

SYNTEC

Mill Controller Manual **(For EZ Series)**

By: SYNTEC

Date: 2010/9/14

Version: 1.0

Version edit record

No.	Modify Content	date	editor	New Vision
01	Mill Operation Manual-EN FOR EZ（5Keys）	2010/09/14	XuMing	V1.0

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
1.3.2	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	F4 : Simulation.....	11
1.3.2.4.1	F1 : STEP	11
1.3.2.4.2	F2 : Continue.....	11
1.3.2.4.3	F3 : Zoom.....	12
1.3.2.4.4	F4 : Graph reset.....	12
1.3.2.4.5	F5 : Abort	12
1.3.2.4.6	F5 : Simu. Setting.....	12
1.3.2.5	F5 : File Manager.....	13
1.3.2.5.1	F1 : New File.....	14
1.3.2.5.2	F2 : Copy File.....	14
1.3.2.5.3	F3 : Delete File.....	14
1.3.2.5.4	F4 : Import.....	15
1.3.2.5.5	F5 : Export.....	16
1.3.2.5.6	F1 : RS232 Import.....	17
1.3.2.5.7	F2 : RS232 Export.....	19
1.3.2.5.8	F3 : Network Import.....	20

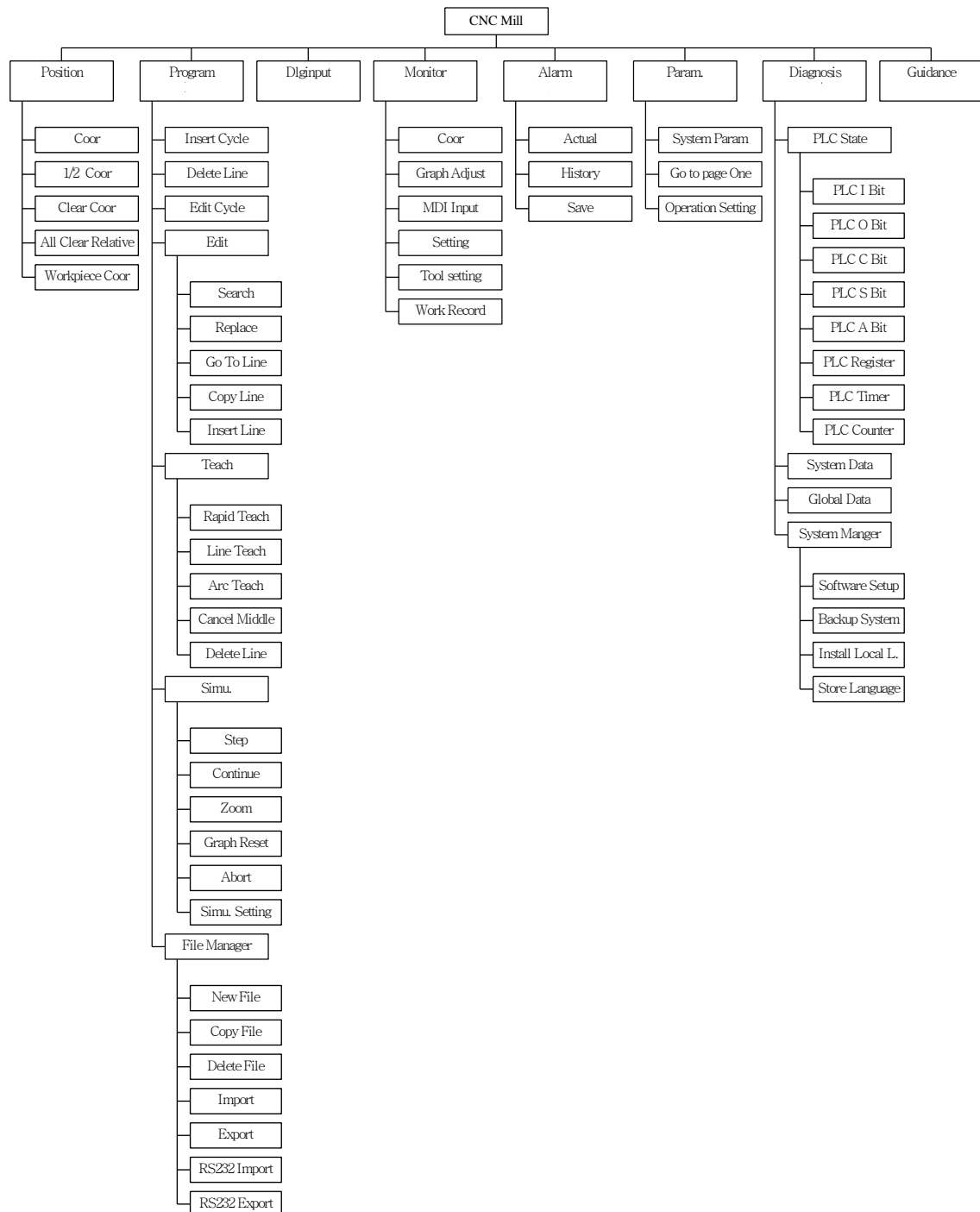
1.3.2.5.9	F4:Multi Import.....	20
1.3.2.5.10	F5:Multi Export.....	20
1.3.2.6	EDIT	20
1.3.2.6.1	EDIT sub function “F1 : Search”	21
1.3.2.6.2	EDIT sub function “F2 : Replace”	22
1.3.2.6.3	EDIT sub function “F3” : Go To Line”	23
1.3.2.6.4	EDIT sub function “F4”: copy line”	23
1.3.2.6.5	EDIT sub function “F5”: Insert line”	23
1.3.2.7	F2 : Teach	24
1.3.2.7.1	F1 : Rapid Teach	24
1.3.2.7.2	F2 : Line Teach	24
1.3.2.7.3	F3 : Arc Teach.....	24
1.3.2.7.4	F4 : Cancel Middle.....	25
1.3.2.7.5	F5 : Radius Teach.....	25
1.3.2.7.6	F1 : Delete Line.....	25
1.3.2.7.7	F2 : Coord Switch	25
1.3.2.7.8	F3 : Point Teach	25
1.3.3	F3 : DlgInput (ONLY for 940M).....	26
1.3.4	F4 : Monitor.....	26
1.3.4.1	F1 : Coord.....	26
1.3.4.2	F2 : Graph Adjust.....	27
1.3.4.2.1	F1 : Zoom.....	27
1.3.4.2.2	F2 : Graph Reset.....	27
1.3.4.2.3	F5 : Simu Setting.....	27
1.3.4.3	F3 : MDI Input	28
1.3.4.4	F4 : SETTING.....	29
1.3.4.5	F5 : Tool Setting	30
1.3.4.6	F5 : Work Record.....	31
1.3.5	F5 : Alarm.....	32
1.3.5.1	F1 : Actual.....	33
1.3.5.2	F2 : History	33
1.3.5.3	F5: Save	33
1.3.6	F1 : Parameter.....	34
1.3.7	F2 : Diagnosis.....	35
1.3.7.1	F1 : PLC State.....	36
1.3.7.2	F2 : System Data	37
1.3.7.3	F3 : Global Variable.....	38
1.3.7.4	F4 : Coord Variable.....	39

