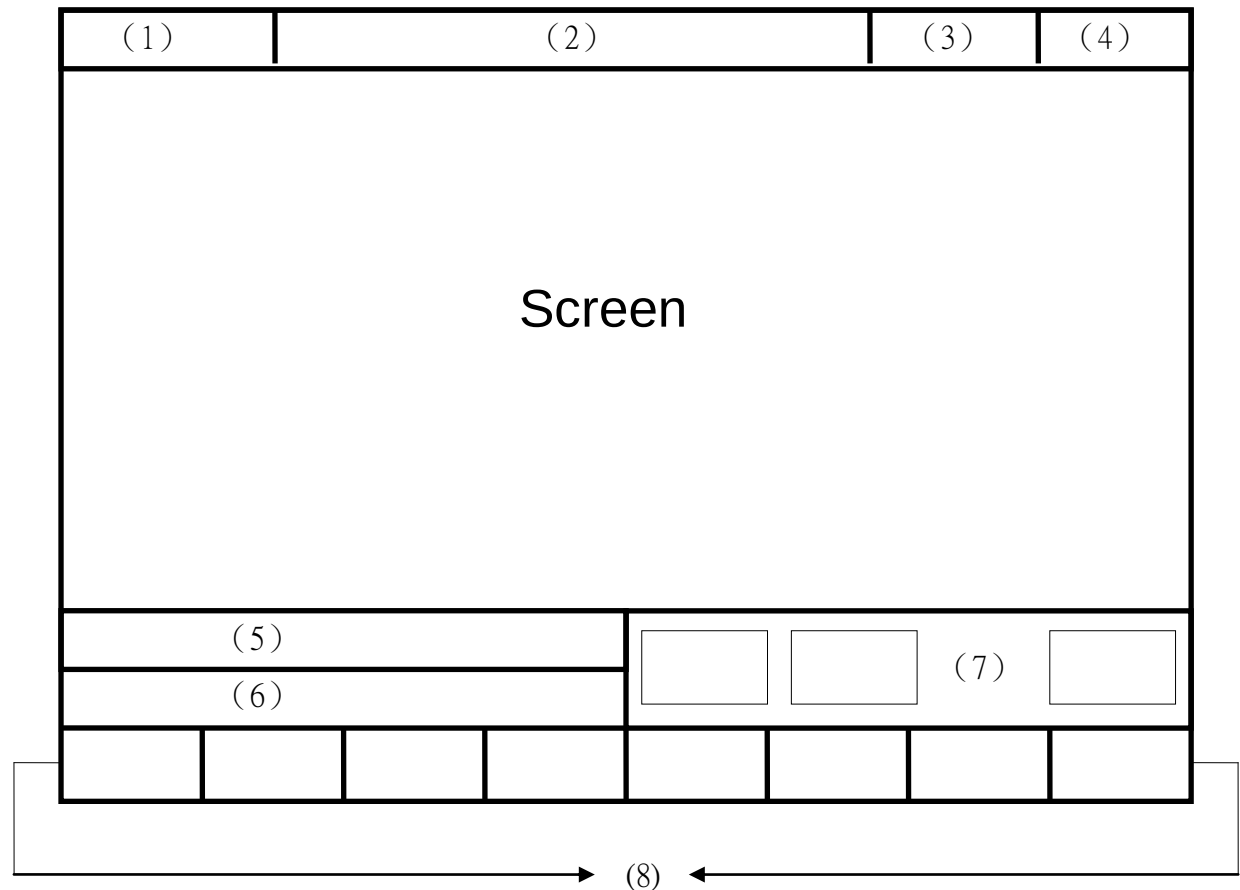
1.1 CNC System Configuration



1.2 Screen Sections

The screen of controller is shown as followings:

Start

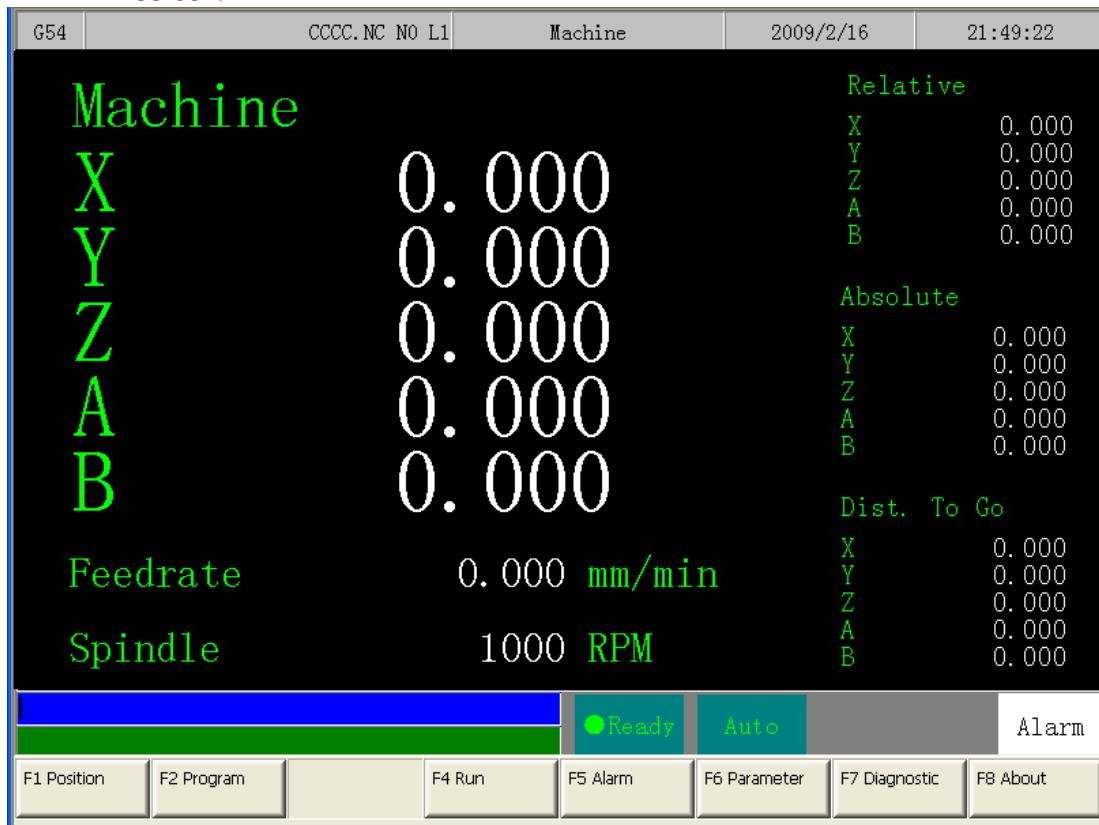


Meanings For Fields on the Display:

- (1) Program Number
- (2) Title
- (3) Time
- (4) Date
- (5) Data Input
- (6) Hint
- (7) Status
- (8) Function Key Switch

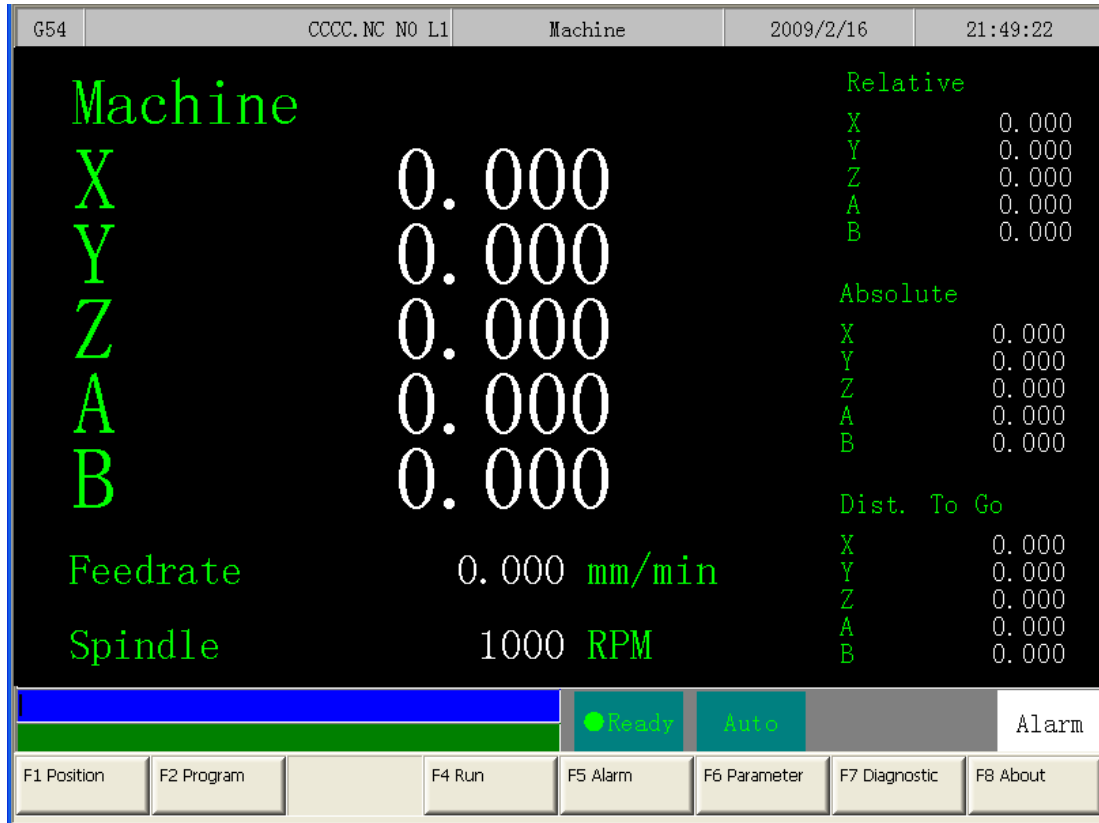
1.3 Main Menu Selections

The following diagram is the main menu selections for SNC Mill controller. To operate SNC Milling controller, users simply make the selections by pressing function keys, F1~F8 located on the bottom of the screen.



1.3.1 F1 : Position

This selection displays coordinate settings of current position. It can also be used to reset the position of relative coordinate. Pressing function key, F1, under the main menu to enter this selection (Note : This is the first screen when the system is booted up)



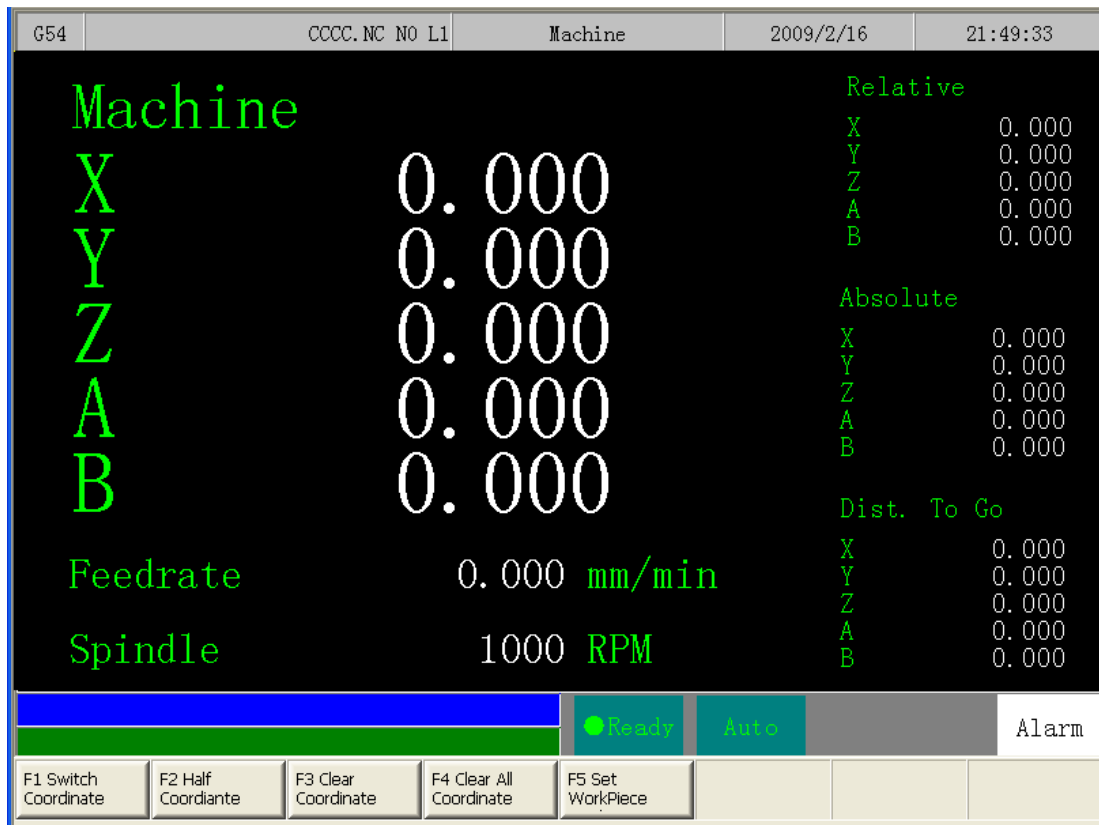
Meaning of fields on the display-----

- X : X axis coordinate.
- Y : Y axis coordinate.
- Z : Z axis coordinate.
- Feedrate : Feedrate of cutting tool at each machining, mm per minute (mm/min).
- Spindle : RPM of spindle speed.
- Machine(Relative coordinate of working platform)
- The current position of cutting tool relative to working platform is shown as machine coordinate on the display.
- Relative
- The current position of cutting tool relate to the previous location.
- Absolute(Programming Coordinate)
- The current position of the origin of user defined coordinate is shown as an absolute position on the display.
- Distance To Go : The distance of the cutting tool that need to move to the next position show on position(+) and negative(-) direction.

Function key selections :

1.3.1.1 F1 : Coor. Switch

- Function : Switch Coordinate Display ◦
- Operation : Under the Position submenu, whenever users press F1 key, the values and coordinate on the left corner of the display will toggle among the four different coordinates with bigger fonts as shown in the following figure:



1.3.1.2 F2 : 1/2 Coor.

- Function: Set the center point of work piece as coordinate origin.
- Operation: Under the Position submenu, when the message line shows “X Input”(or “Y Input” or “Z Input”), press “F2 1/2 Coordinating” and the origin of the coordinate will move to the center point of work piece.

1.3.1.3 F3 : Clear Coor.

- Function : Reset the value of X(or Y or Z) axis relative coordinate to zero. (No effect on other axes)
- Operation : Under the Position submenu, when the message line shows “X Input”(or “Y Input” or “Z Input”), press F3 to reset the value of X(or Y or Z) axis relative coordinate to zero.

1.3.1.4 F4 : Clear All Relative

- Function : Reset XYZ relative coordinate to zero.(No effect on other coordinates)
- Operation : Under the Position submenu, pressing “F4” will reset XYZ relative coordinate to zero.

1.3.1.5 F5 : Workpiece Coord.

- Function : Relative to machine coordinate setting for G54~G59.
- Operation : Under the Position submenu, press "F5" key and the following screen will show up. The users can begin to set the auto machine coordinate settings of G54~G59. (The system needs to be in Ready Status)
 1. “External Shift” : operator can set the all coordinate G54..G59 at the same time .
 2. CNC default G54 ,if user don't set any G54..G59 in the NC file

G54		110.NC N0 L1		Position		12:27:40		2009/02/14		
External Shift		G54		G55		Machine				
X	0.000	X	10.000	X	0.000	X	0.000			
Y	0.000	Y	0.000	Y	0.000	Y	0.000			
Z	0.000	Z	0.000	Z	0.000	Z	0.000			
A	0.000	A	0.000	A	0.000	A	0.000			
B	0.000	B	0.000	B	0.000	B	0.000			
G56		G57		G58		Relative				
X	0.000	X	0.000	X	0.000	X	0.000			
Y	0.000	Y	0.000	Y	0.000	Y	0.000			
Z	0.000	Z	0.000	Z	0.000	Z	0.000			
A	0.000	A	0.000	A	0.000	A	0.000			
B	0.000	B	0.000	B	0.000	B	0.000			
						Aux. Coordinate				
						X	0.000			
						Y	0.000			
						Z	0.000			
					READY		AUTO		ALARM	
Coor. Latched		Rel. Latched		Aux. Latched		Middle Func.		Tool Tip Measure		

1.3.1.5.1 F1 : Coord. Latched

Set the current machine coordinate to the input box

1.3.1.5.2 F2 : Rel. Latched

Set the current relative coordinate to the input box

1.3.1.5.3 F3 : Aux. Latched

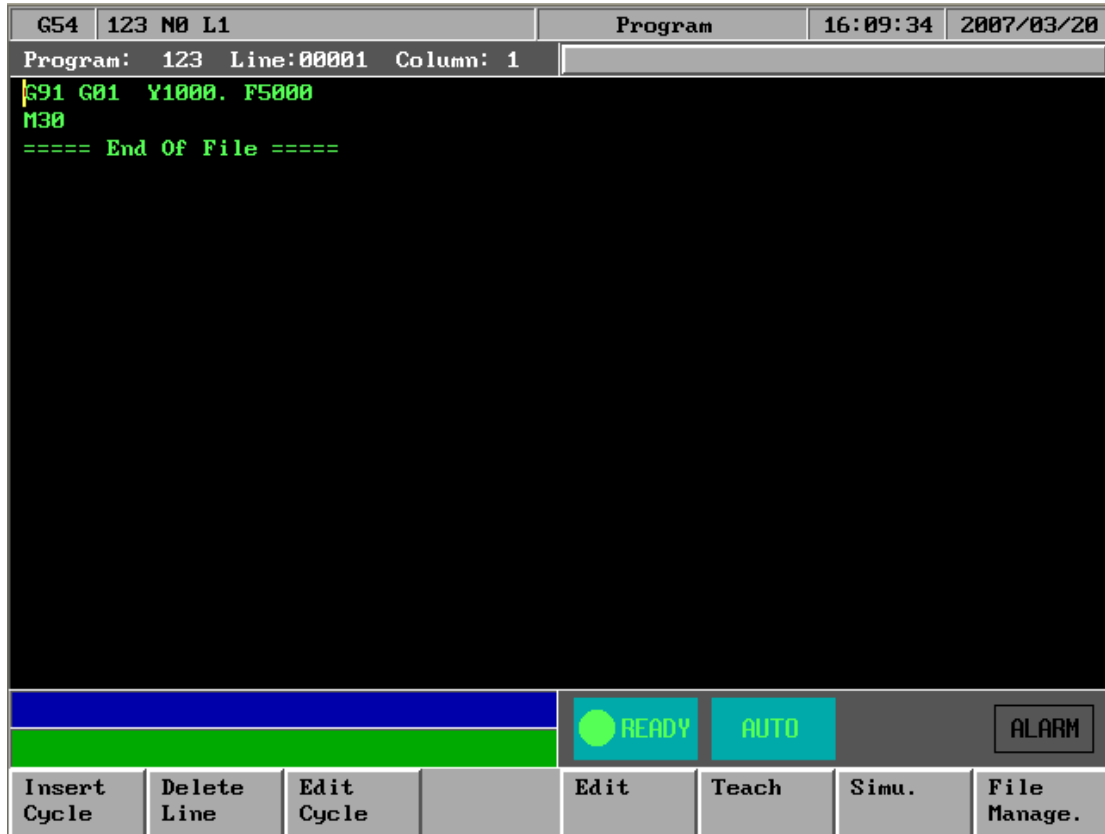
Set the current relative coordinate to the input box