1.3.7.5	F5 : System Manager	40
1.3.8	F3 : Guidance	41
2	MACHINE OPERATION PANEL.....	42
2.1	2nd machine operation panel.....	42
2.1.1	POWER ON	42
2.1.2	POWER OFF.....	42
2.1.3	Emergency STOP	42
2.1.4	Home mode and Home function.....	42
2.1.5	Continues JOG (Rapid JOG)	42
2.1.6	Incremental JOG.....	43
2.1.7	MPG JOG	43
2.1.8	AUTO mode NC file execute	43
2.1.9	MDI mode single block execute	43
2.1.10	MPG Simulation.....	44
2.1.11	Dry Run	44
2.1.12	Single block.....	44
2.1.13	Option Stop.....	45
2.1.14	Option Skip.....	45
2.1.15	Spindle control	45
2.1.16	Working led.....	46
2.1.17	Working Liquid	46
2.1.18	Aux table backward.....	46
2.1.19	Aux table forward.....	46
2.2	Text key description:	47
3	HOW TO OPERATE SYNTEC 900ME.....	49
3.1	Manual function (JOG, INC_JOG, MPG).....	50
3.2	HOME.....	51
3.3	Open a file (EDIT / FLOPPY / RS232)	52
3.4	Tool setting (G40/G41/G42, G43/G44/G49)	53
3.5	Tool Length measurement (G43/G44/G49).....	54
3.6	Setting the Workpiece origin offset value (G54...G59).....	55

3.7	Manual Data Input (MDI).....	56
3.8	Assigned an executing NC file (AUTO).....	57
3.9	Graphic Simulation.....	58
3.10	How to check NC file in SYNTEC controller.....	59
3.11	Auto Center.....	60
3.11.1	Manual Center	60
3.11.2	Auto center operation	62
3.12	Z-axis Auto Tool.....	64
3.13	RS232 FUNCTION	69
3.14	DNC Function.....	71
3.15	The software of SYNTEC controller replace the software of DNC to execute RS232 function73	

1 CNC Milling Controller Interface

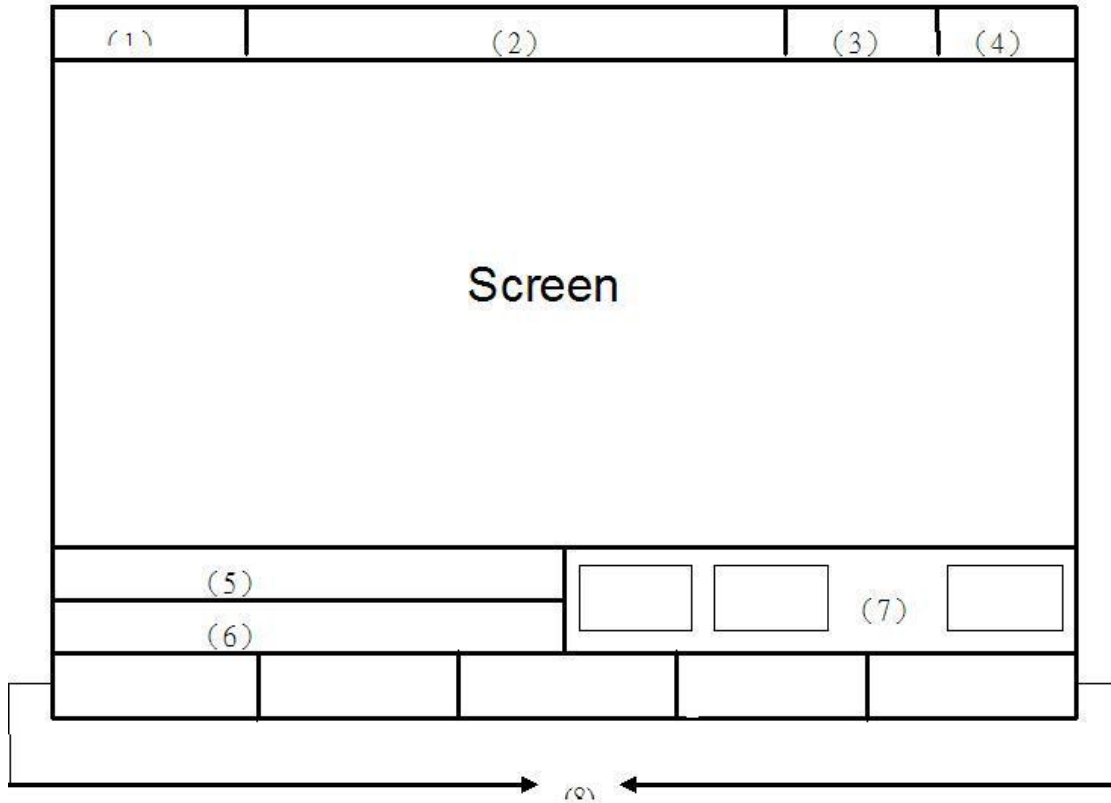
1.1 CNC System Configuration



1.2 Screen Sections

The screen of controller is shown as followings:

Start

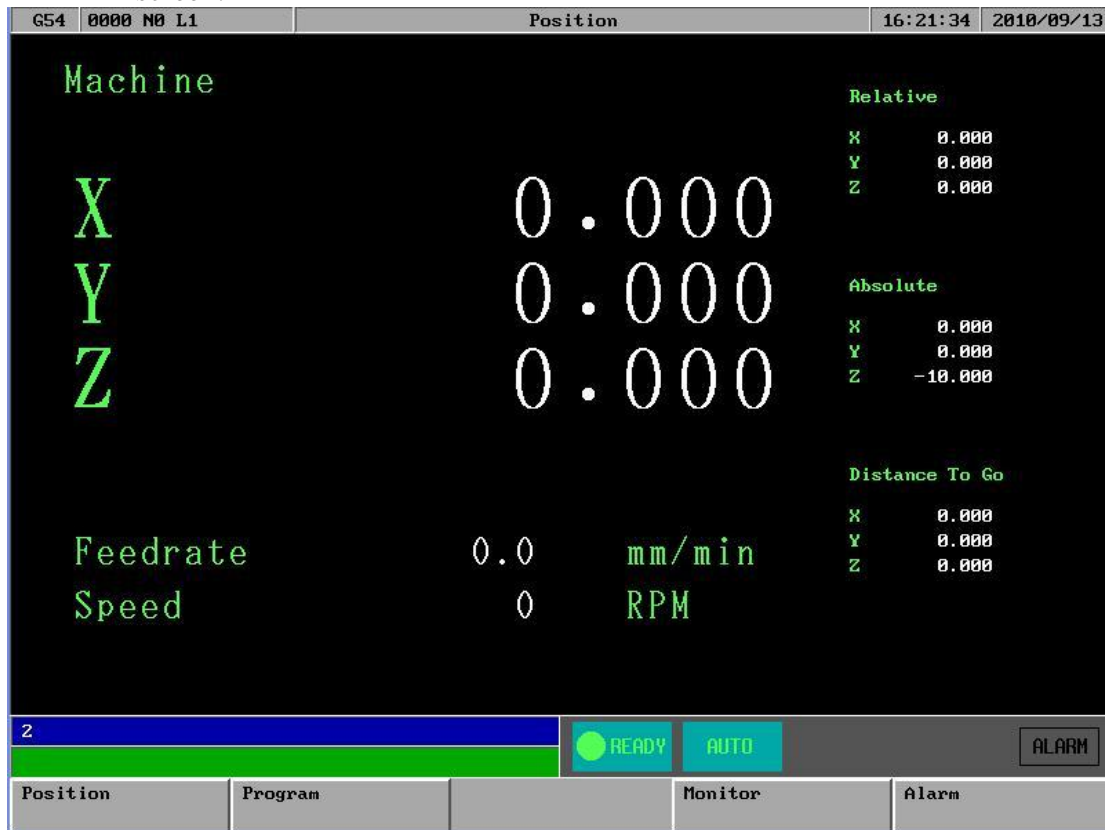


Meanings For Fields on the Display:

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

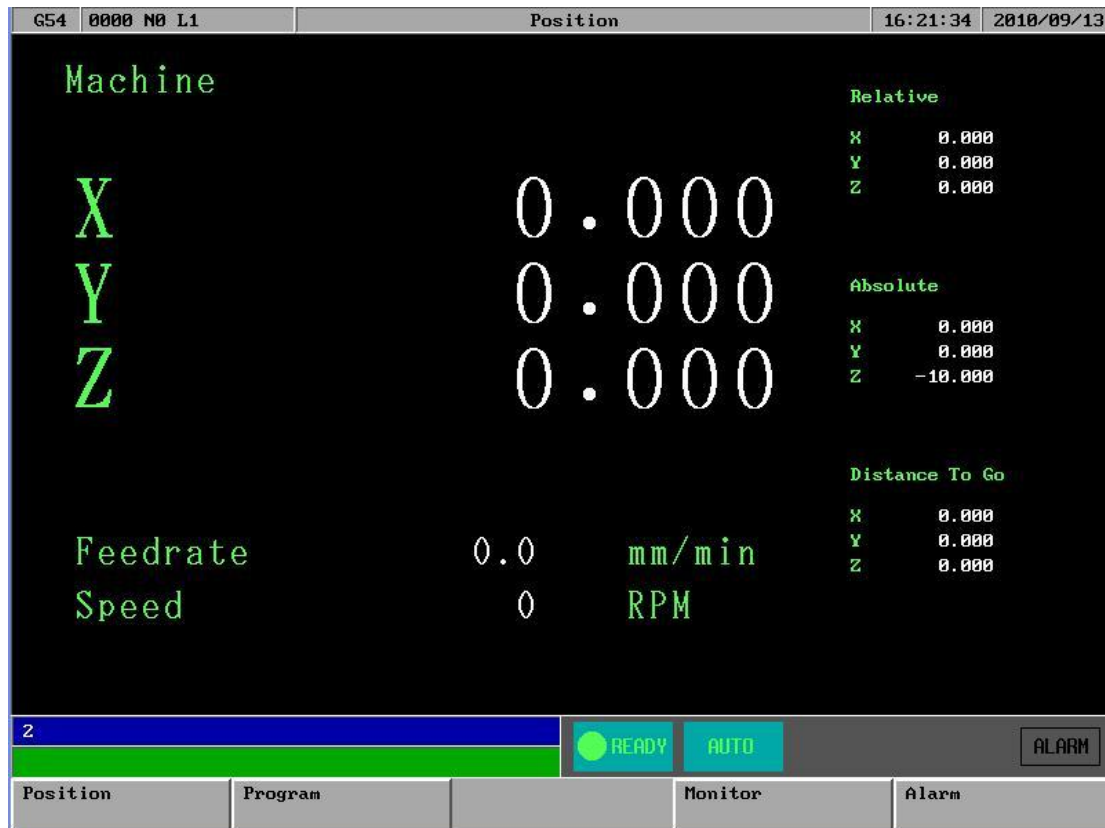
1.3 Main Menu Selections

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~F5 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.

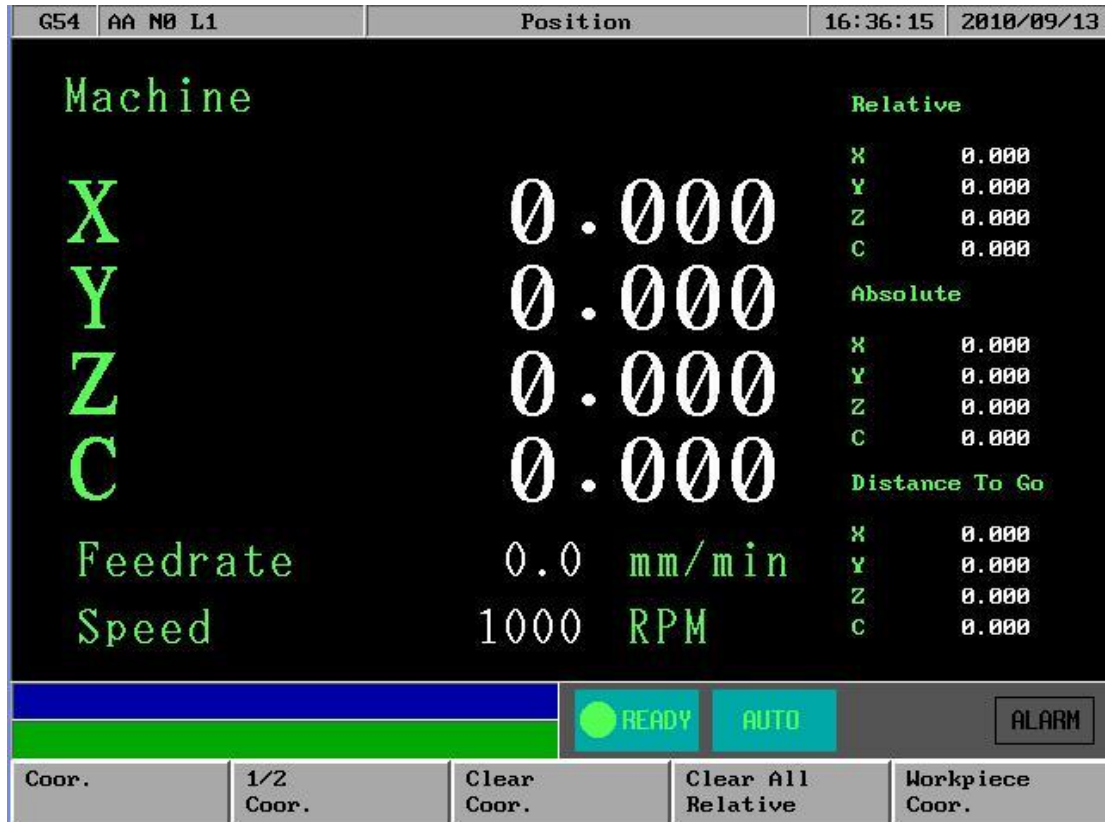
Distance To Go: The distance of the cutting tool that need to move to the next position show on position(+) and negative(-) direction.