1.3.1.5.4 F5 : Middle Point Func.

1.3.1.5.5 F6 : Tool Tip Measure

1.3.2 F2 : Program

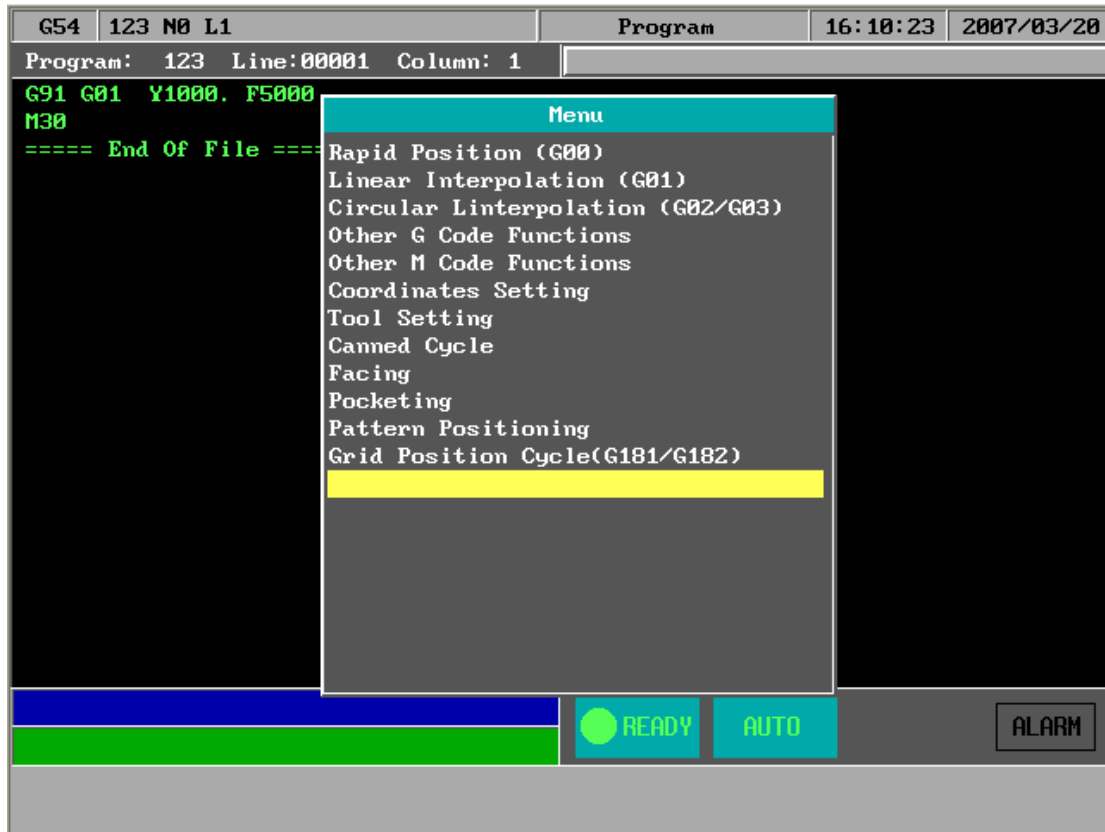
This selection provides users with program file management and editing. With a full screen editor, users can use arrow keys (↑、↓、←、→) to move the cursor to anywhere on the screen for editing purpose. Press F2 under the main menu to enter this selection. The full screen editor is shown as follows:



Program Sub menu Key Selections:

1.3.2.1 F1 : Insert Cycle

- Function : Insert a block or cycle by conversation
- Operation : Under Program submenu, press F1 to insert a line or cycle before cursor position.(See Graphic Input Interface(900ME) User Guide)

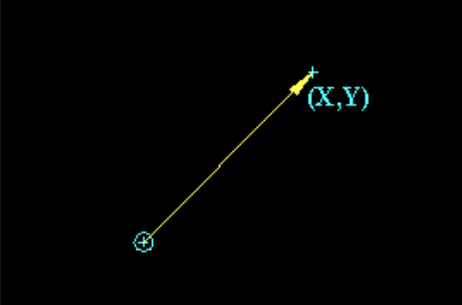


1.3.2.2 F2 : Delete Line

- Function : Delete a line at cursor position.
- operation: Under Program submenu, press F2 to delete a line where the cursor is located.

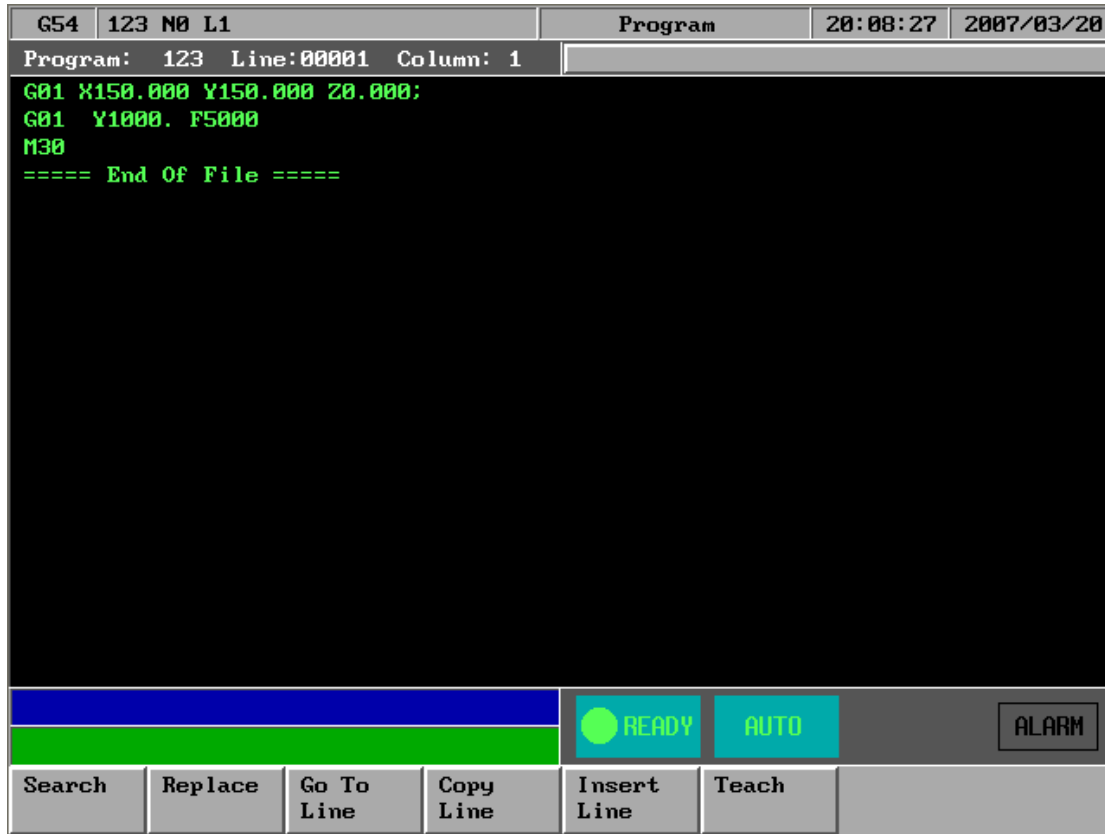
1.3.2.3 F3 : Edit Cycle

- Function : Edit an old block or cycle by conversation input
- Operation: Under Program submenu, press F3 to edit a line where the cursor is located.

G54	123 N0 L1	Program	17:26:00	2007/03/20
Prog	Cycle			
G01 %	LineNo:1	Linear Interpolation(G01)		
G01				
M30				
=====				
		End Point		
		X Axis	150.000	
		Y Axis	150.000	
		Z Axis	0.000	
		Chamfer or radius of corner		
		Chamfer(C)		
		Radius(R)		
Speed(S)				
Feedrate(F)				
End point X				
OK				
		READY	AUTO	ALARM

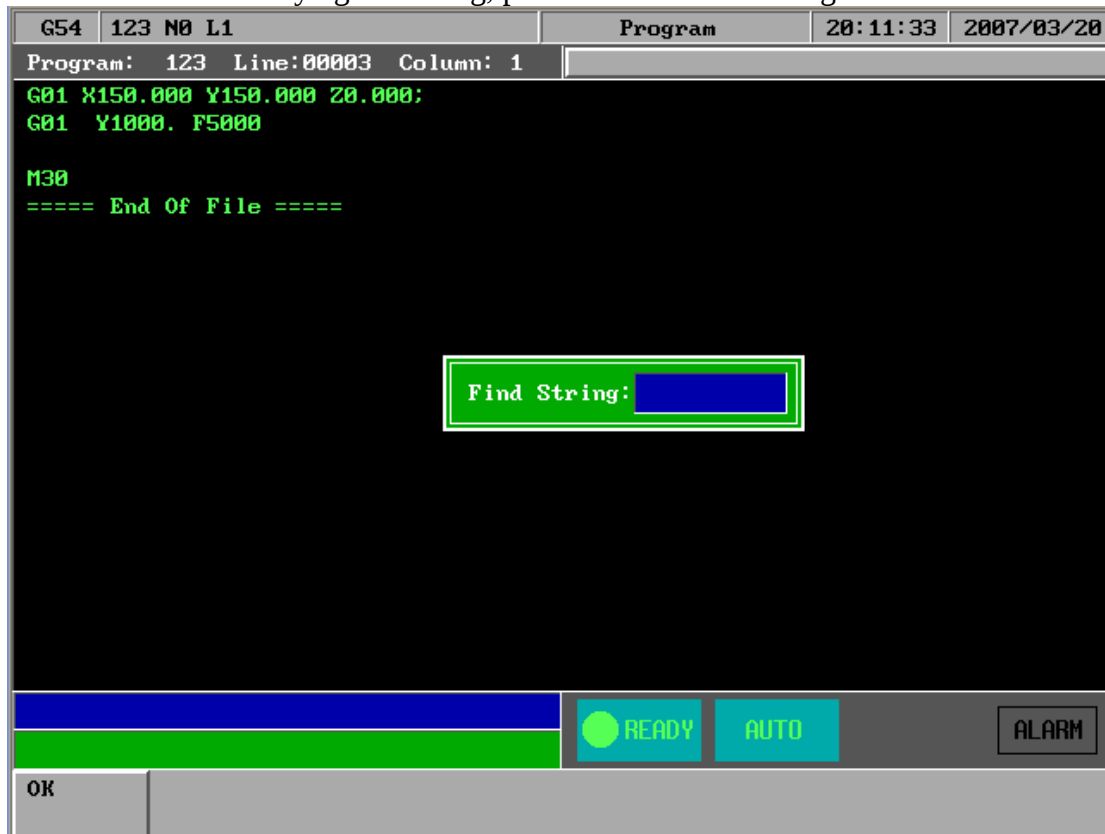
1.3.2.4 F5 : EDIT

- Function : Edit sub function “ Search “ “Replace” “Goto line” “Copy line” ”Insert line”
- Operation : User can use this menu for more edit sub function



1.3.2.4.1 EDIT sub function “F1 : Search”

- Function : Search String .
- Operation : Under Program submenu, press F5”EDIT” and then F1 “Search” to search string. An dialog box will pop up asking users to input a string as shown in the following figure. After keying in a string, press F1 to start searching.



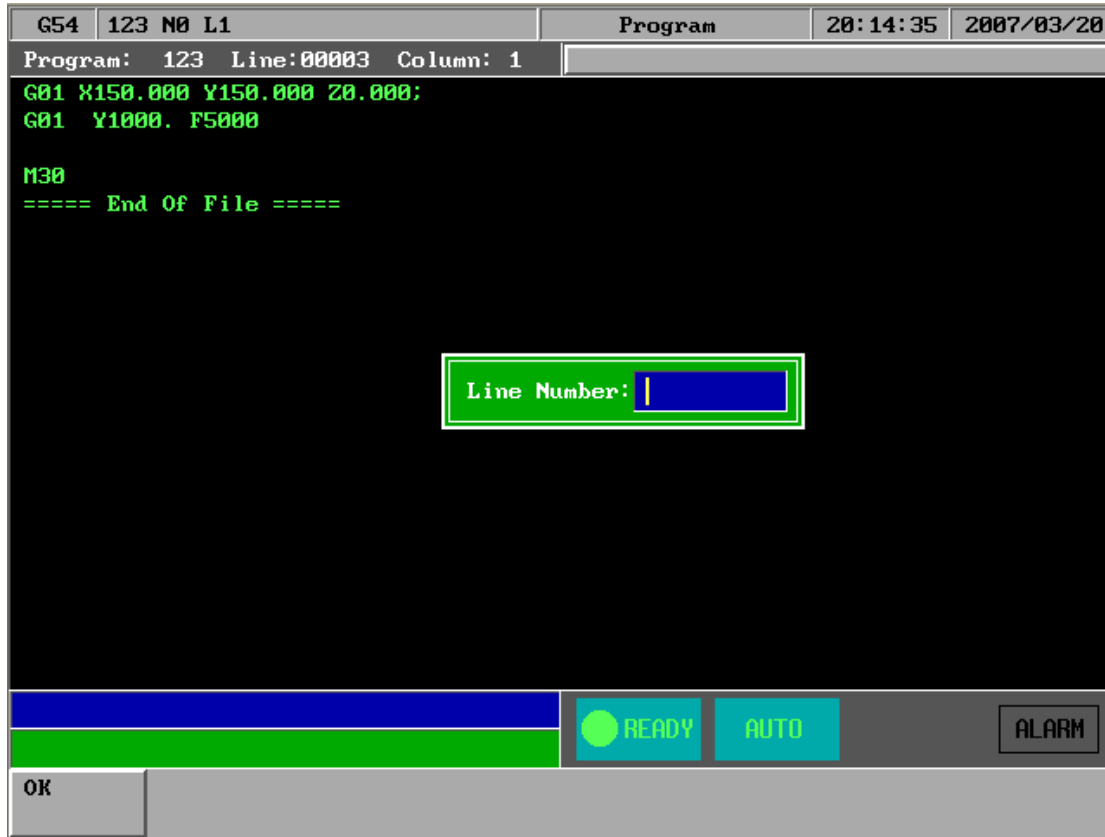
1.3.2.4.2 EDIT sub function “F2 : Replace”

- Function : Replace String.
- Operation : Under Program submenu, press F5 “EDIT” and then F2”Replace” to replace string. An dialog box will pop up asking users to input the replacing string and the new string as shown in the following figure. After keying in a string, press F1 to start replacing.

G54	123 N0 L1	Program	20:13:43	2007/03/20
Program: 123 Line:00003 Column: 1				
<pre>G01 X150.000 Y150.000 Z0.000; G01 Y1000. F5000 M30 ===== End Of File =====</pre>				
Find What: Replace With:				
		● READY	AUTO	ALARM
Replace	Skip	Cancel		

1.3.2.4.3 EDIT sub function “F3” : Go To Line”

- Function : Go to a line number
- Operation : Under Program submenu, press F4”EDIT” and then F3”GOTO line” to go to the line number. A dialog box will pop up asking users to input a line number. After keying in a number, press F1 to go to the desired line.



1.3.2.4.4 EDIT sub function “F4”: copy line”

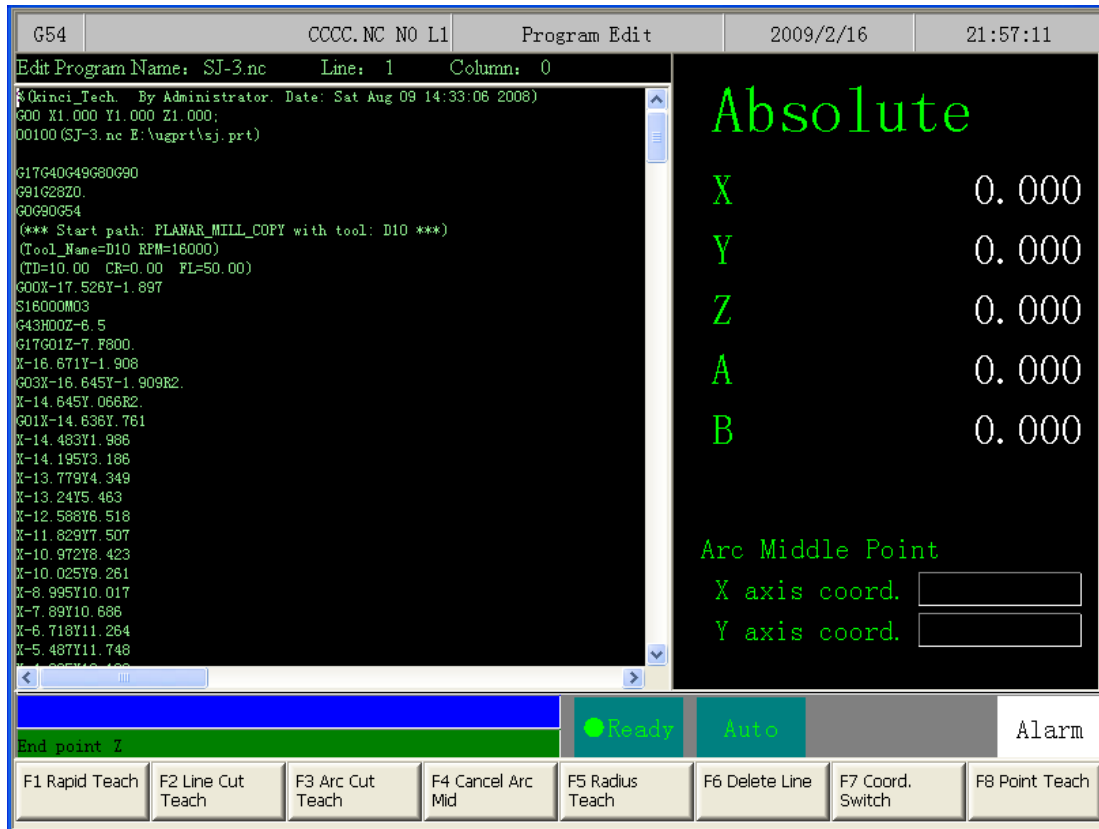
- Function : Copy a line from current cursor to next line
- Operation : Under Program submenu, press F5 “EDIT” and then F4”Copy line” to go to the next line.

1.3.2.4.5 EDIT sub function “F5”: Insert line”

- Function : Insert a space line above current cursor line
- Operation : Under Program submenu, press F5 “EDIT” and then F5”Insert line” to Insert a new space line

1.3.2.5 F6 : Teach

- Function: Teach present absolute coordinate to NC files
- Operation : Under Program submenu, press F6”Teach”



Teach sub Key Selections :

1.3.2.5.1 F1 : Rapid Teach

- Function: Add “G00” code to NC files, G00 to current absolute coordinate,

1.3.2.5.2 F2 : Line Teach

- Function: Add “G01” code to NC files, G01 to current absolute coordinate,

1.3.2.5.3 F3 : Arc Teach

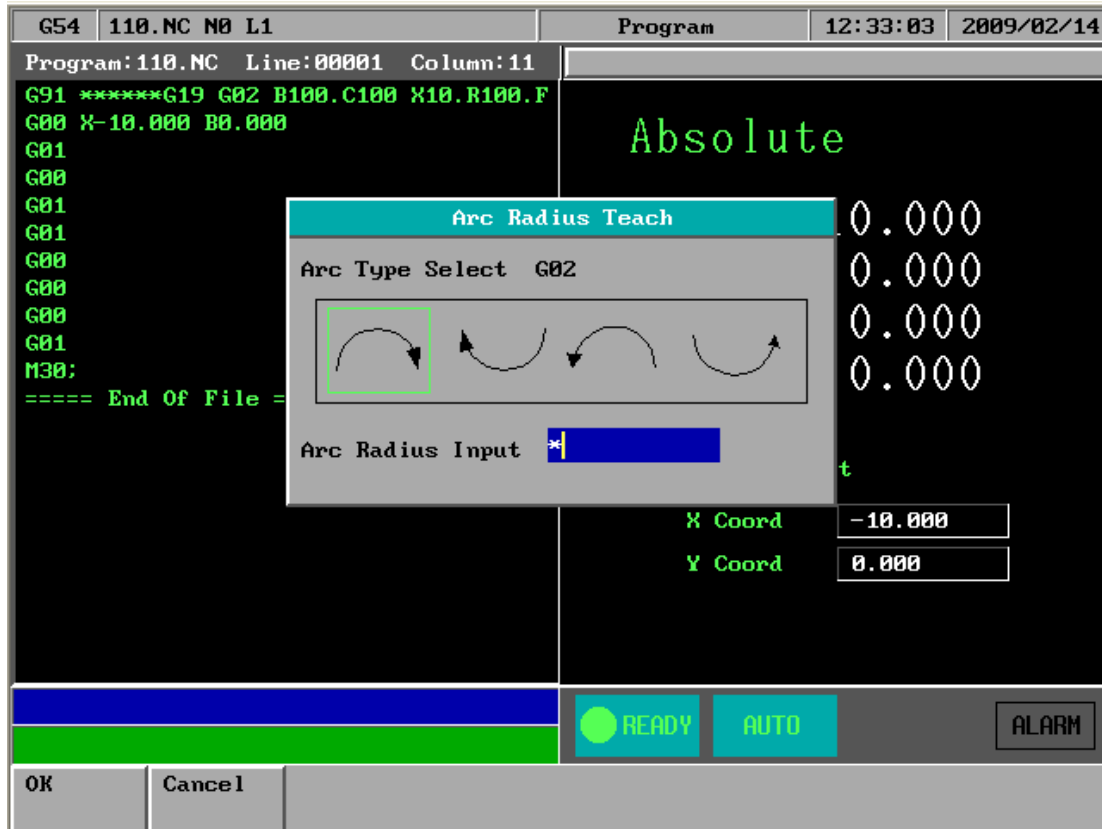
- Function: Add “G02” or “G03” code to NC files
 1. 1st time press this key “arc teach “ ,CNC auto put current value to Arc middle point
 2. 2nd time press this key “arc teach “ ,CNC auto calculate G02 or G03 ,and filled the complete code to NC files

1.3.2.5.4 F4 : Cancel Middle

- Function: When arc teach ,user can use this key to abort middle point teach .