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.

1.3.1.5.1 F1 : Coord. Latched

Set the current machine coordinate to the input box

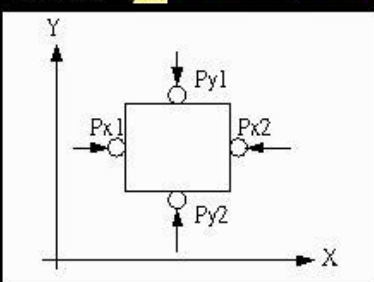
1.3.1.5.2 F2 : Rel. Latched

Set the current relative coordinate to the input box

1.3.1.5.3 F3 : Aux. Latched

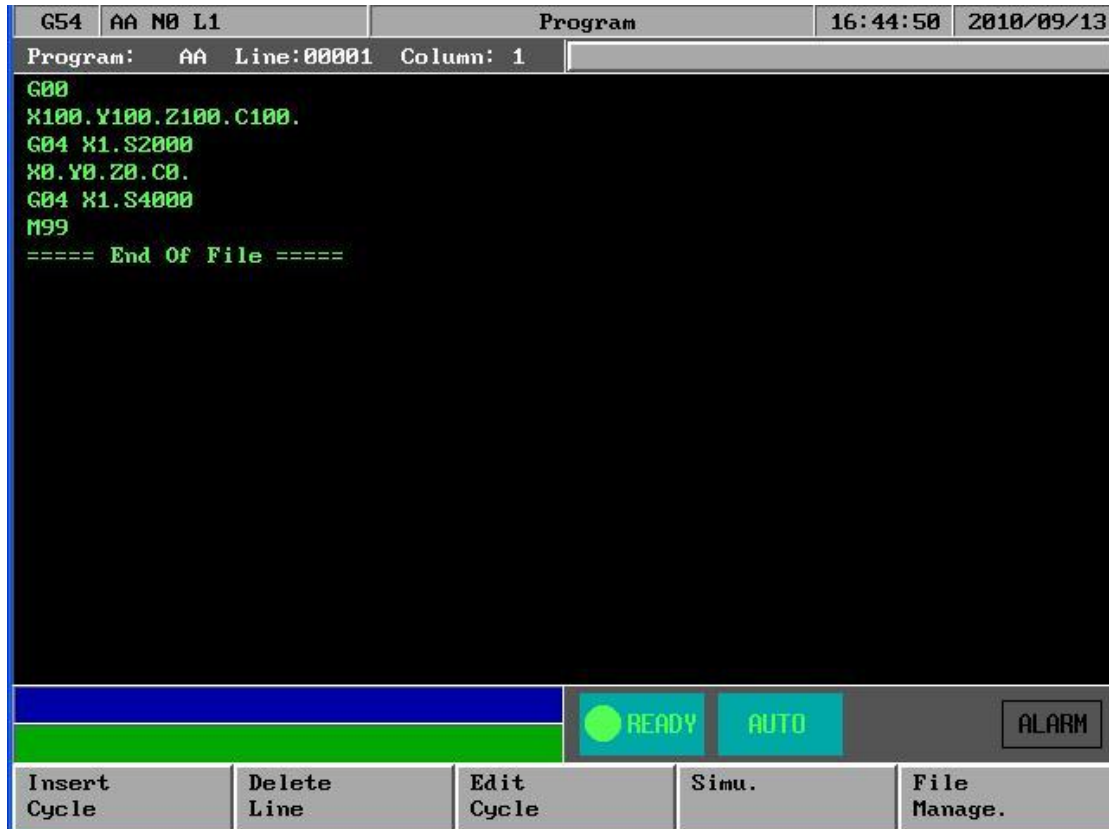
Set the current relative coordinate to the input box

1.3.1.5.4 F5 : Middle Point Func.

G54	N0 L1	Position	11:34:19	2010/09/14
<div style="display: flex; justify-content: space-between;"> <div style="width: 30%;"> <p>Manual 0 (0:Manual, 1:Auto)</p>  <p>Manual Steps:</p> <ol style="list-style-type: none"> 1:Take tool to Px1, Do Px1 Set 2:Take tool to Px2, Do Px2 Set 3:Take tool to Py1, Do Py1 Set 4:Take tool to Py2, Do Py2 Set 5:Back to Workpiece Screen and do Aux. Latched </div> <div style="width: 40%;"> <p>Machine Coord.</p> <p>Px1 0.000</p> <p>Px2 0.000</p> <p>Pxm 0.000</p> <p>Py1 0.000</p> <p>Py2 0.000</p> <p>Pym 0.000</p> </div> <div style="width: 30%;"> <p>Record</p> <p>X 0.000</p> <p>Y 0.000</p> <p>Z 0.000</p> <p>C 0.000</p> <p>Relative</p> <p>X 0.000</p> <p>Y 0.000</p> <p>Z 0.000</p> <p>C 0.000</p> <p>Aux. Coordinate</p> <p>X 0.000</p> <p>Y 0.000</p> <p>Z 0.000</p> </div> </div>				
		NOTREADY		UNSELECT
ALARM				
PX1 Set	PX2 Set	PY1 Set	PY2 Set	

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

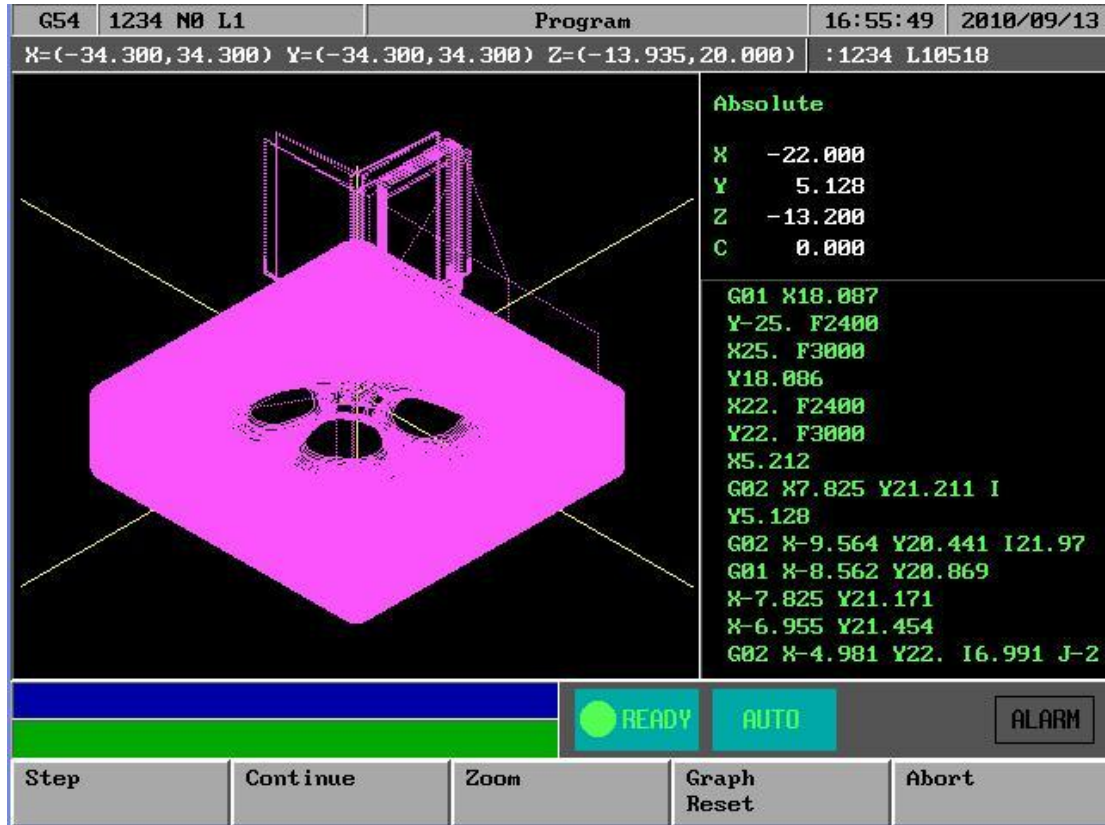
Operation: Under Program submenu, press F3 to edit a line where the cursor is located.

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

1.3.2.4 F4 : Simulation

Function: Simulating the workpiece program can prove the accuracy of the editing program.

Operation : Under Program submenu, press F4



F4 “simulation” sub Key Selections :

1.3.2.4.1 F1 : STEP

Function: To simulate NC files STEP by STEP

Operation: Under Program submenu, press F4”Simulation” and then F1”Step” .The operator can use this function ,to check NC file STEP by STEP

1.3.2.4.2 F2 : Continue

Function: To simulation NC file one time .

Operation: Under Program submenu, press F4 “Simulation” and then F2”Continue” .The operator can use this function to check NC file whole picture ,when push button.

1.3.2.4.3 F3 : Zoom

Function: To enlarge the workpiece graph.

Operation: Under Program submenu, press F5 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 .

1.3.2.4.4 F4 : Graph reset

Function: To recover the zoomed workpiece graph.

1.3.2.4.5 F5 : Abort

Function: To abort simulation action

1.3.2.4.6 F5 : Simu. Setting

Function: To set simulation parameter

Operation: Under Program submenu, press  and then F5.



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 .

1.3.2.5 F5 : 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



Key Selections :

1.3.2.5.1 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.

1.3.2.5.2 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.