1.3.2.5.5 F5 : Radius Teach

- Function: Add “G02” or “G03” code to NC files by G_code menu and Arc Radius



1.3.2.5.6 F6 : Delete Line

- Function: When user use Teach function , user can use this key “Delete line “ to delete line

1.3.2.5.7 F7 : Coord Switch

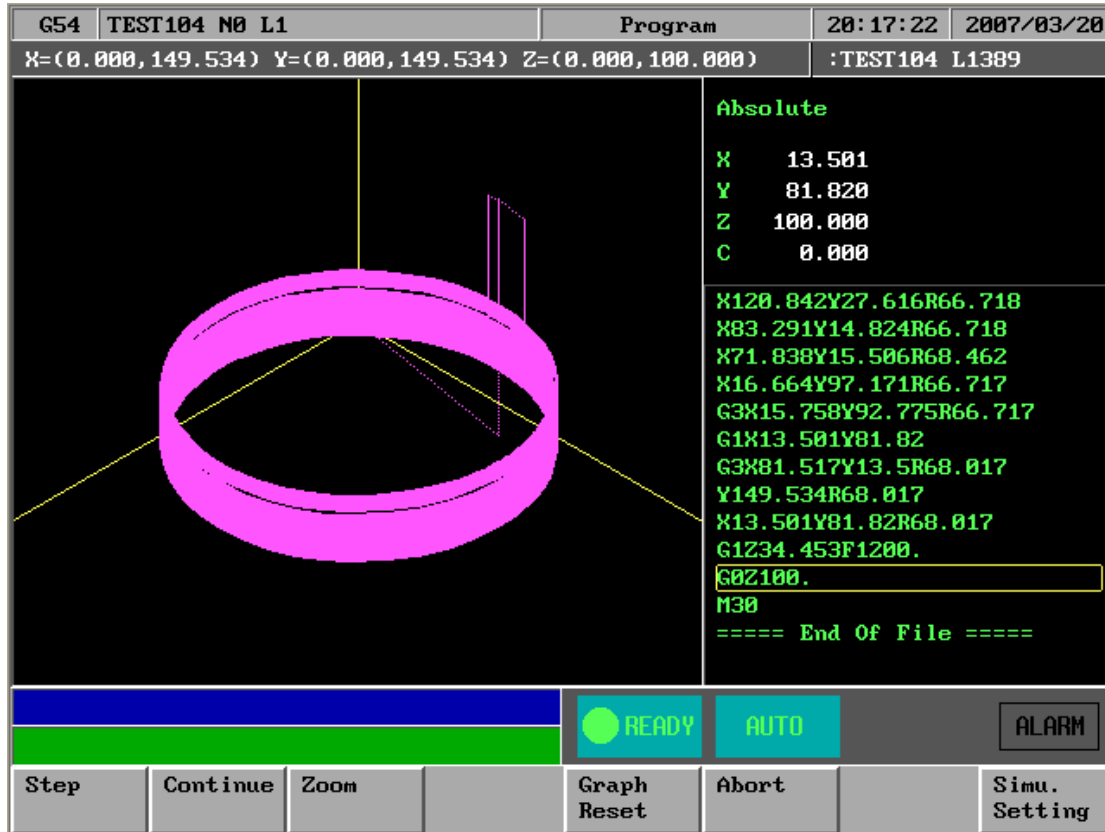
- Function: When the number of axis is more than 4-axis ,user can use this key to switch absolute coordinate of other axis.

1.3.2.5.8 F8 : Point Teach

- Function: Add current absolute coordinate to NC files.

1.3.2.6 F7 : Simulation

- Function: Simulating the workpiece program can prove the accuracy of the editing program.
- Operation : Under Program submenu, press F7



F7 “simulation” sub Key Selections :

1.3.2.6.1 F1 : STEP

- Function: To simulate NC files STEP by STEP
- Operation: Under Program submenu, press F7”Simulation” and then F1”Step” .The operator can use this function ,to check NC file STEP by STEP

1.3.2.6.2 F2 : Continue

- Function: To simulation NC file one time .
- Operation: Under Program submenu, press F7 “Simulation” and then F2”Continue” .The operator can use this function to check NC file whole picture ,when push button.

1.3.2.6.3 F3 : Zoom

- Function: To enlarge the workpiece graph.
- Operation: Under Program submenu, press F7 and then F3. The operator can use the “←”, “↑”, “→”, “↓” cursor to move the frame to the determined area. And use “PageUp” “PageDn” to Enlarge this area .

1.3.2.6.4 F5 : Graph reset

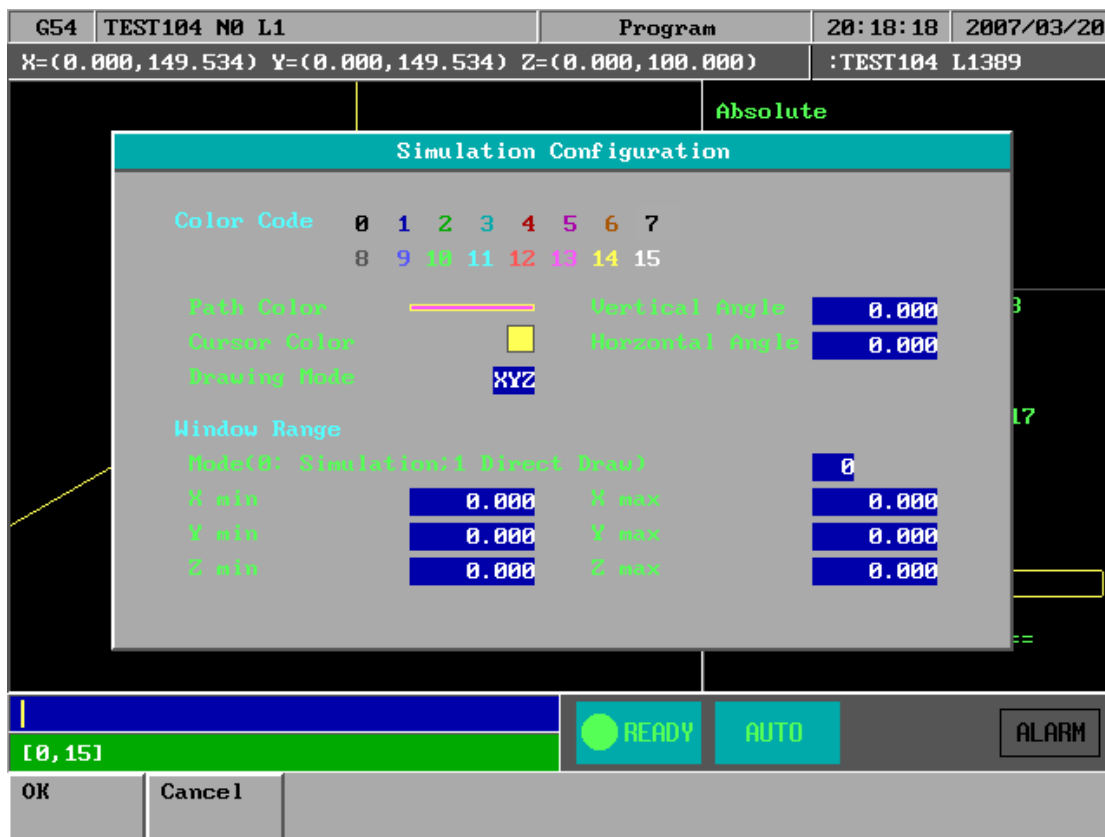
- Function: To recover the zoomed workpiece graph.

1.3.2.6.5 F6 : Abort

- Function: To abort simulation action

1.3.2.6.6 F8 : Simu. Setting

- Function: To set simulation parameter



Simulation Parameter description :

- Path color : User can select cutting path color by this parameter
- Cursor color : User can select cutting point color by this parameter
- Drawing mode : User can select simulation plane by this parameter
- Vertical / Horizontal angle : when XYZ drawing mode ,user can select 3D View angle by these 2 parameter

Window range :

Mode (0: simulation ,1: direct draw)

- 0 : When operator change his main screen to F4"Monitor" ,CNC would automatically simulation at that screen
- 1 : When operator change his main screen to F4"Monitor" ,CNC would not simulation at that screen but direct drawing the cutting cursor .

Xmin/Xmax, Ymin/Ymax ,Zmin/Zmax :

When "direct draw" mode ,operator must set draw window by these parameter ,the best way : after simulation use simulation result which is located at the top of this screen to X,Y,Z range .

1.3.2.7 F8 : File Manager

Under Program submenu, press F5 and the following diagram will show up. Users can use arrow keys(↑、↓)to select file to be edited. After pressing **【ENTER】**, content of the file will show up on the screen

G54	TEST104 N0 L1	Program	20:19:04	2007/03/20
Program storage free space 2133655552 bytes				
111	864790	01/30/07 02:23	pm	
123	28	02/07/07 01:59	pm	G01 X150.000 Y150.000 Z0.000;
92103023	4212	02/01/07 00:01	pm	M48
A0001	345	06/01/04 06:21	pm	
FLYHOH~1	31533	03/06/07 06:26	pm	
FLYHOH~2	39425	03/06/07 06:26	pm	
MDIBLOCK	1	08/14/06 02:17	pm	
00001	68	11/24/03 03:07	pm	G00X50.Y50.Z50.
01000	258	11/18/03 07:59	pm	銑床測試程式 FOR G73
01010	263	11/18/03 07:59	pm	銑床測試程式 FOR G74
03295	1053	03/03/07 06:04	pm	齒輪 標//
TEST101	37052	01/30/07 07:28	pm	00001
TEST102	32862	01/30/07 07:41	pm	00001
TEST103	32893	01/30/07 08:35	pm	00001
TEST104	30152	02/10/07 11:03	am	00001
TEST105	32862	02/13/07 08:20	pm	00001
TEST2	321346	02/28/07 02:14	pm	
WANG	379	04/14/06 01:20	pm	
RS232				DNC Program
			READY	AUTO
			ALARM	
New File	Copy File	Delete File	Import	Export
			RS232 Import	RS232 Export
				Network Import

Key Selections :

1.3.2.7.1 F1 : New File

■ Operation :

Step 1: A dialog box will prompt users with “New File ”. Type in the new file name and press **【ENTER】** .

Step 2: An empty screen shows up waiting users to type in a new program.

1.3.2.7.2 F2 : Copy File

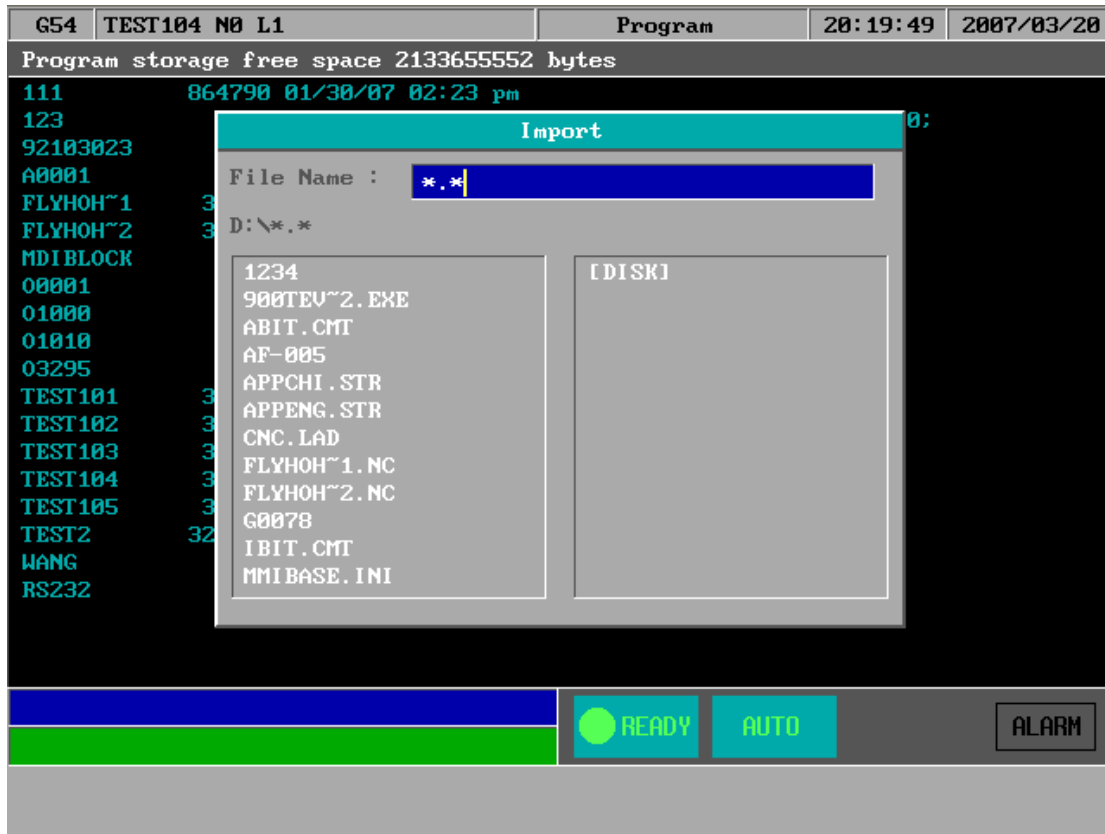
■ Operation: After pressing F2, a dialog box will prompt users to type in a file name and press **【ENTER】** . The current file is then copy to the hard disk with a different file name.

1.3.2.7.3 F3 : Delete File

■ Operation: Select a file to be delete by pressing (↑、↓) . A dialog box will pop up to confirm this operation.

1.3.2.7.4 F4 : Import

- Function : Input file from floppy.
- Operation : Insert a disk to the floppy drive and then press F4. Select a file name by pressing (↑、↓、←、→) . Press **【ENTER】** to input the file from a floppy disk

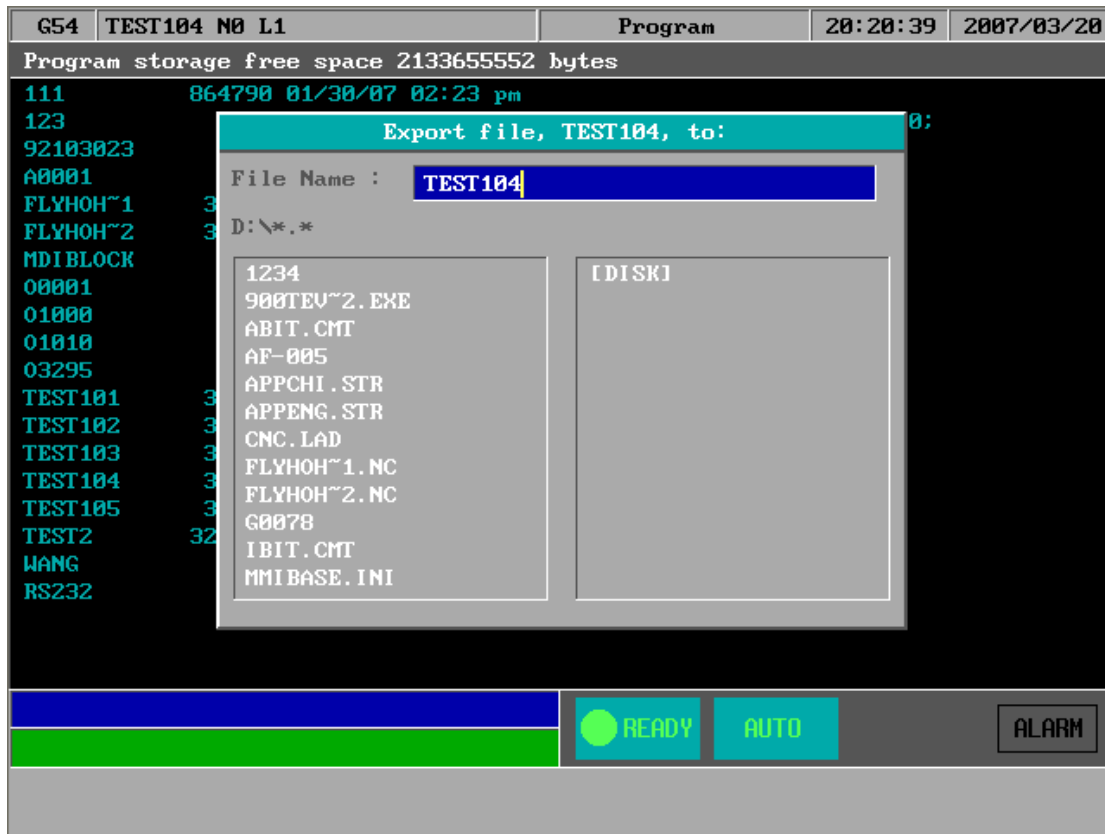


P.S.

1. floppy disk file format is ASCII code
2. SYNTEC CNC also can accept *.zip format ,when import from floppy disk ,cnc would unzip automatically
3. if NC file too big ,operator can use more than ONE floppy disk to import nc FILE , separates a big file to some floppy disks ,use the same file name ,then import THIS file DISK BY DISK ,operator choose append but not overWRITE ,IT IS VERY EASY TO INSTALL A BIG FILE
4. IF OPERATOR INSTALL ETHERNET ,IT IS MORE EASY TO INSTALL A BIG FILE FROM NET .

1.3.2.7.5 F5 : Export

- Function: Output file to floppy disk
- Operation: Select a file by pressing (↑ 、 ↓ 、 ← 、 →) and then press F5. After following the prompt in the dialog box, confirm this operation by pressing 【ENTER】 .



1.3.2.7.6 F6 : RS232 Import

- Function: use RS232 communication function to import NC files
- Operation: Follow the prompts in the dialog box and type in needed data.

G54	TEST104 N0 L1	Program	20:21:19	2007/03/20
Program storage free space 2133655552 bytes				
111	864790 01/30/07 02:23 pm			
123	28 02/07/07 01:59 pm	G01 X150.000 Y150.000 Z0.000;		
92103023	4212 02/01/07 00:01 pm	M48		
A0001	345 06/01/04 06:21 pm			
FLYHOH~1	31533 03/06/07 06:26 pm			
FLYHOH~2	39425 03/06/07 06:26 pm			
MDIBLOCK	1 08/14/06 02:17 pm			
00001	68 11/24/03 03:07 pm	G00X50.Y50.Z50.		
01000	258 11/18/03 07:59 pm	銑床測試程式 FOR G73		
01010	263 11/18/03 07:59 pm	銑床測試程式 FOR G74		
03295	1053 03/03/07 07:11 pm			
TEST101	37052 01/30/07 07:11 pm			
TEST102	32862 01/30/07 08:35 pm	00001		
TEST103	32893 01/30/07 08:35 pm	00001		
TEST104	30152 02/10/07 11:03 am	00001		
TEST105	32862 02/13/07 08:20 pm	00001		
TEST2	321346 02/28/07 02:14 pm			
WANG	379 04/14/06 01:20 pm			
RS232		DNC Program		

Import file, RS232, to:

OK

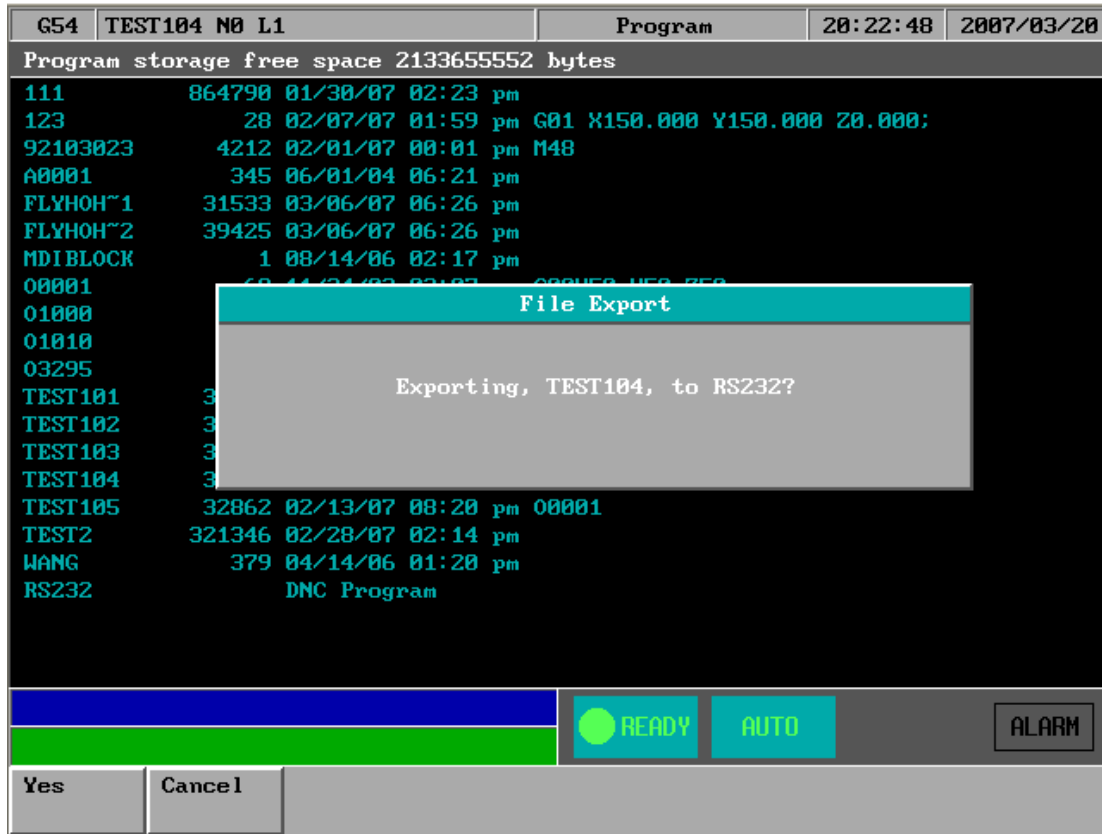
READY

AUTO

ALARM

1.3.2.7.7 F7 RS232 Export

- Function: Use RS232 communication function to Export NC files
- Operation: Follow the prompts in the dialog box and type in needed data.



1.3.2.7.8 RS232 communication parameter setting

CNC parameter settings from : 3901 to 3949

G54	TEST104 N0 L1	Parameter	20:35:43	2007/03/20
No.	DescrUseron	End User	ModiTime: 12:58:21	2007/03/20
3903	File transfer port number(1:COM1;2:COM2)		1	
3905	Macro program port number(1:COM1;2:COM2)		1	
3921	COM1 baud(0:24;1:48;2:96;3:192;4:384;5:576;6:115)		3	
3922	COM1 data bit number		8	
3923	COM1 exchange code type(0:ASCII;1:EIA;2:ISO)		0	
3924	COM1 control code(0:No;1:DC2;2:DC4;3:DC2DC4)		0	
3925	COM1 end-of-block output code(0:E0B;1:CR+E0B)		0	
3926	COM1 DC3 control code parity(0:Off;1:On)		0	
3927	COM1 flow control(0:No;1:CtsRts;2:X0nOff;3:RS485)		1	
3928	COM1 parity check(0:No;1:Odd;2:Even)		0	
3929	COM1 stop bit number(1:1Bit;2:2Bit)		1	
3941	COM2 baud(0:24;1:48;2:96;3:192;4:384;5:576;6:115)		3	
3942	COM2 data bit number		8	
3943	COM2 exchange code type(0:ASCII;1:EIA;2:ISO)		0	
3944	COM2 control code(0:No;1:DC2;2:DC4;3:DC2DC4)		0	
3945	COM2 end-of-block output code(0:E0B;1:CR+E0B)		0	
[7,8]				
User Param		App Param	Mechanism Param	Pitch Table
Goto Param		System Setting		ALARM

1.3.3 F3 : DlgInput (ONLY for 940M)

Pressing "F3" under the main menu to begin dialog box input as shown in the following figure. After users follow the prompts shown on the dialog box to key in every needed parameter, the CNC milling controller can start milling a work piece immediately.

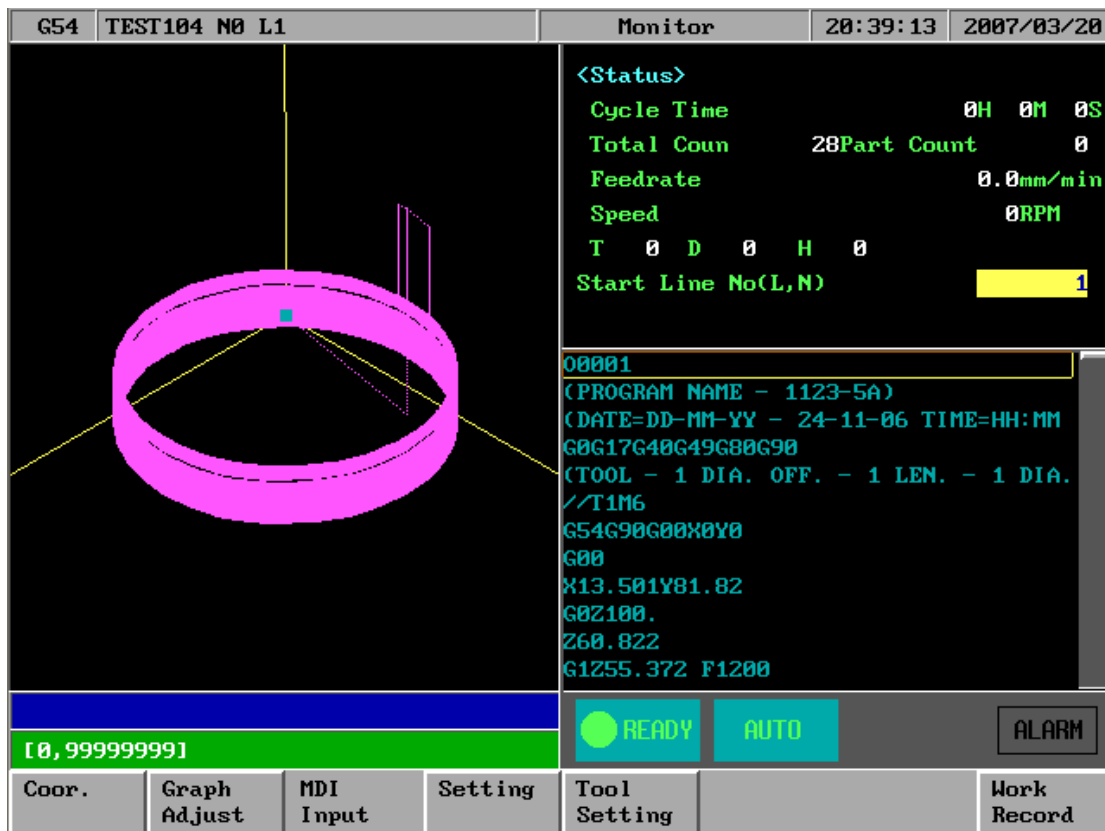
1.3.4 F4 : Monitor

This selection displays machining speed, time, manual data input (MDI) and some machine information such as coordinate, range or program at the run time. Press F4 under the main menu to select this function.

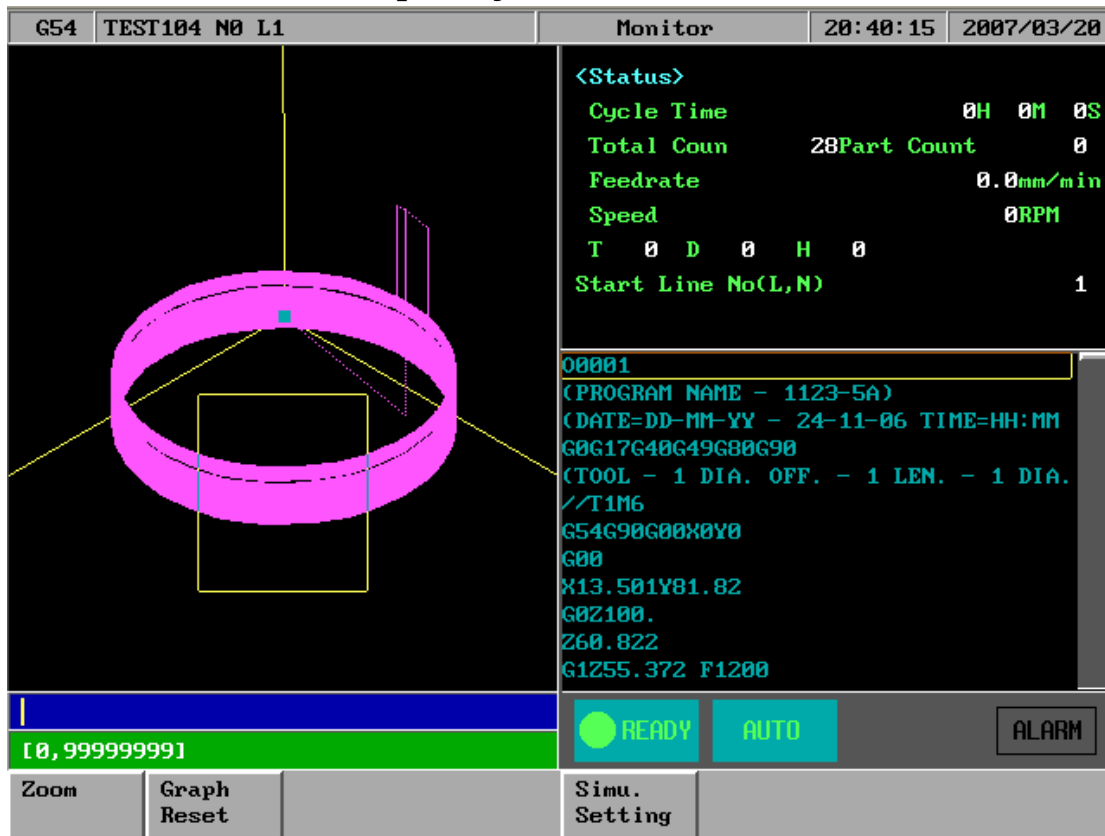
Key Selections :

1.3.4.1 F1 : Coord

- Function: Toggle way of display among the four different coordinate systems, graphical working paths display and the absolute coordinate. (Absolute coordinate display at the upper right corner of the left half screen.)
- Operation: Under Monitor submenu, press F1 to toggle coordinate display between the four coordinates as shown as the following figure:



1.3.4.2 F2 : Graph Adjust



Key Selections :

1.3.4.2.1 F1 : Zoom

- Function: To enlarge the workpiece graph.
- Operation: Under Monitor submenu, press F2 and then F1. The operator can use the cursor to move the frame to the determined area.

1.3.4.2.2 F2 : Graph Reset

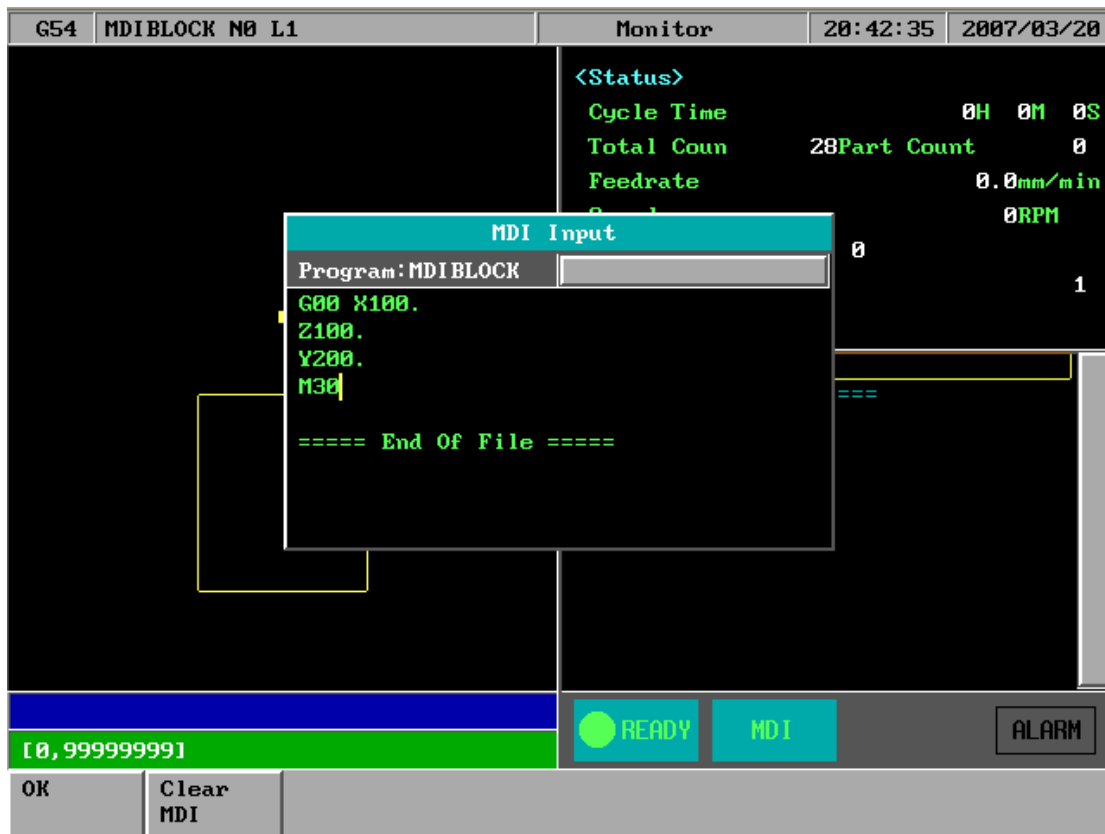
- Function: To recover the zoomed workpiece graph.
- Operation: Under Monitor submenu, press F2 "Graph Adjust" and then F2 "Graph Reset"

1.3.4.2.3 F5 : Simu Setting

- Function: To set simulation parameter
- Operation: Under Monitor submenu, press F2 "Graph Adjust" and then F5 "Simu Setting" .

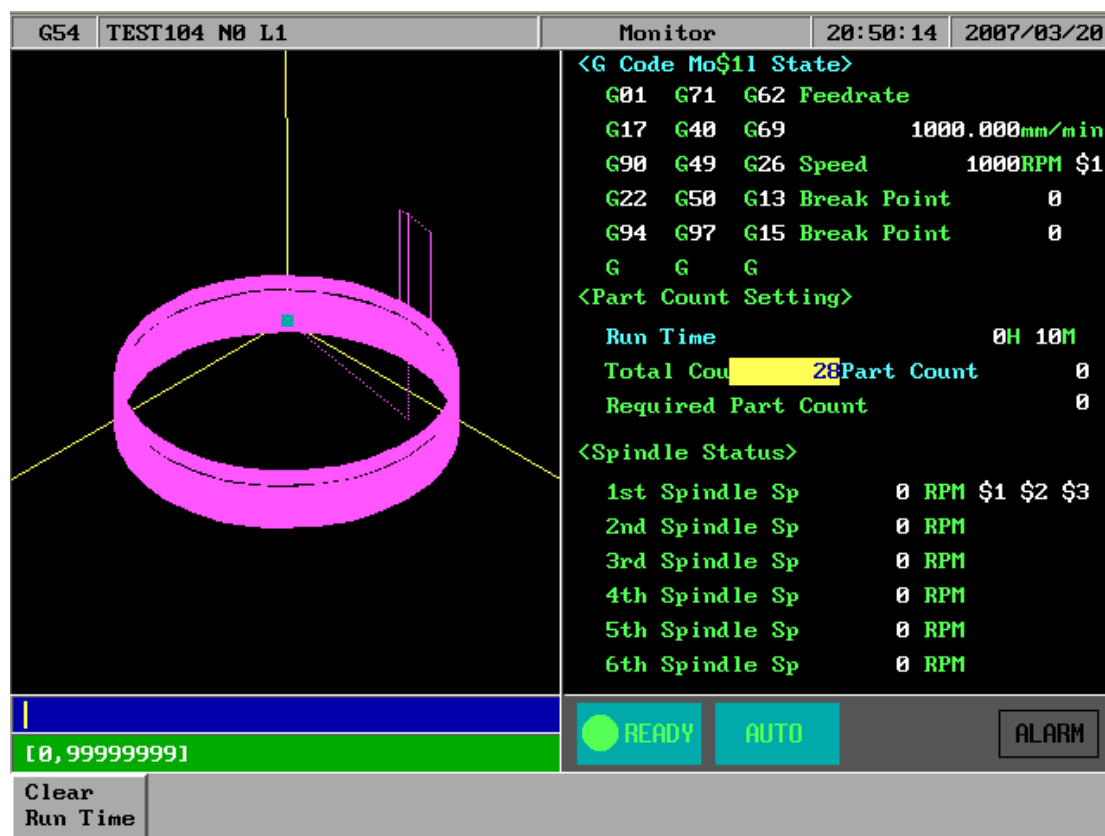
1.3.4.3 F3 : MDI Input

- Function : Manual Data Input
- Operation : Users can operate SNC Milling Controller manually in MDI mode. Press F3 under Monitor submenu and type in single-line G or M code. Press F1 (OK) to confirm the input command. The typed-in command line will show on right upper corner of the screen. Users simply press **【CYCLE START】** on the machine panel to execute the single-line command. The following figure shows an example of this function.



1.3.4.4 F4 : SETTING

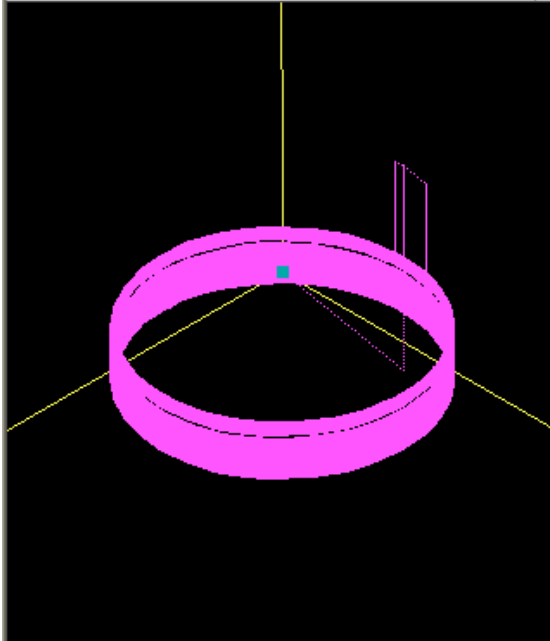
- Function : To set the part count and also set required current
- Operation : From this screen users can set the part count what he needs
 1. When CNC execute M02 ,M30 ,M99 ,part count would add 1 automatically ,
 2. When part count reach required part count ,CNC would stop executing.