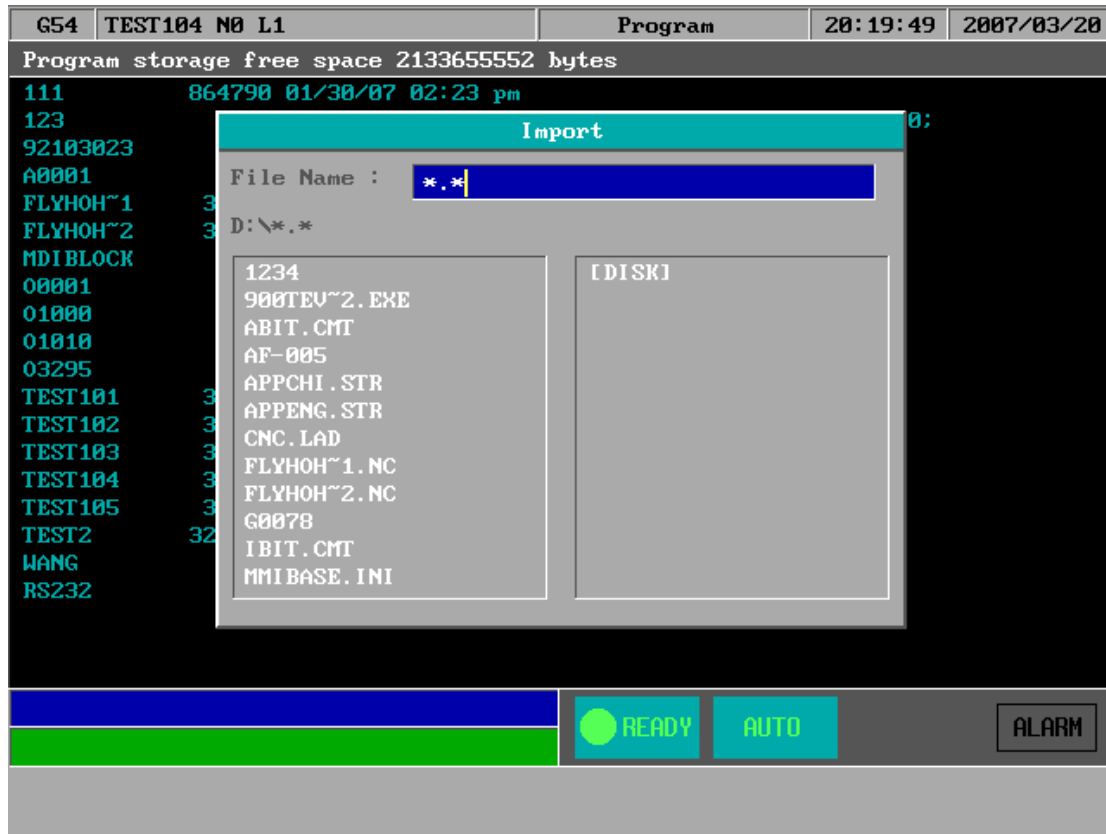
1.3.2.5.3 F3 : Delete File

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

1.3.2.5.4 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



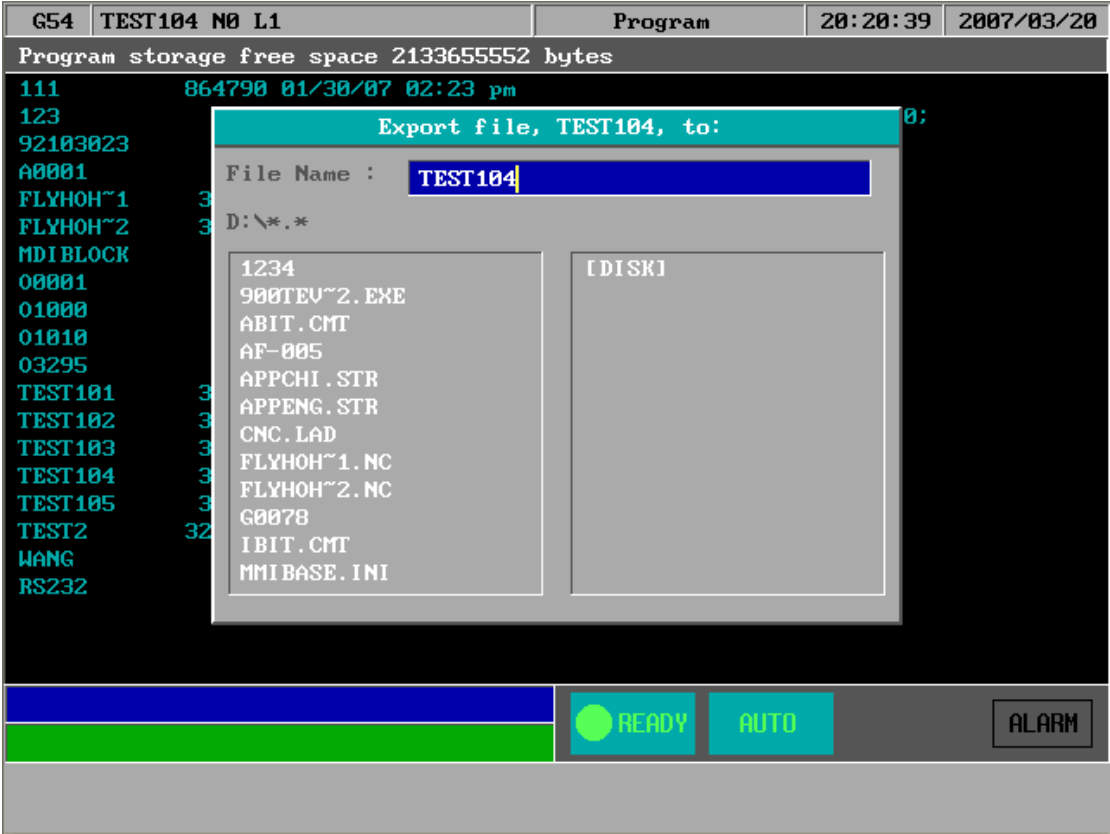
P.S.

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 .

1.3.2.5.5 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】 .



1.3.2.5.6 F1 : RS232 Import

Function: use RS232 communication function to import NC files

Operation: Under Program submenu, press  and then F1. Follow the prompts in the dialog box and type in needed data.

G54	1234 N0 L1	Program	17:59:21	2010/09/13
Program storage free space 110540800 bytes				
1234	241629	03/22/07 11:17 pm		
4-06R0~1	2	11/03/06 10:23 am		
AA	74	11/02/06 01:19 pm	G00	
FF	7	08/25/05 05:03 pm	Y100.	
00001	68	12/02/05 02:25 pm	G00X50.Y50.Z50.	
01000	258	12/02/05 02:25 pm	銑床測試程式 FOR G73	
01010	263	12/02/05 02:25 pm	銑床測試程式 FOR G74	
TEST	2	11/03/06 09:30 am		
RS232	DNC Program			
		 READY	AUTO	ALARM
RS232 Import	RS232 Export	Network Import	Multi Import	Multi Export

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		
FLYH0H~1	31533	03/06/07 06:26 pm		
FLYH0H~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 08:35 pm	00001	
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:

● READY

AUTO

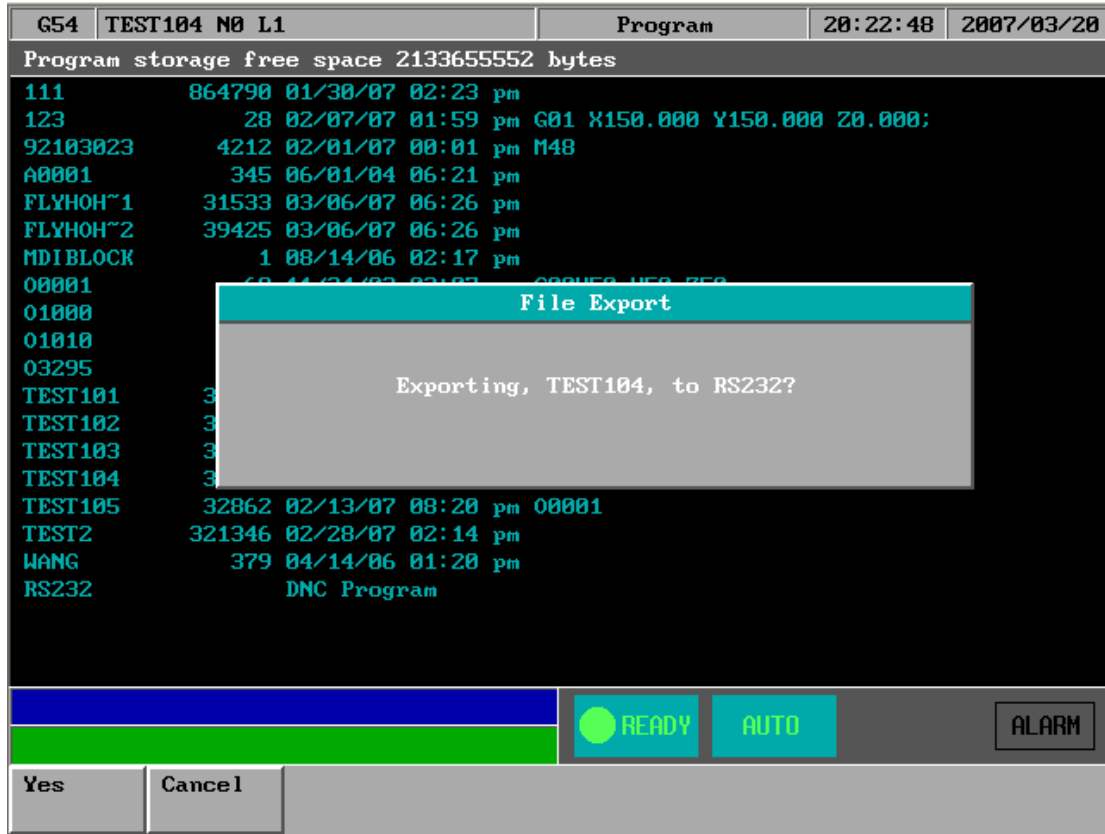
ALARM

OK

1.3.2.5.7 F2 :RS232 Export

Function: Use RS232 communication function to Export NC files

Operation: Under Program submenu, press  and then F2.