1.3.4.5 F5 : Tool Setting

- Function : To set the tool compensation value
- Operation :
 - Radius : G41/G42 tool radius Dn compensation (not diameter)
 - Radius wear : for small radius dimension adjust
 - Length : G43/G44 tool length Hn compensation
 - Length wear : for small length dimension adjust

G54	TEST104 N0 L1	Monitor	20:52:55	2007/03/20
-----	---------------	---------	----------	------------



Input Mode(A:ABS;I:INC) ABS
(X..Z:TeachIn)

<Tool Offset>

	Radius	Rad Wear	Length	Len Wear
01	0.000	0.000	0.000	0.000
02	0.000	0.000	0.000	0.000
03	0.000	0.000	0.000	0.000
04	0.000	0.000	0.000	0.000
05	0.000	0.000	0.000	0.000
06	0.000	0.000	0.000	0.000
07	0.000	0.000	0.000	0.000
08	0.000	0.000	0.000	0.000
09	0.000	0.000	0.000	0.000
10	0.000	0.000	0.000	0.000
11	0.000	0.000	0.000	0.000
12	0.000	0.000	0.000	0.000

READY

AUTO

ALARM

Clear X
Relative

Clear Y
Relative

Clear Z
Relative

Tool No
Data

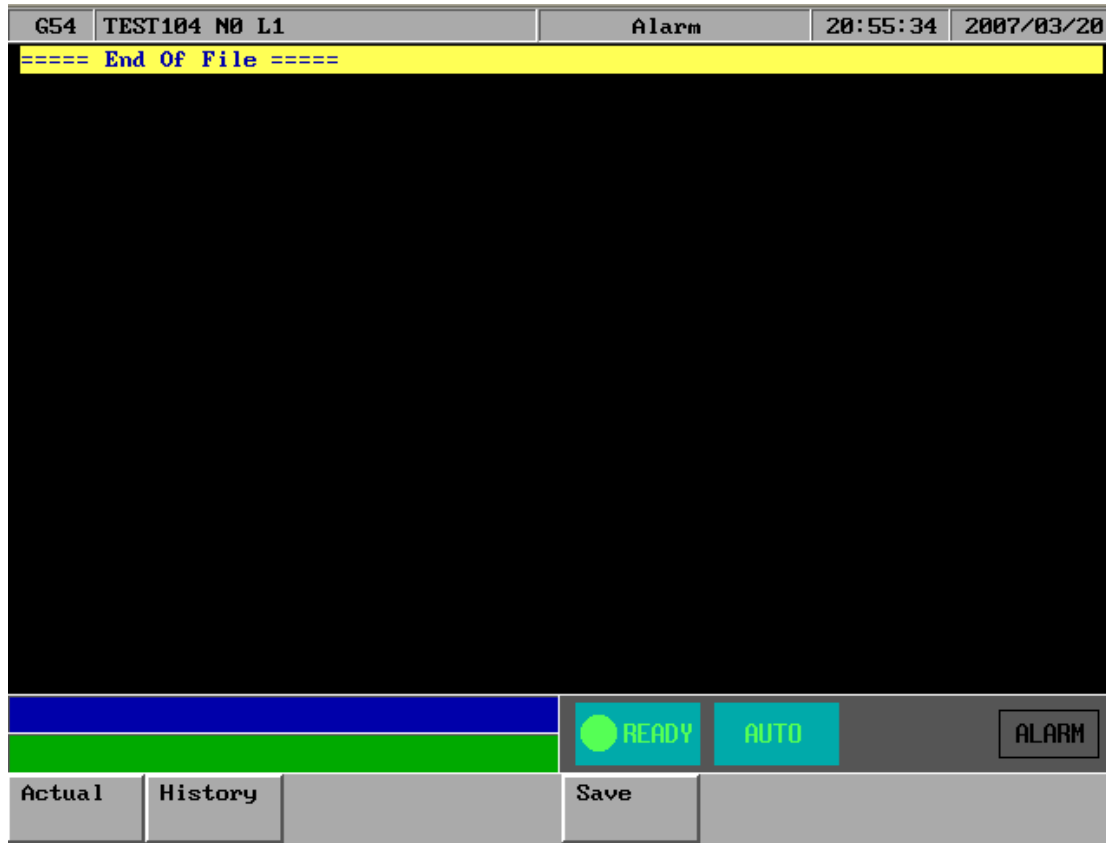
1.3.4.6 F8 : Work Record

- Function : This table can record 300 sets executed NC file ,this is very helpful to know the end of user working history .

[illegible]

1.3.5 F5 : Alarm

Whenever the system or the program stops running due to some errors, there will be an alarm message shown on the screen. In order to clear the errors, users can press F5 in the main menu for Alarm submenu as shown in the following figure.



Key Selections:

1.3.5.1 F1 : Actual

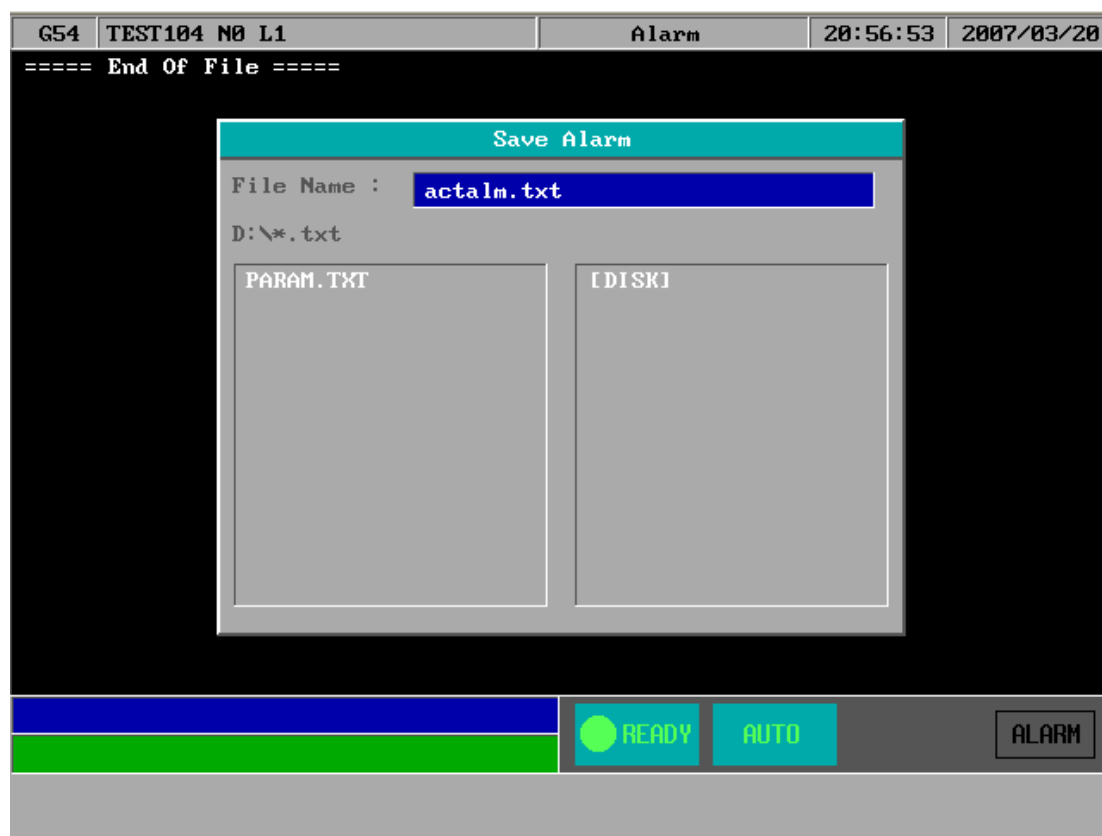
- Operation: Under the Alarm submenu, press F1 to show current alarm situation.

1.3.5.2 F2 : History

- Operation: Under the Alarm submenu, press F2 to show the alarm history of the system.

1.3.5.3 F5: Save

- Function : Save Alarm History To File.
- Operation: Under the Alarm submenu, press F5 to save alarm history to a file as shown in the following figure. A dialog box will prompt users to type in file name to be saved. After selecting a disk drive by using (↑、↓、←、→), users press **【ENTER】** to confirm this operation.



1.3.6 F6 : Parameter

Under the main menu, press F8 and then F1 to enter this function as shown in the following figure.

[illegible]

1.3.7 F7 : Diagnosis

This selection provides users with direct access to the memory area for parameter checking, parameter settings and NC diagnosis function. It can also be used to maintain and debug the control devices. Under the main menu, press F6 and then F2 to access this function as shown in the following figure.

G54 TEST104 N0 L1											Diagnosis											21:00:09				2007/03/20			
I Bits																													
No.	00	01	02	03	04	05	06	07	08	09	No.	00	01	02	03	04	05	06	07	08	09								
000	00	00	00	00	00	00	00	00	00	00	010	00	00	00	00	00	00	00	00	00	00								
020	00	00	00	00	00	00	00	00	00	00	030	00	00	00	00	00	00	00	00	00	00								
040	00	00	00	00	00	00	00	00	00	00	050	00	00	00	00	00	00	00	00	00	00								
060	00	00	00	00	00	00	00	00	00	00	070	00	00	00	00	00	00	00	00	00	00								
080	00	00	00	00	00	00	00	00	00	00	090	00	00	00	00	00	00	00	00	00	00								
100	00	00	00	00	00	00	00	00	00	00	110	00	00	00	00	00	00	00	00	00	00								
120	00	00	00	00	00	00	00	00	00	00	130	00	00	00	00	00	00	00	00	00	00								
140	00	00	00	00	00	00	00	00	00	00	150	00	00	00	00	00	00	00	00	00	00								
160	00	00	00	00	00	00	00	00	00	00	170	00	00	00	00	00	00	00	00	00	00								
180	00	00	00	00	00	00	00	00	00	00	190	00	00	00	00	00	00	00	00	00	00								
200	00	00	00	00	00	00	00	00	00	00	210	00	00	00	00	00	00	00	00	00	00								
220	00	00	00	00	00	00	00	00	00	00	230	00	00	00	00	00	00	00	00	00	00								
240	00	00	00	00	00	00	00	00	00	00	250	00	00	00	00	00	00	00	00	00	00								

</

Key Selections:

1.3.7.1 F1 : PLC State

- Function: for upgrade system software ,or Ladder ,system parameter ...

G54 TEST104 N0 L1											Diagnosis											21:01:17				2007/03/20			
I Bits																													
No.	00	01	02	03	04	05	06	07	08	09	No.	00	01	02	03	04	05	06	07	08	09								
000	00	00	00	00	00	00	00	00	00	00	010	00	00	00	00	00	00	00	00	00	00								
020	00	00	00	00	00	00	00	00	00	00	030	00	00	00	00	00	00	00	00	00	00								
040	00	00	00	00	00	00	00	00	00	00	050	00	00	00	00	00	00	00	00	00	00								
060	00	00	00	00	00	00	00	00	00	00	070	00	00	00	00	00	00	00	00	00	00								
080	00	00	00	00	00	00	00	00	00	00	090	00	00	00	00	00	00	00	00	00	00								
100	00	00	00	00	00	00	00	00	00	00	110	00	00	00	00	00	00	00	00	00	00								
120	00	00	00	00	00	00	00	00	00	00	130	00	00	00	00	00	00	00	00	00	00								
140	00	00	00	00	00	00	00	00	00	00	150	00	00	00	00	00	00	00	00	00	00								
160	00	00	00	00	00	00	00	00	00	00	170	00	00	00	00	00	00	00	00	00	00								
180	00	00	00	00	00	00	00	00	00	00	190	00	00	00	00	00	00	00	00	00	00								
200	00	00	00	00	00	00	00	00	00	00	210	00	00	00	00	00	00	00	00	00	00								
220	00	00	00	00	00	00	00	00	00	00	230	00	00	00	00	00	00	00	00	00	00								
240	00	00	00	00	00	00	00	00	00	00	250	00	00	00	00	00	00	00	00	00	00								

1.3.7.2 F2 : System Data

■ Function :

G54	TEST104 N0 L1				Diagnosis				21:02:42	2007/03/20
	NO	DATA	NO	DATA	NO	DATA	NO	DATA	NO	DATA
	0000	17 658	0016	11111	0032	0	0048	0	0064	0
	0001	586890	0017	11111	0033	0	0049	0	0065	0
	0002	293445	0018	11111	0034	0	0050	0	0066	0
	0003	586890	0019	11111	0035	0	0051	0	0067	0
	0004	4883	0020	0	0036	0	0052	0	0068	0
	0005	9766	0021	0	0037	0	0053	0	0069	0
	0006	546	0022	718	0038	-1	0054	0	0070	0
	0007	22468680	0023	22	0039	-1	0055	0	0071	0
	0008	0	0024	0	0040	0	0056	0	0072	0
	0009	0	0025	0	0041	0	0057	0	0073	0
	0010	0	0026	0	0042	0	0058	0	0074	0
	0011	0	0027	0	0043	0	0059	0	0075	0
	0012	0	0028	0	0044	0	0060	0	0076	0
	0013	0	0029	0	0045	0	0061	0	0077	0
	0014	0	0030	0	0046	0	0062	0	0078	0
	0015	0	0031	0	0047	0	0063	0	0079	V10.80

				 READY	AUTO	ALARM
PLC State	System Data	Global Var.	Coord Var.			System Manager

1.3.7.3 F3 : Global Variable

G54	TEST104 N0 L1			Diagnosis	21:03:38	2007/03/20
No	Value	No	Value	No	Value	
0		16		32		
1		17		33		
2		18		34		
3		19		35		
4		20		36		
5		21		37		
6		22		38		
7		23		39		
8		24		40		
9		25		41		
10		26		42		
11		27		43		
12		28		44		
13		29		45		
14		30		46		
15		31		47		
				READY	AUTO	ALARM
PLC State	System Data	Global Var.	Coord Var.			System Manager

1.3.7.4 F4 : Coord Variable

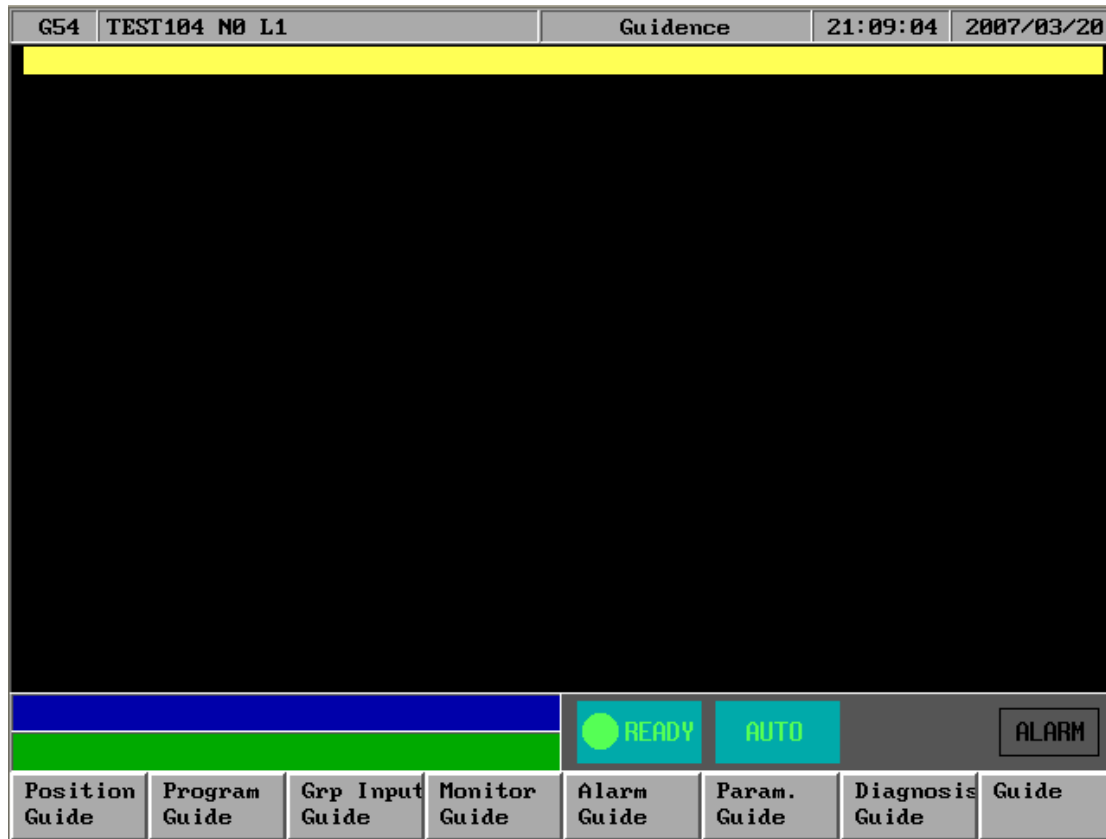
[illegible]

1.3.7.5 F8 : System Manager

[illegible]

1.3.8 F8 : Guidance

If users have any problem about SNC Mill controller, in addition to user manual, users can also use this function for on-line help. Under the main menu, press “F8” for on-line help.



2 Machine operation panel

2.1 2nd machine operation panel

2.1.1 POWER ON

Turn on main power

2.1.2 POWER OFF

Turn off power

2.1.3 Emergency STOP

For safety reason ,press this button ,CNC would stop all movement ,and also stop all main power . So ,people and machine safety is guarantee .

2.1.4 Home mode and Home function

- Description : When CNC power is on ,please do the home function
- Operation :
 1. Mode select to HOME mode
 2. Press axis manual key X+,X-,Y+,Y-,Z+,Z-:
 3. CNC would start the machine home function

2.1.5 Continues JOG (Rapid JOG)

- Description : User can use this function to move the machine by press JOG key
- Operation :
 1. Mode select to CON JOG mode
 2. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move
 3. Operator can use JOG% or G01% adjust Jog federate
 4. When operator press manual key and rapid Key “” at the same time
 5. CNC would move the work table “RAPID speed”
 6. Operator can use G00% Rapid Jog federate

G00 % : Adjust G00 % (F0 .25% .50% .100%)

G01 % : Adjust G01/G02 /G03 feedrate override %:

2.1.6 Incremental JOG

- Description : User can use this function to move the machine by press JOG key
- Operation :
 1. Mode select to INC JOG mode
 2. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move a fixed distance
 3. Operator can set the incremental distance by G00 rotary switch
 4. , *1 : 1um , *10 : 10um , *100 : 100um

2.1.7 MPG JOG

- Description : User can use this function to move the machine by MPG(Manual Pulse Generator)
- Operation :
 1. Mode select to MPG mode
 2. Select axis by hand box
 3. Select incremental distance
 4. Press axis manual key X+,X-,Y+,Y-,Z+,Z- ,work table would move a fixed distance *1 : 1um , *10 : 10um , *100:100um , *1000 : 1000um

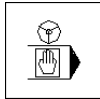
2.1.8 AUTO mode NC file execute

- Description : User use this function to execute NC file
- Operation :
 1. mode select to AUTO mode
 2. After Home function .AUTO mode is available
 3. Set workpiece coor.(G54..G59) ,CNC default G54 ,if user don't set any G54..G59 in the NC file
 4. Set to "Tool Setting" ,to select tool radius and tool length .
 5. Press "START" key to start the NC program .
 6. Press "Feedhold" key to feedhold the NC program ,if it's necessary

2.1.9 MDI mode single block execute

- Description : User use this function to execute a block without NC file
- Operation :
 1. mode select to MDI mode
 2. After Home function .MDI mode is available
 3. Main function select F4"Monitor"
 4. Press F3 "MDI Input", screen would pop up a window.
 5. After key in data ,press "ENTER" key to input the data
 6. Press "START" key to start the MDI block.
 7. If MDI block syntax is correct ,data in MDI menu would disappear

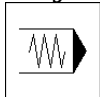
2.1.10 MPG Simulation



- Description : User can use this function to check NC file
- Operation :
 1. Mode select to AUTO mode
 2. Press this button ,and button led light “ON”
 3. Press “START” key to start the NC file.
 4. CNC would change machine status from “READY” to “BUSY”
 5. Machine is still not moving
 6. Operator can use rotate MPG to start the NC file
 7. MPG rotate faster ,machining speed is faster
 8. When MPG stop ,CNC stop.
 9. This function can “Enable” “ Disable” immediately

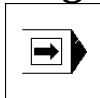
P.S. this function is very friendly for user to check his programs

2.1.11 Dry Run



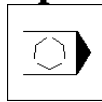
- Description : User can use this function to check NC file
- Operation :
 1. Mode select to AUTO mode
 2. Press this button ,and button led light “ON”
 3. Press “START” key to start the NC file.
 4. CNC would change machine status from “READY” to “BUSY”
 5. This function can “Enable” “ Disable” immediately

2.1.12 Single block



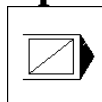
- Description : User can use this function to check NC file
- Operation :
 1. Mode select to AUTO mode
 2. Press this button ,and button led light “ON”
 3. Press “START” key to start the NC file.
 4. CNC would execute NC file only one block and STOP
 5. CNC would change machine status from “BUSY ” to “B_STOP”
 6. Press “START” again ,then CNC execute next block
 7. This function is for user to check his NC file Block by Block

2.1.13 Option Stop



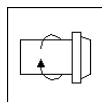
- Description : User can use this function to decide NC file M01 is STOP or not
- Operation :
 1. Mode select to AUTO mode
 2. Press this button ,and button led light “ON”
 3. Press “START” key to start the NC file.
 4. When CNC execute “M01” ,CNC would STOP
 5. CNC would change machine status from “BUSY ” to “Feedhold”
 6. This function uses to change tool or check workpiece

2.1.14 Option Skip

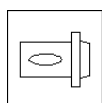


- Description : User can use this function to decide NC file ‘/’ is skip or not
- Operation :
 1. Mode select to AUTO mode
 2. Press this button ,and button led light “ON”
 3. Press “START” key to start the NC file.
 4. When CNC execute “/” ,CNC would skip this block
 5. If this key is not pressed ,CNC would execute this block

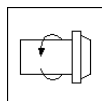
2.1.15 Spindle control



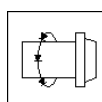
Spindle CW rotate



Spindle stop

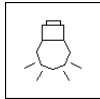


Spindle CCW rotate



Spindle low speed: When spindle is rotate, press this key, spindle would rotate with low speed

2.1.16 Working led



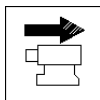
ON/OFF working led

2.1.17 Working Liquid

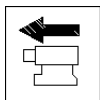


Flush working liquid

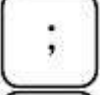
2.1.18 Aux table backward



2.1.19 Aux table forward



2.2 Text key description:

	~		: English 26 character key
	~		: numerical key
			: delete a character
			: Insert /replace mode switch
			: for select keyboard the other textkey
			: for select keyboard the other textkey
			: "Backspace" delete a character
			: "RESET " abort the CNC status ,so please be careful to use this key
			: to input current data to input box
			: press this key ,user can get help message about this screen
			: for optional skip key input
			: end of block
			: decimal fraction

“(, “)”, “[, “]”, “[, “!”, “&”, “\$”, “#”, “<”, “>”, “=”, “%”, “@”, “*”, “:”, “,”, “+”, “-“

Above symbols are used for “Program Edit” mode.



: edit cursor Page Up /Page Down



: edit cursor control key

3 How to operate SYNTEC 900ME

This chapter is written for user task,when user operates this controller ,operator can follow task description as below STEP by STEP ,so very easy to use this controller ,TASK description as below :

1. Manuel function(JOG ,INC_JOG ,MPG)
2. HOME
3. Open a file (EDIT / FLOPPY /RS232)
4. Tool setting (G40/G41/G42 ,G43/G44/G49)
5. Tool Length measurement (G43/G44/G49)
6. Setting the Workpiece origin offset value(G54..G59)
7. Manual Data Input(MDI)
8. Assigned an executing NC file (AUTO)
9. Graphic Simulation
10. How to check NC file in SYNTEC controller
11. Auto center
12. Auto Tool

3.1 Manual function (JOG, INC_JOG, MPG)

When power on SYNTEC CNC, there are 3 mode to manual machine

- 1st CON_JOG :
 1. Release emergency stop button ,CNC status “NOT READY” change to “READY ”
 2. Mode select switch rotate to JOG mode
 3. Press axis direction key(X+,X-,Y+,Y-,Z+...) ,table would move
 4. Operator can use JOG% adjust JOG speed
 5. Operator can press axis direction key and rapid key “~” at the same time, machine will move by rapid speed
 6. Rapid JOG speed can be adjusted by G00%

- Incremental JOG :
 1. Release emergency stop button ,CNC status “NOT READY” change to “READY ”
 2. Mode select switch rotate to INC JOG mode
 3. Press axis direction key(X+,X-,Y+,Y-,Z+...) ,table will move a fixed distance once
 4. Operator can select incremental distance by G0% (*1,*10,*100)

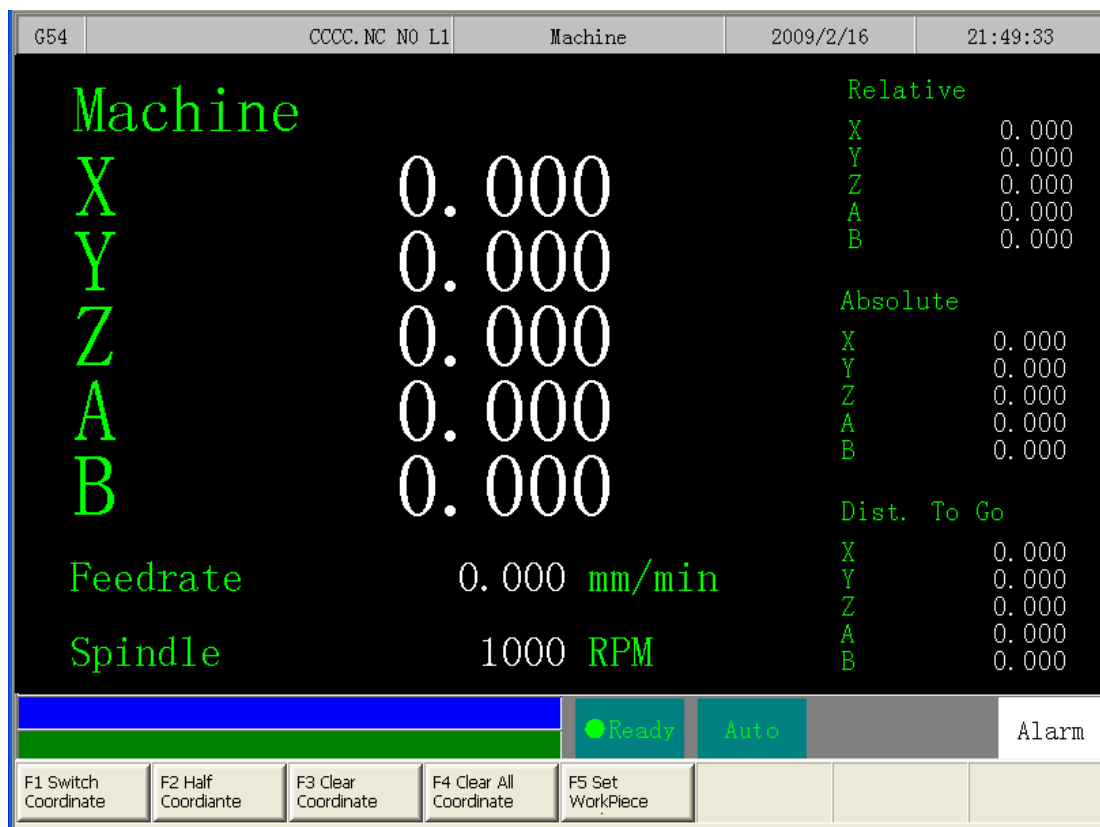
- MPG incremental jog (MPG):
 1. Release emergency stop button ,CNC status “NOT READY” change to “READY ”
 2. Mode select switch rotate to MPG INC JOG mode
 3. Select movement axis
 4. Select movement distance (*1,*10,*100)
 5. Rotate MPG ,table would move .

3.2 HOME

Because tool setting ,workpiece coordinate setting is based on Machine zero point .So ,it is necessary to make sure where is machine zero (HOME). When CNC bootup, execute HOME function is very important ,otherwise SYNTEC CNC controller would not be allowed to start AUTO NC files

■ Procedure :

1. Release emergency stop button ,CNC status “NOT READY” change to “ READY ”
2. Mode select switch rotate to HOME mode
3. Press axis direction key(X+,X-,Y+,Y-,Z+...) ,axis would start HOMING
4. Home direction is defaulted in the CNC parameter
5. Home function can run 3 axis at the same time
6. After home function, machine coordinate would be zero .
7. After home function completed, software limit protection is available .Therefore, please don't run machine too fast, before HOME function.



3.3 Open a file (EDIT / FLOPPY / RS232)

■ Procedure :

1. Press Group function key “Program”
2. Press submenu function key “File manage”
3. Screen display file system screen

01000 N0 L1		Program		13:48:37	2001/02/01	
Program storage free space 2147155968 bytes						
00001	51790	01/18/01	04:14	pm	G00X50.Y50.Z50.	
01000	260	02/01/01	01:41	pm	銑床測試程式 FOR G73	
01010	265	02/01/01	01:09	am	銑床測試程式 FOR G74	
RS232 DNC Program						

4. Press F1 “New file” ,to open a new file
5. Press F2”copy file” ,to copy current hilight file to target file
6. Press F3 “delete file“ ,to delete current hilight file
7. Press F4 “Inport” ,to inport a new file from floppy disk
8. Press F5 “Export “ ,to export hilight file to floppy disk
9. Press F6 “RS232 Inport” ,to inport a new file from RS232
10. Press F7 “RS232 Export “ ,to export hilight file to RS232

3.4 Tool setting (G40/G41/G42, G43/G44/G49)

- Procedure for setting Tool offset value :
 1. Press Group function key “Monitor”
 2. Press function key “Tool Setting”
 3. Move the cursor to the compensation value to be set or change using page keys and cursor keys
 4. Type “A” or “I” key to selected input type is “Absolute” or “Incremental”
 5. Generally use Absolute type to input Tool radius and Tool length
 6. Use Incremental type to input radius wear and Length wear for small value adjust
 7. (Tool radius + radius wear) is real G41/G42 compensation value
 8. (Tool length + length wear) is real G43/G44 compensation value

G54 RS232 N0 L1		Monitor		12:38:39	2009/02/14																																																																	
Absolute X -10.000 Y 0.000 Z 0.000 A 0.000 B 0.000 Machine Relative X 0.000 X 0.000 Y 0.000 Y 0.000 Z 0.000 Z 0.000 A 0.000 A 0.000 B 0.000 B 0.000			Input Mode(A:ABS;I:INC) ABS (X..Z:TeachIn) <Tool Offset> <table border="1"> <thead> <tr> <th></th> <th>Radius</th> <th>Rad Wear</th> <th>Length</th> <th>Len Wear</th> </tr> </thead> <tbody> <tr><td>01</td><td>0.000</td><td>0.000</td><td>28.301</td><td>0.000</td></tr> <tr><td>02</td><td>0.000</td><td>0.000</td><td>0.000</td><td>0.000</td></tr> <tr><td>03</td><td>0.000</td><td>0.000</td><td>41.017</td><td>0.000</td></tr> <tr><td>04</td><td>0.000</td><td>0.000</td><td>28.301</td><td>0.000</td></tr> <tr><td>05</td><td>0.000</td><td>0.000</td><td>0.000</td><td>0.000</td></tr> <tr><td>06</td><td>0.000</td><td>0.000</td><td>15.626</td><td>0.000</td></tr> <tr><td>07</td><td>0.000</td><td>0.000</td><td>28.301</td><td>0.000</td></tr> <tr><td>08</td><td>0.000</td><td>0.000</td><td>0.000</td><td>0.000</td></tr> <tr><td>09</td><td>0.000</td><td>0.000</td><td>15.626</td><td>0.000</td></tr> <tr><td>10</td><td>0.000</td><td>0.000</td><td>0.000</td><td>0.000</td></tr> <tr><td>11</td><td>0.000</td><td>0.000</td><td>0.000</td><td>0.000</td></tr> <tr><td>12</td><td>0.000</td><td>0.000</td><td>0.000</td><td>0.000</td></tr> </tbody> </table>				Radius	Rad Wear	Length	Len Wear	01	0.000	0.000	28.301	0.000	02	0.000	0.000	0.000	0.000	03	0.000	0.000	41.017	0.000	04	0.000	0.000	28.301	0.000	05	0.000	0.000	0.000	0.000	06	0.000	0.000	15.626	0.000	07	0.000	0.000	28.301	0.000	08	0.000	0.000	0.000	0.000	09	0.000	0.000	15.626	0.000	10	0.000	0.000	0.000	0.000	11	0.000	0.000	0.000	0.000	12	0.000	0.000	0.000	0.000
	Radius	Rad Wear	Length	Len Wear																																																																		
01	0.000	0.000	28.301	0.000																																																																		
02	0.000	0.000	0.000	0.000																																																																		
03	0.000	0.000	41.017	0.000																																																																		
04	0.000	0.000	28.301	0.000																																																																		
05	0.000	0.000	0.000	0.000																																																																		
06	0.000	0.000	15.626	0.000																																																																		
07	0.000	0.000	28.301	0.000																																																																		
08	0.000	0.000	0.000	0.000																																																																		
09	0.000	0.000	15.626	0.000																																																																		
10	0.000	0.000	0.000	0.000																																																																		
11	0.000	0.000	0.000	0.000																																																																		
12	0.000	0.000	0.000	0.000																																																																		
 			● READY AUTO ALARM																																																																			
Clear X Relative	Clear Y Relative	Clear Z Relative	Tool No Data																																																																			

3.5 Tool Length measurement (G43/G44/G49)

■ Procedure :

1. Use manual operation to move the reference tool until it touches the specified position on the machine.
2. Press Group function key “Position” and clear relative coordinate to zero
3. Press Group function key “Monitor” and press “Tool Setting” to this screen
4. Use manual operation to move the tool until it touches the same specified position that have been measured. The difference of measured that is between the length of the reference tool and the tool is displayed in the relative coordinates on the screen.
5. Move the cursor to the compensation number for the target tool(the cursor can be moved in the same way as for setting tool compensation values)

3.6 Setting the Workpiece origin offset value (G54...G59)

■ Procedure :

1. Press group function key “Position”
2. Press sub selection soft key “Workpiece coord.”

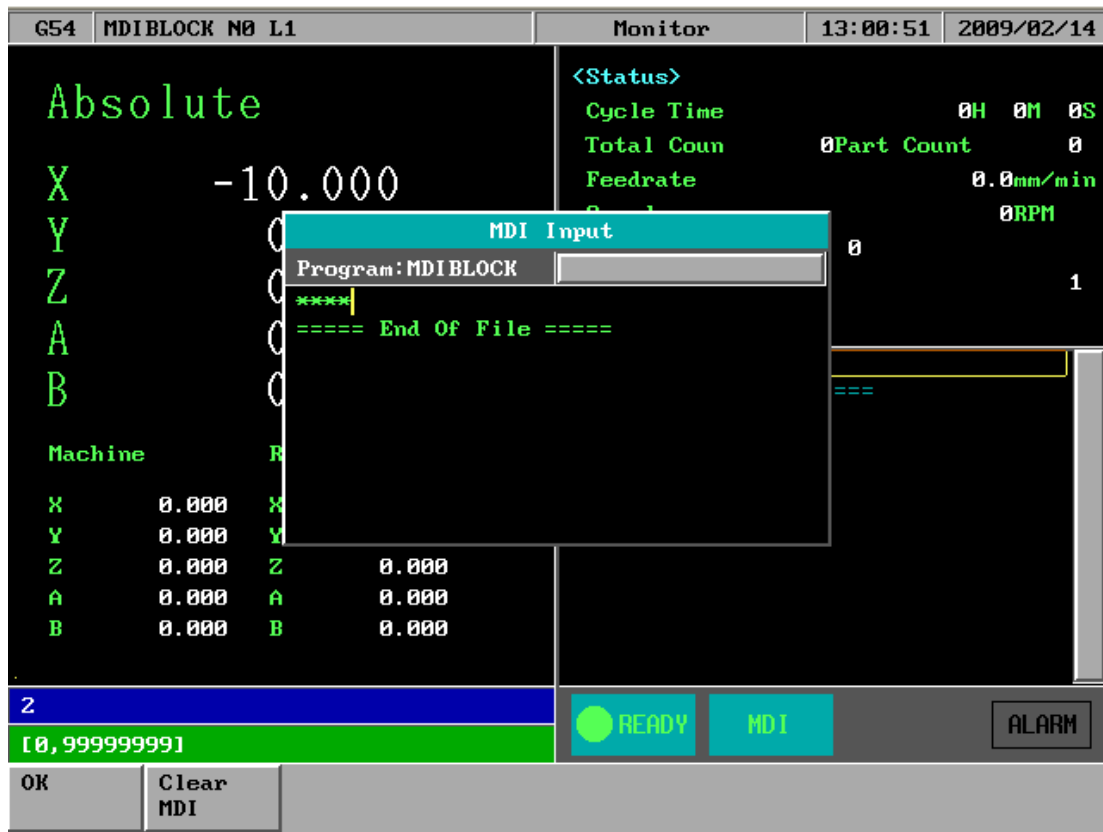
G54		RS232 N0 L1		Position		12:39:36		2009/02/14	
G59		G59.1		G59.2		Machine			
X	0.000	X	0.000	X	0.000	X	0.000		
Y	0.000	Y	0.000	Y	0.000	Y	0.000		
Z	0.000	Z	0.000	Z	0.000	Z	0.000		
A	0.000	A	0.000	A	0.000	A	0.000		
B	0.000	B	0.000	B	0.000	B	0.000		
G59.3		G59.4		G59.5		Relative			
X	0.000	X	0.000	X	0.000	X	0.000		
Y	0.000	Y	0.000	Y	0.000	Y	0.000		
Z	0.000	Z	0.000	Z	0.000	Z	0.000		
A	0.000	A	0.000	A	0.000	A	0.000		
B	0.000	B	0.000	B	0.000	B	0.000		
						Aux. Coordinate			
						X	0.000		
						Y	0.000		
						Z	0.000		
						A	0.000		
						B	0.000		

3. The screen for displaying the workpiece origin offset values consists two pages . Display a desired page by Press PageUp/PageDn key
4. Move the cursor to the workpiece origin offset to changed the values.
5. “Extenal shift” input the value ,which can shift the whole coordinate(G54..G59.8) simultaneous
6. F1”coor. Latched”: user can press this function key ,and CNC would latch current machine coordinate to the screen where sursor is located .

3.7 Manual Data Input (MDI)

■ Procedure :

1. Mode select switch rotate to MDI mode
2. Press group function key “Monitor ”
3. Press sub selection soft key “MDI input”

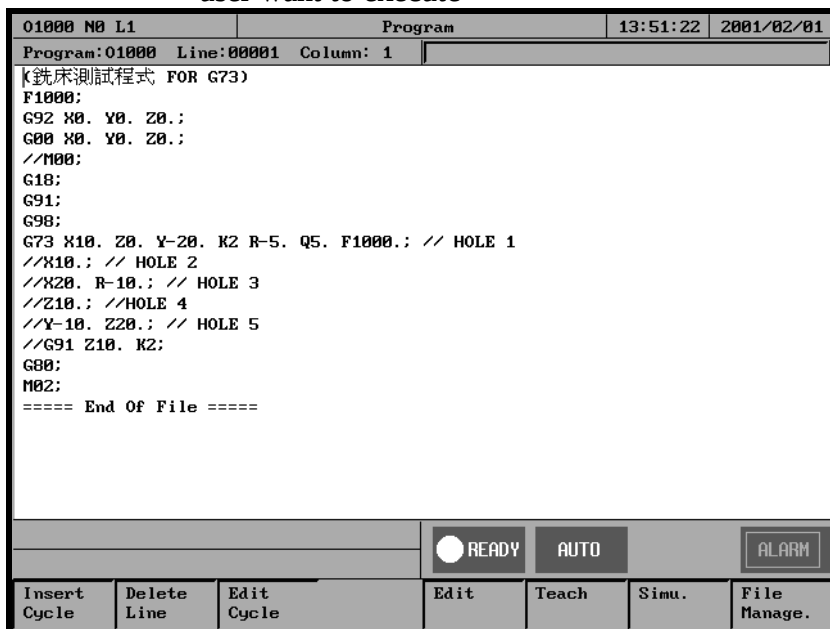


4. The screen display the MDI input window
5. Key in MDI data at input bar and press “ENTER”
6. Press 2nd operation panel “START” to execute the current block
7. If current block SYNTAX is correct ,the data in the window would be disappear .