1.3.2.5.8 F3:Network Import

Function: Use Net communication function to Import NC files

Operation: Under Program submenu, press  and then F3.


1.3.2.5.9 F4:Multi Import

Function: Import multi NC files once time

Operation: Under Program submenu, press  and then F4.

1.3.2.5.10 F5:Multi Export

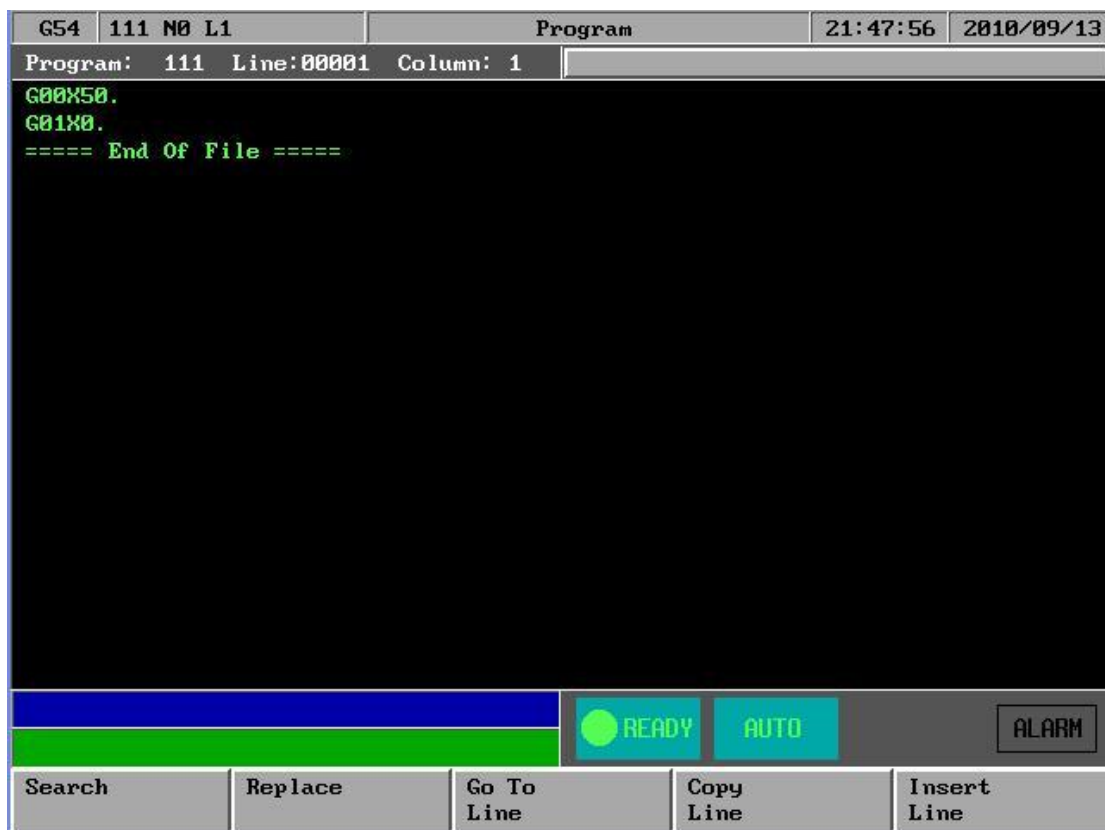
Function: Export multi NC files once time

Operation: Under Program submenu, press  and then F5.

1.3.2.6 EDIT

Function : Edit sub function “ Search “ “Replace” “Goto line” “Copy line” ”Insert line”

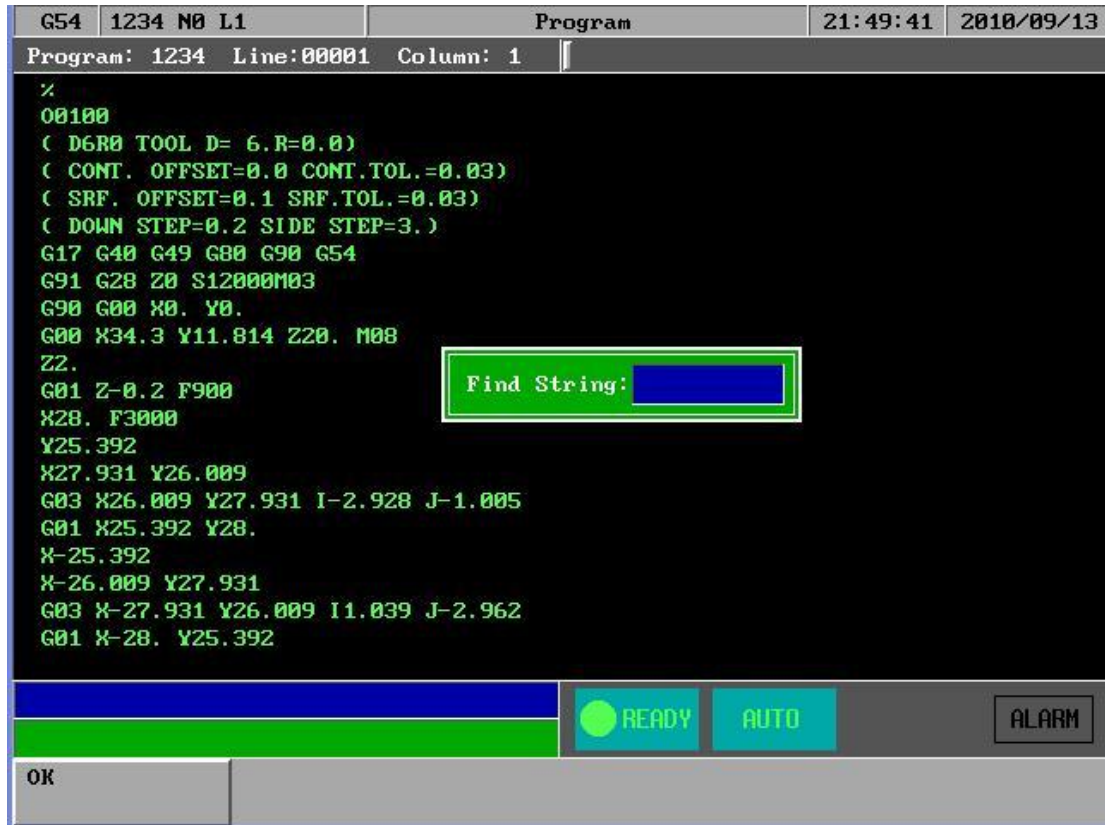
Operation : Under Program submenu, press  and then F1. User can use this menu for more edit sub function



1.3.2.6.1 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.