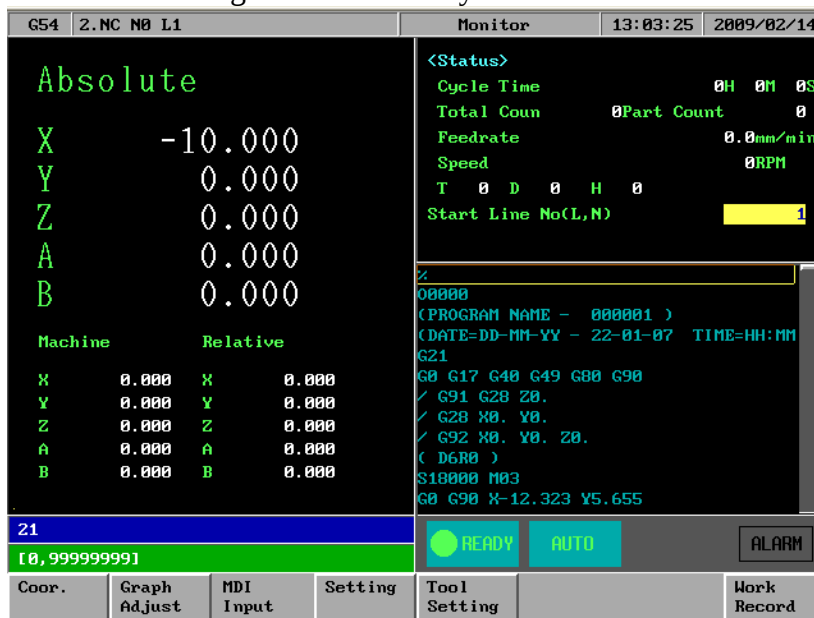
3.8 Assigned an executing NC file (AUTO)

■ Procedure :

1. Mode select switch rotate to AUTO mode
2. Make sure CNC status is "READY"
3. Press group function key "Program", select NC file what user want to execute



4. Press group function key "Monitor", then executing file is assigned automatically.

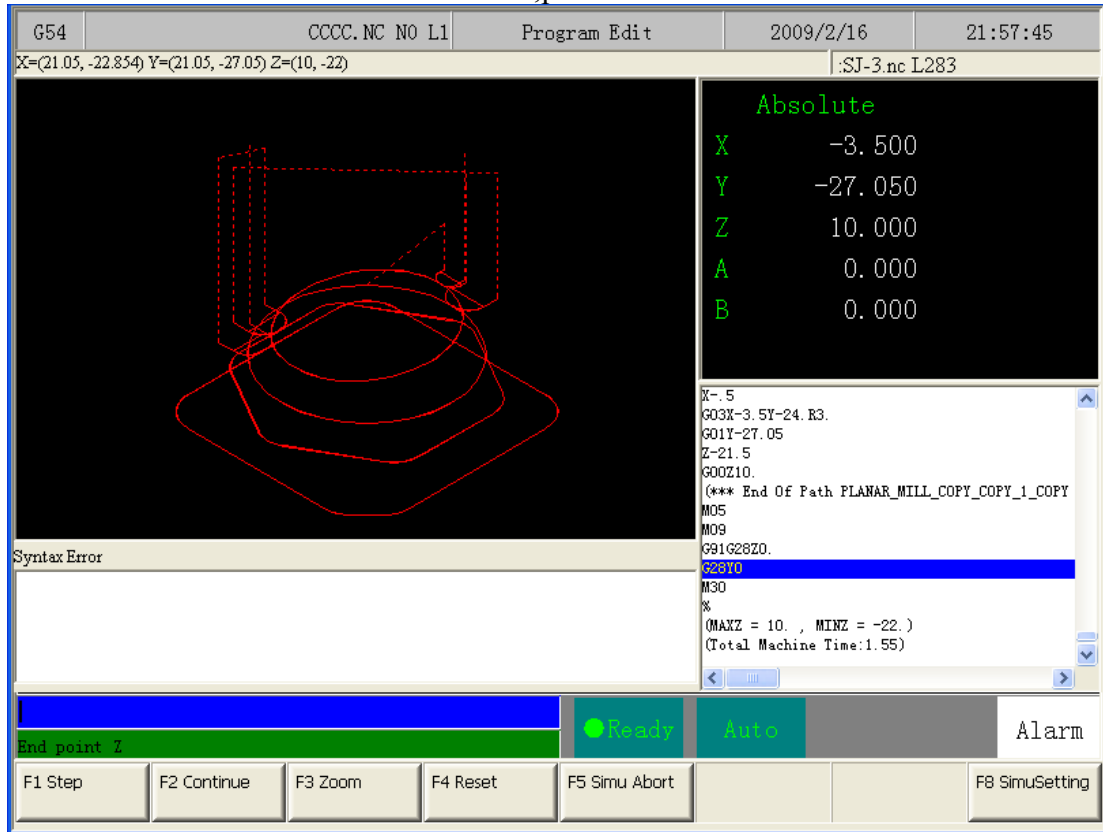


5. Please make sure CNC status is "READY", that is the only available status to assigned executing NC file.

3.9 Graphic Simulation

■ Procedure :

1. Mode select switch rotate to AUTO mode
2. Press group function key “Program”, select NC file what user want to execute ,press sub menu “Simulation”

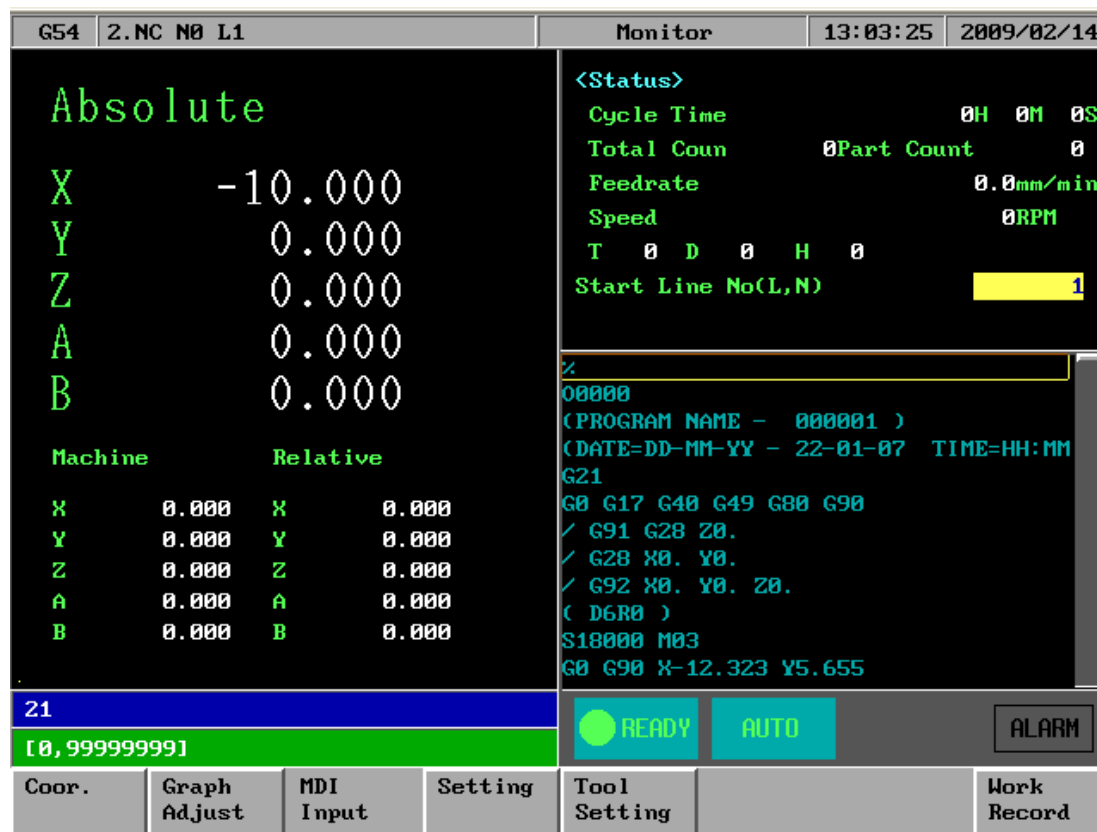


3. User can use “STEP” to check NC file step by step
4. Use “Continue” to simulation whole picture
5. Use “Zoom” check more detail
6. Use “Simulation setting” set simulation parameter
7. CNC status is “READY”, Press group function key “Monitor”
8. that is the available status to assigned executing NC file .

3.10 How to check NC file in SYNTEC controller

■ MPG simulation Procedure :

1. Mode select switch rotate to AUTO mode
2. Press group function key “Monitor”



3. Press “MPG simulation “key enable this function (led on)
4. Press “Start “key ,CNC status from “READY” to “BUSY”
5. Machine table current status is static
6. Operator rotate MPGs ,then table is start moving along cutting path
7. MPG rotates more fast ,table moves more fast ,MPG stop ,table is stop from monitor screen ,operator can see the cutting cursor move along simulation path
8. Operator also can press “Single Block” key ,enable single block function when “MPG simulation “ function is ON ,then user can use two function simultaneous ,check NC file STEP by STEP ,with” MPG simulation “ function .
9. “ MPG simulation “ can control table forward ,also can control backward too ,but NC file would stop at M,S,T code when backward .

3.11 Auto Center

Specification:

It is usually used to look for the center of the coordinate which is the origination of the executing procedure. So it needs to touch both sides of the tool. The center workpiece can be calculated by system after getting their coordinate, then the operator chooses a group of workpiece coordinate (G54~G59.9) as the origination of the executing procedure.

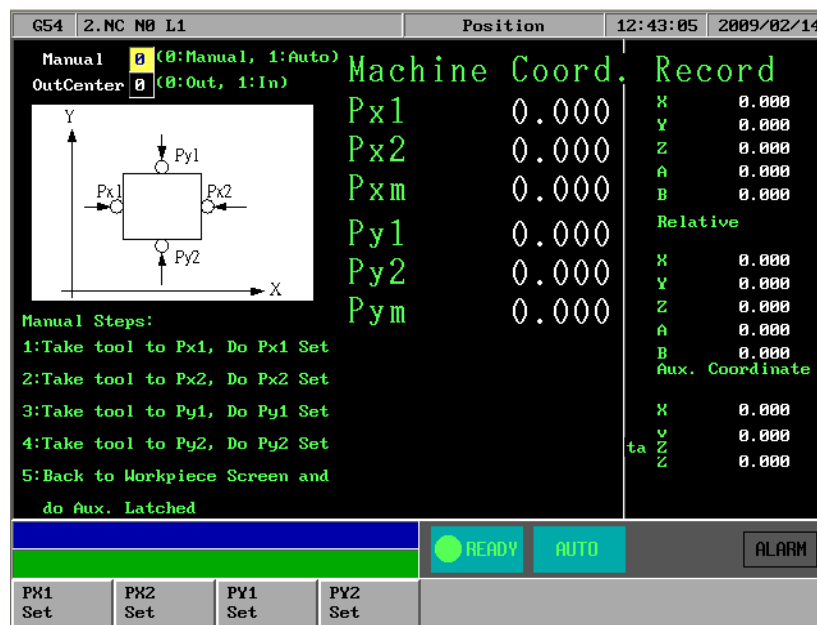
Here we supply the operation introduction of manual center and auto center as follows.

3.11.1 Manual Center

Operator moves tool to touch the edge of the workpiece by rotating MPG, the center coordinate can be calculated by system after pressing function key.

■ Operation:

1. To enter center submenu, press F1"Positon"=>F5"workpiece Coord"=>F5"Middle Func".
- 2.
3. To change middle function to manual center, input 0(manual center) in center submenu, then press enter, manual center submenu can be displayed as follows.



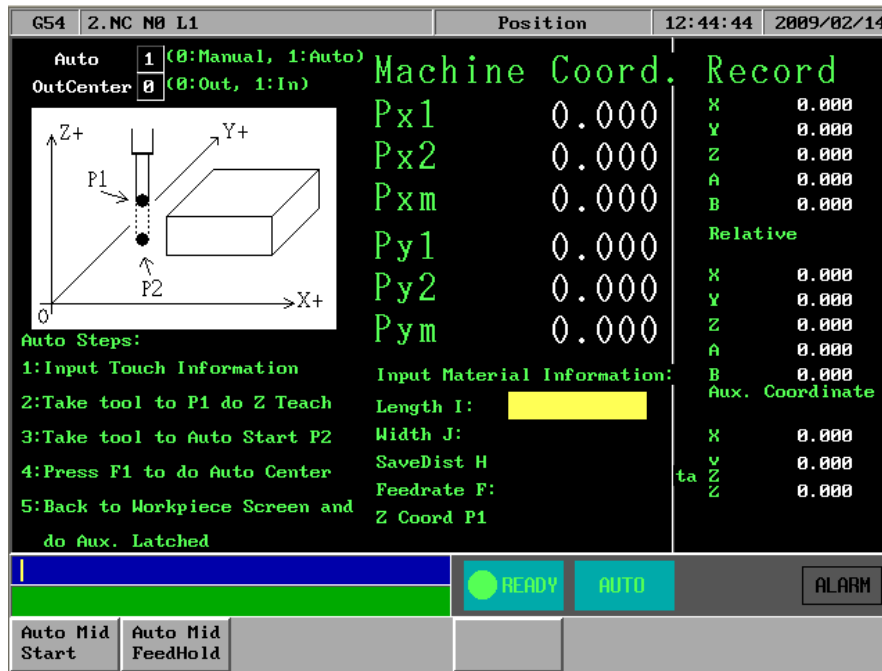
4. Take the tool to touch Px1 spot in the figure by handwheel, press F1"Px1", the X machine coordinate value of Px1 can be noted in the screen and calculate X intermediate machine coordinate with Px2, the value can be displayed in PxM and X-axis of Aux coordinate.
5. Take the tool to touch Px2 spot in the figure by handwheel, press F2"Px2", the X machine coordinate value of Px2 can be

noted in the screen and calculate X intermediate machine coordinate with Px1, the value can be displayed in Pxm and X-axis of Aux coordinate.

6. Take the tool to touch Py1 spot in the figure by handwheel, press F3"Py1", the Y machine coordinate value of Py1 can be noted in the screen and calculate Y intermediate machine coordinate with Py2, the value can be displayed in Pxm and Y-axis of Aux coordinate.
7. Take the tool to touch Py2 spot in the figure by handwheel, press F4"Py2", the Y machine coordinate value of Py2 can be noted in the screen and calculate Y intermediate machine coordinate with Py1, the value can be displayed in Pxm and Y-axis of Aux coordinate.
8. Press "Esc" to exit "Auto Center" and return to "Workpiece Coor" submenu.
9. In "Workpiece Coor" submenu move cursor to the location of the workpiece coordinate which will be set, press F3 "AUX Latched". At this time the system will set the value of the AUX coordinate into the filed in accordance with the axis where the cursor stop.

3.11.2 Auto center operation

Auto center function is different to manual center function. The operator only need to input the size of the workpiece and set the border coordinates of the workpiece. Move the tool to the start point and press startup, the system will find the center coordinate automatically.



■ Operation :

1. To enter center submenu, press F1 "Positon" => F5 "workpiece Coor" => F5 "Middle Func".
2. To change middle function to auto center, input 1(auto center) in middle function submenu, then press enter, auto center submenu can be displayed as follows.
3. Choose the quadrant to process and input workpiece data, the description of the field to input an follows.
 - ◆ Workpiece length I: the workpiece actual length in X-axis direction
 - ◆ Workpiece width J: the workpiece actual width in Y-axis direction
 - ◆ Safe distance H: this is the length more than the distance between the start point of the tool P2 and workpiece, regardless of X-axis direction or Y-axis direction.
 - ◆ Feed rate: detection rate of auto center.
 - ◆ Z-axis safe height: this is the height that the tool will not encounter the workpiece when it moves above the workpiece, P1 in the figure as follows.

4. Move to Z-axis Safe Height Teach, the location of P1 as displayed in the figure. Press F8 "Z safe height teach", the value of Z-axis machine coordinate will be noted simultaneously as the safe height when tool moves above the workpiece.
5. Move the tool down to P2 under the surface of work piece as the start point of auto center.
6. Press F2"Start", the tool will be moved to touch the workpiece according to the data which has been set and the value will be displayed on the screen. Finally the center coordinate of the workpiece in XY-axis direction can be calculated.
7. Press "Esc" to exit "center function" back to workpiece coordinate submenu.
8. Move the cursor to the location of the workpiece coordinate which will be set in workpiece coordinate submenu. Press F3"AUX Latched" and the system will set the AUX coordinate Pxm or Pym into the workpiece coordinate system according to the axis that the cursor stops.

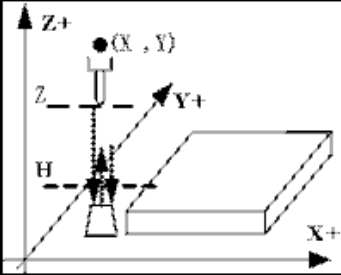
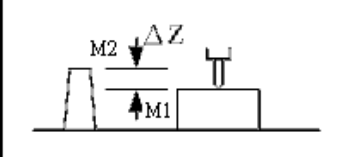
3.12 Z-axis Auto Tool

■ Specification:

Z-axis auto tool function measures the location of tool point of different tool number through the tool calibrator on the machine, then operator makes the distance that the location of the tool point to the datum plane of the workpiece is fixed to amend the processing datum plane. Operator can teach the distance into workpiece coordinate system as the gist of the tool offset when processing.

■ Operation:

After starting, F1"Positon"=>F5"workpiece Coor"=>F6"Auto Tool"

G54 2.NC N0 L1		Position	12:43:46	2009/02/14
Tool Tip Meas 		WorkPiece No. P 0 Feedrate F 0.000 Use Reference 0 Ref Coord. X 0.000 Ref Coord. Y 0.000 Start Coord. Z 0.000 Min. Z Mach. 0.000 Select if use Ref Point 1:Set All measure parameter 2:If not use Ref, Take tool tip to upper of measurement 3:Press F1, Measure Start		
Delta Z Set 		Delta Z 0.000 Do tool tip measure before do Delta 1:Take tool tip to top of good 2:Press F3, Delta Z Set		
		Record X 0.000 Y 0.000 Z 0.000 A 0.000 B 0.000 Relative X 0.000 Y 0.000 Z 0.000 A 0.000 B 0.000 Aux. Coordinate X 0.000 Y 0.000 Z 0.000		
0:Aux, 1:G54, 2:G55, 3:G56, 4:G57, 5:G		● READY AUTO ALARM		
Measure Start	Measure FeedHold	Delta Z Set	XY Ref. Teach	Mach. Z Teach

■ Auto tool length measurement

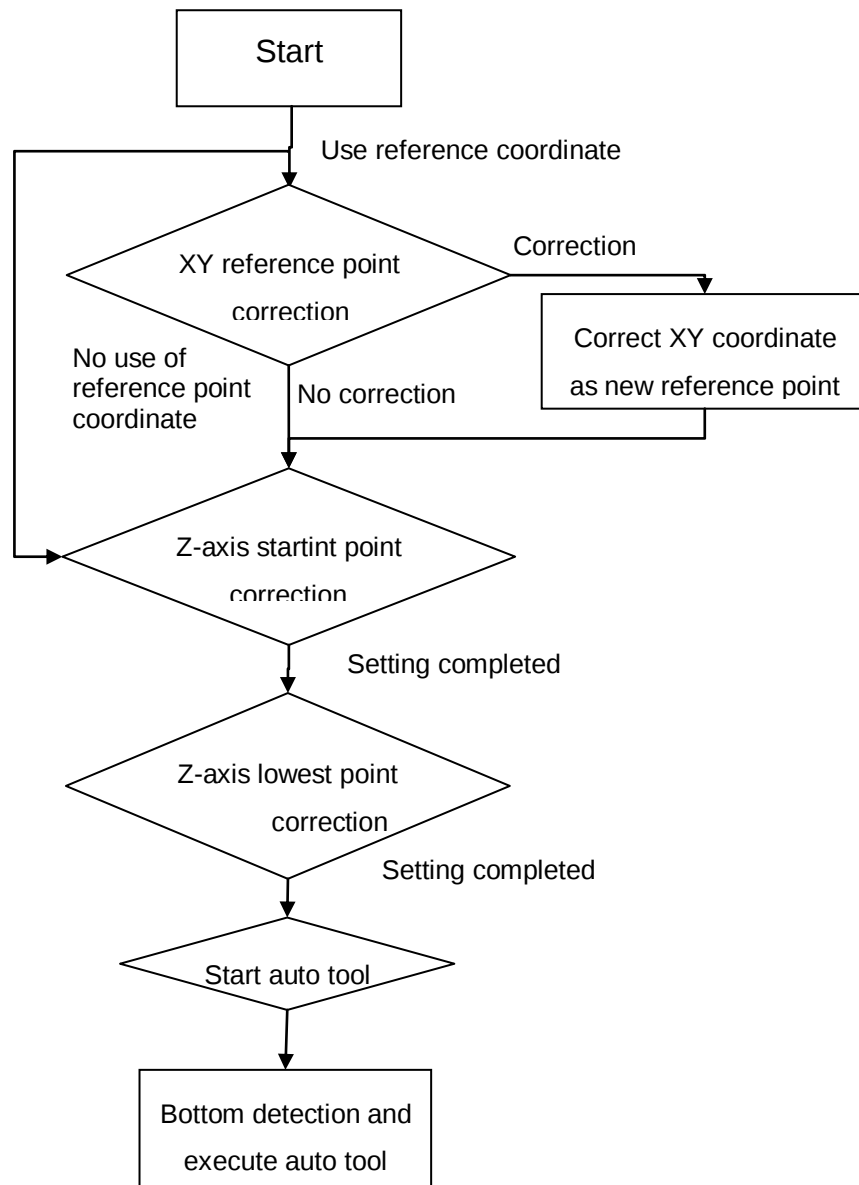
It needs to measure the length of the tool again when change another tool in order to compensate the correct tool length to the processing path.

1. Coordinate workpiece number P:0: auxiliary point,1: 1:G54, 2:G55, 3:G56, 4:G57, 5:G58, 6:G59, 7:G59.1 ... ;
2. Measurement rate F: set auto tool bottom detection of the first time and rebounding speed of every time.
3. Use reference point coordinate: set whether to re-move to the reference point and then auto tool or not.

“0”→ the current location ,as the start point of auto tool and directly do bottom detection.

”1”→The system will first complete the following steps and then do auto tool.

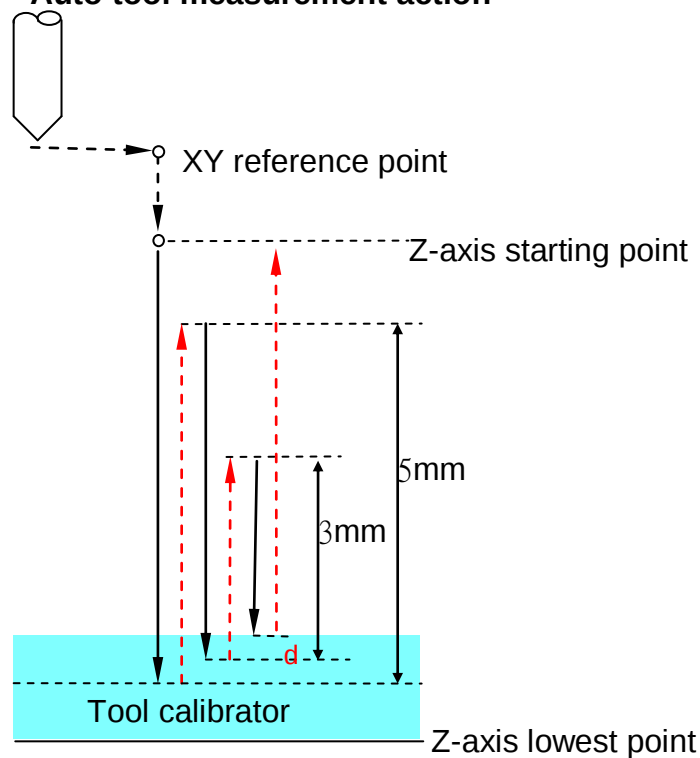
- I. Z-home
 - II. XY-axis move to the location of the second reference point
 - III. Z-axis down to the start point of auto tool.
4. Reference point X in X-axis direction: this field can first move spindle to new auto tool position coordinate and then through 【XY machine coordinate teach】 to correct the location of the auto tool reference point in X-axis(if no correction system, the previous value will be set as the preset value);
 5. Reference point Y in Y-axis direction: this field can first move spindle to new auto tool position coordinate and then through 【XY machine coordinate teach】 to correct the location of the auto tool reference point in Y-axis(if no correction system, the previous value will be set as the preset value);
 6. Start point Z in Z-axis direction: this field can first move spindle to the new location of auto tool start point in Z-axis and then through 【Z machine coordinate teach】 to set Z-axis start point of auto tool ;
 7. Z-axis lowest machine coordinate H: the height to down Z-axis that the spindle will not hit the tool calibrator, press function key 【Z machine coordinate teach】 to set the value;



■ Delta z set:

Delta z set is mainly used to measure the distance between the surface of the workpiece and the tool calibrator when change another workpiece and the system will note the value to external shift. The value can automatically combine and calculate with the tool length measured to produce the correct path.

1. Delta z value: display the height delta value between the surface of the workpiece and the tool calibrator, correspond to Z-axis set value in external shift.
2. Operation: first do tool length measurement then set delta
Step 1: take tool tip to top of workpiece
Step 2: press F3, **【delta z set】**

Auto tool measurement action

d: the value of error

Tool calibrator action specification:

1. First move the spindle to XY-axis tool calibrator reference point
2. Z-axis moves to start point at 7000mm/min
3. Z-axis moves to **【Z-axis min coordinate】** that the operator input at measuring speed and stops immediately while touching tool calibrator.(if don't touch the tool calibrator after arriving at the lowest coordinate, there will be an alarm[Z min coordinate set error alarm] immediately)
4. Back Z-axis 5mm at the speed measured
5. Z-axis moves down 6mm at 50mm/min and will stop moving and note Z-axis machine coordinate if touch the tool calibrator in the process.(if don't touch the tool calibrator after moving 6mm, there will be an alarm[Auto tool length measurement error alarm] immediately)
6. Back Z-axis 3mm at the speed measured
7. Z-axis moves down 4mm at 50mm/min and will stop moving and note Z-axis machine coordinate if touch the tool calibrator in the process.(if don't touch the tool calibrator after moving 4mm, there will be an alarm[Auto tool length measurement error alarm] immediately)
8. Judge if the error noted in 0.01mm,input the min unit integer into Z-axis coordinate that operator chooses after averaging; if the error exceeds 0.01mm,there will be an alarm [Auto tool length measurement error alarm] immediately
9. G90 G53 Z0. back tool to Z-axis original point of machinery.

3.13 RS232 FUNCTION

DNC SOFTWARE ⇔ CONTROLLER (SYNTEC NC CONTROLLER SOFTWARE)

■ DESCRIPTION :

Users can use software of CAD/CAM transfer data to controller by RS232 cable. Let user can select be transferred NC file to working.

■ Operation :

1. Step I : The hardware line link to COM1
2. Step II : Parameter of SYNTEC software system setting
(Suggest value of default)

Param.	Controller	Meaning of parameter
3901	0	Dc protocol role, 0 : CNC, 1 : Device or PC
3903	1	File transfer port number(1:Com1, 2:Com2)
3921	2	Com1 baud(0:24;1:48;2:96;3:192;4:384..)
3922	8	Com1 data bit number
3923	0	Com1 exchange code type (0:ASCII;1:EIA;2:ISO)
3924	0	Com1 control code (0:No;1:DC2;2:DC4;3:DC2DC4)
3925	1	Com1 End-of-block output code(0:EOB;1:CR+EOB)
3926	0	Com1 DC3 control code parity (0:Off;1:On)
3927	2	Com1 flow control(0:No;1:CrsRts;2:XonXoff)
3928	1	Com1 parity check(0:No;1:Odd;2:Even)
3929	1	Com1 stop bit number(1:1 bit;2:2 bit)

3. Step III : Turn on the controller power. The RS232 file input is ready.

Operation has been stated as follows.

- I. Press [program] [file manage] to file manage submenu.
And than to press [=>] show up as follow:

01010 N0 L1	Program	18:05:06	2001/02/01
Program storage free space 2147155968 bytes			
00001	60 02/01/01 10:12 am G00X50.Y50.Z50.		
00100	958 02/01/01 10:12 am OUT SIDE ROUGH CUTTING 800 R.P.M ,F0.25		
00109	451 02/01/01 10:12 am T11;		
00200	989 02/01/01 10:12 am G92 S2000;		
00300	863 02/01/01 10:12 am G73/G74		
00400	851 02/01/01 10:12 am G72/G73/G74		
00500	989 02/01/01 10:12 am G92 S2000;		
01000	358 02/01/01 10:12 am 車床測試程式 FOR G73		
01010	298 02/01/01 10:12 am 車床測試程式 FOR G73		
01020	308 02/01/01 10:12 am 車床測試程式 FOR G74		
01030	272 02/01/01 10:12 am 車床測試程式 FOR G75		
01040	856 02/01/01 10:12 am G72/G73/G74		
01400	848 02/01/01 10:12 am G72/G73/G74		
01600	275 02/01/01 10:12 am G72/G73		
01601	242 02/01/01 10:12 am ,C,R,A demo program		
01602	419 02/01/01 10:12 am G00 X50.;		
01603	201 02/01/01 10:12 am G00 X50.;		
01604	439 02/01/01 10:12 am M3 S3000 M8;		
01605	376 02/01/01 10:12 am G92 S2000;		
01606	457 02/01/01 10:12 am		
01607	567 02/01/01 10:12 am		
		READY	AUTO
		ALARM	
New File	Copy File	Delete File	Import
Export	RS232 Import	RS232 Export	

- II. press [RS232 Import].show up as follow :

Import file, RS232, to: 00002

- III. After input the file name, Press [Ok] . Then the program will wait for receive state. The transfer state will has shown on transfer process.

File Import

Import RS232 to 00002

Total received bytes :

4. Step IV : Executing DNC software. Does the RS232 transfer setting adjust is the same controller, or controller adjust is the same of DNC software. Executing the send function of DNC software. It can be transferred into which the file of output.

■ Attention :

When operating. The file input should be doing firstly. Let controller stay on the receive file state. And then to set up transfer file function of DNC software. It can be sure transfer correctly.

3.14 DNC Function

DNC SOFTWARE ⇔ CONTROLLER (SYNTEC NC CONTROLLER SOFTWARE)

■ DESCRIPTION :

When user is use the CAD/CAM software. The controller doesn't capacity in saving the too large size program in working. Use this function to execute that aside transfer and aside working.

■ Operation :

1. Step I : The hardware line link to COM1
2. Step II : Parameter of SYNTEC software system setting
(Suggest value of default)

Param.	Controller	Meaning of parameter
3901	0	Dc protocol role, 0 : CNC, 1 : Device or PC
3903	1	File transfer port number(1:Com1, 2:Com2)
3921	2	Com1 baud(0:24;1:48;2:96;3:192;4:384..)
3922	8	Com1 data bit number
3923	0	Com1 exchange code type (0:ASCII;1:EIA;2:ISO)
3924	0	Com1 control code (0:No;1:DC2;2:DC4;3:DC2DC4)
3925	1	Com1 End-of-block output code(0:EOB;1:CR+EOB)
3926	0	Com1 DC3 control code parity (0:Off;1:On)
3927	2	Com1 flow control(0:No;1:CrsRts;2:XonXoff)
3928	1	Com1 parity check(0:No;1:Odd;2:Even)
3929	1	Com1 stop bit number(1:1 bit;2:2 bit)

3. Step III : Executing DNC software. Does the RS232 transfer setting adjust is the same of controller, or controller adjust is the same of the DNC software. Executing the send function of DNC software. It can be transferred into the file of output.
4. Step IV : Turn on controller power. Into system by press [program] [file manage] to file manage submenu. Select the RS232 DNC Program is a working file. It is shown as follows.

01000 N0 L1		Program		13:32:43	2001/02/01	
Program storage free space 2147155968 bytes						
00001	51790	01/18/01	04:14 pm	G00X50.Y50.Z50.		
01000	259	02/01/01	10:24 am	銑床測試程式 FOR G73		
01010	265	02/01/01	01:09 am	銑床測試程式 FOR G74		
RS232 DNC Program						
				● READY	AUTO	ALARM
New File	Copy File	Delete File	Import	Export	RS232 Import	RS232 Export

5. Step V : press “monitor”. Controller will read into file to working by RS232

■ Attention :

When operating. It should be doing that the working file from DNC software output wait controller to reading firstly. And select RS232 DNC Program is working file on controller. Finish, Setup working. It can be sure transfer correctly.

3.15 The software of SYNTEC controller replace the software of DNC to execute RS232 function

PC(SYNTEC software end of PC) ⇔ controller

■ **DESCRIPTION :**

The SYNTEC controller software made user transfer into controller working by RS232 that CAD/CAM generate or writing file. But the SYNTEC controller software can't offer the DNC software that aside transfer and aside working function.

■ **Operation :**

1. Step I : The hardware line link to COM1
2. Step II : Parameter of SYNTEC software system setting
(suggest setting value. Exception of 3910 other all the same)

Param.	Controller	Meaning of parameter
3901	0	Dc protocol role, 0 : CNC, 1 : Device or PC
3903	1	File transfer port number(1:Com1, 2:Com2)
3921	2	Com1 baud(0:24;1:48;2:96;3:192;4:384..)
3922	8	Com1 data bit number
3923	0	Com1 exchange code type (0:ASCII;1:EIA;2:ISO)
3924	0	Com1 control code (0:No;1:DC2;2:DC4;3:DC2DC4)
3925	1	Com1 End-of-block output code(0:EOB;1:CR+EOB)
3926	0	Com1 DC3 control code parity (0:Off;1:On)
3927	2	Com1 flow control(0:No;1:CrsRts;2:XonXoff)
3928	1	Com1 parity check(0:No;1:Odd;2:Even)
3929	1	Com1 stop bit number(1:1 bit;2:2 bit)

3. Step III : Turn on the controller power. The RS232 file input is ready. Operation has been stated as follows.

Operation has been stated as follows.

- I. Press [program] [file manage] to file manage submenu,
And than to press [=>]show up as follow :

01010 N0 L1		Program		18:05:06		2001/02/01	
Program storage free space 2147155968 bytes							
00001	68	02/01/01	10:12	am	G00X50.Y50.Z50.		
00100	950	02/01/01	10:12	am	OUT SIDE ROUGH CUTTING 800 R.P.M ,F0.25		
00109	451	02/01/01	10:12	am	T11;		
00200	989	02/01/01	10:12	am	G92 S2000;		
00300	863	02/01/01	10:12	am	G73/G74		
00400	851	02/01/01	10:12	am	G72/G73/G74		
00500	989	02/01/01	10:12	am	G92 S2000;		
01000	358	02/01/01	10:12	am	車床測試程式 FOR G73		
01010	298	02/01/01	10:12	am	車床測試程式 FOR G73		
01020	308	02/01/01	10:12	am	車床測試程式 FOR G74		
01030	272	02/01/01	10:12	am	車床測試程式 FOR G75		
01040	856	02/01/01	10:12	am	G72/G73/G74		
01400	848	02/01/01	10:12	am	G72/G73/G74		
01600	275	02/01/01	10:12	am	G72/G73		
01601	242	02/01/01	10:12	am	,C,R,A demo program		
01602	419	02/01/01	10:12	am	G00 X50.;		
01603	201	02/01/01	10:12	am	G00 X50.;		
01604	439	02/01/01	10:12	am	M3 S3000 M8;		
01605	376	02/01/01	10:12	am	G92 S2000;		
01606	457	02/01/01	10:12	am			
01607	567	02/01/01	10:12	am			
				☒ READY		AUTO	
						ALARM	
New File	Copy File	Delete File	Import	Export	RS232 Import	RS232 Export	

- II. press [RS232 Import].Show up as follow :

Import file, RS232, to: 00002

- III. After input the file name. Press [Ok].Then the program will enter into waiting for receive state. The transfer state will be shown on transfer process.

File Import Import RS232 to 00002 Total received bytes :

4. Step IV : Upper End of PC. Executing c:\cnc\cnc into the controller simulation software. Does RS232 file output. Operation has been stated as follows.

- I. Press [program] [file manage] to file manage submenu. Than to press [=>] key to appearing. And press up/down key select that want output of file. Such as O1000.

01010 N0 L1	Program	18:05:06	2001/02/01
Program storage free space 2147155968 bytes			
00001	60 02/01/01 10:12 am G00X50.Y50.Z50.		
00100	958 02/01/01 10:12 am OUT SIDE ROUGH CUTTING 000 R.P.M ,F0.25		
00109	451 02/01/01 10:12 am T11;		
00200	989 02/01/01 10:12 am G92 S2000;		
00300	863 02/01/01 10:12 am G73/G74		
00400	851 02/01/01 10:12 am G72/G73/G74		
00500	989 02/01/01 10:12 am G92 S2000;		
01000	358 02/01/01 10:12 am 車床測試程式 FOR G73		
01010	298 02/01/01 10:12 am 車床測試程式 FOR G73		
01020	308 02/01/01 10:12 am 車床測試程式 FOR G74		
01030	272 02/01/01 10:12 am 車床測試程式 FOR G75		
01040	856 02/01/01 10:12 am G72/G73/G74		
01400	848 02/01/01 10:12 am G72/G73/G74		
01600	275 02/01/01 10:12 am G72/G73		
01601	242 02/01/01 10:12 am ,C,R,A demo program		
01602	419 02/01/01 10:12 am G00 X50.;		
01603	201 02/01/01 10:12 am G00 X50.;		
01604	439 02/01/01 10:12 am M3 S3000 M8;		
01605	376 02/01/01 10:12 am G92 S2000;		
01606	457 02/01/01 10:12 am		
01607	567 02/01/01 10:12 am		

☐ READY
 ☐ AUTO

New File	Copy File	Delete File	Import	Export	RS232 Import	RS232 Export
----------	-----------	-------------	--------	--------	--------------	--------------

- II. press[RS232 Import].show up as follow :

File Export

Exporting, 00001, to RS232?

- III. After press [Ok] .Start transfer program. And appearing state of transfer.

File Export

Export 00001 to RS232

Total sent bytes: 1832

■ Attention :

1. When operating. The file input should be doing firstly. Let controller stay on the receive file state ,and to setup transfer file function of DNC software. It can be sure transfer correctly.
2. 2、 Be careful. SYNTEC controller software can offer RS232 file transfer function. But can't offer DNC software that aside transfer and aside working function.
3. 3、 The most our must be careful. When PC executing. We suggest that add up /M /P parameter behind batch of cnc. Let controller software can execute simulation controller of function on PC. The /M is operate plate of simulation. User can setup working from keyboard. The /P will simulate ISR and cover off Interrupt. Convenient software can execute upper the Windows. **But will be invalid of transfer function if behind batch of cnc add up /P parameter.**