1.3.2.6.2 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.

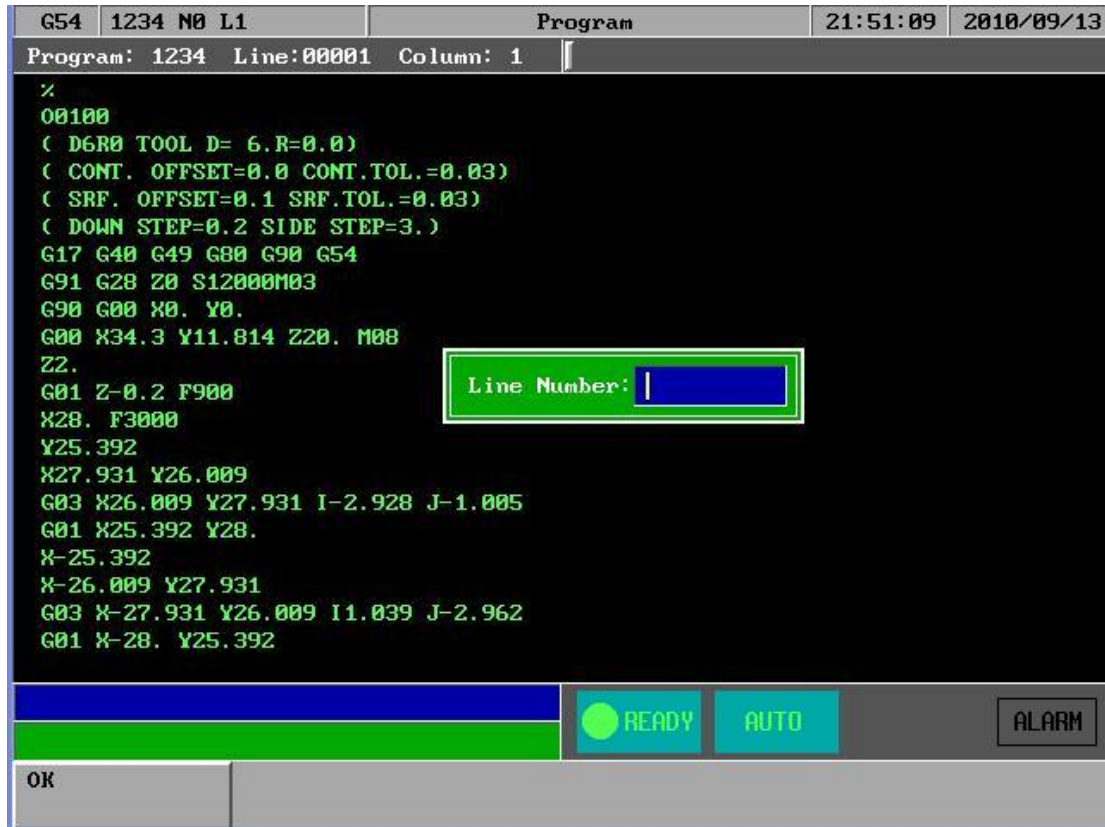
G54	1234 N0 L1	Program	21:50:23	2010/09/13
Program: 1234 Line:00001 Column: 1				
<pre> % 00100 (D6R0 TOOL D= 6.R=0.0) (CONT. OFFSET=0.0 CONT.TOL.=0.03) (SRF. OFFSET=0.1 SRF.TOL.=0.03) (DOWN STEP=0.2 SIDE STEP=3.) G17 G40 G49 G80 G90 G54 G91 G28 Z0 S12000M0 G90 G00 X0. Y0. G00 X34.3 Y11.814 Z Z2. G01 Z-0.2 F900 X28. F3000 Y25.392 X27.931 Y26.009 G03 X26.009 Y27.931 I-2.928 J-1.005 G01 X25.392 Y28. X-25.392 X-26.009 Y27.931 G03 X-27.931 Y26.009 I1.039 J-2.962 G01 X-28. Y25.392 </pre>				
		Find What:	<input type="text"/>	
		Replace With:	<input type="text"/>	
		READY	AUTO	ALARM
Replace	Skip	Cancel		

1.3.2.6.3 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.



1.3.2.6.4 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.

1.3.2.6.5 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

1.3.2.7 F2 : Teach

Function: Teach present absolute coordinate to NC files

Operation : Under Program submenu, press  and then F2.

G54	1234 N0 L1	Program	21:53:59	2010/09/13
Program: 1234 Line:00001 Column: 1				
<pre> % 00100 (D6R0 TOOL D= 6.R=0.0) (CONT. OFFSET=0.0 CONT.TOL.=0.03) (SRF. OFFSET=0.1 SRF.TOL.=0.03) (DOWN STEP=0.2 SIDE STEP=3.) G17 G40 G49 G80 G90 G54 G91 G28 Z0 S12000M03 G90 G00 X0. Y0. G00 X34.3 Y11.814 Z20. M08 Z2. G01 Z-0.2 F900 X28. F3000 Y25.392 X27.931 Y26.009 G03 X26.009 Y27.931 I-2.928 J-1.005 G01 X25.392 Y28. X-25.392 X-26.009 Y27.931 G03 X-27.931 Y26.009 I1.039 J-2.962 G01 X-28. Y25.392 </pre>		<p>Absolute</p> <p>X 0.000</p> <p>Y 0.000</p> <p>Z 0.000</p> <p>C 0.000</p> <p>Arc Middle Point</p> <p>X Coord <input type="text"/></p> <p>Y Coord <input type="text"/></p>		
		<p>READY AUTO ALARM</p>		
Rapid Teach	Line Teach	Arc Teach	Cancel Middle	Radius Teach

Teach sub Key Selections :

1.3.2.7.1 F1 : Rapid Teach

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

1.3.2.7.2 F2 : Line Teach

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

1.3.2.7.3 F3 : Arc Teach

Function: Add “G02” or “G03” code to NC files

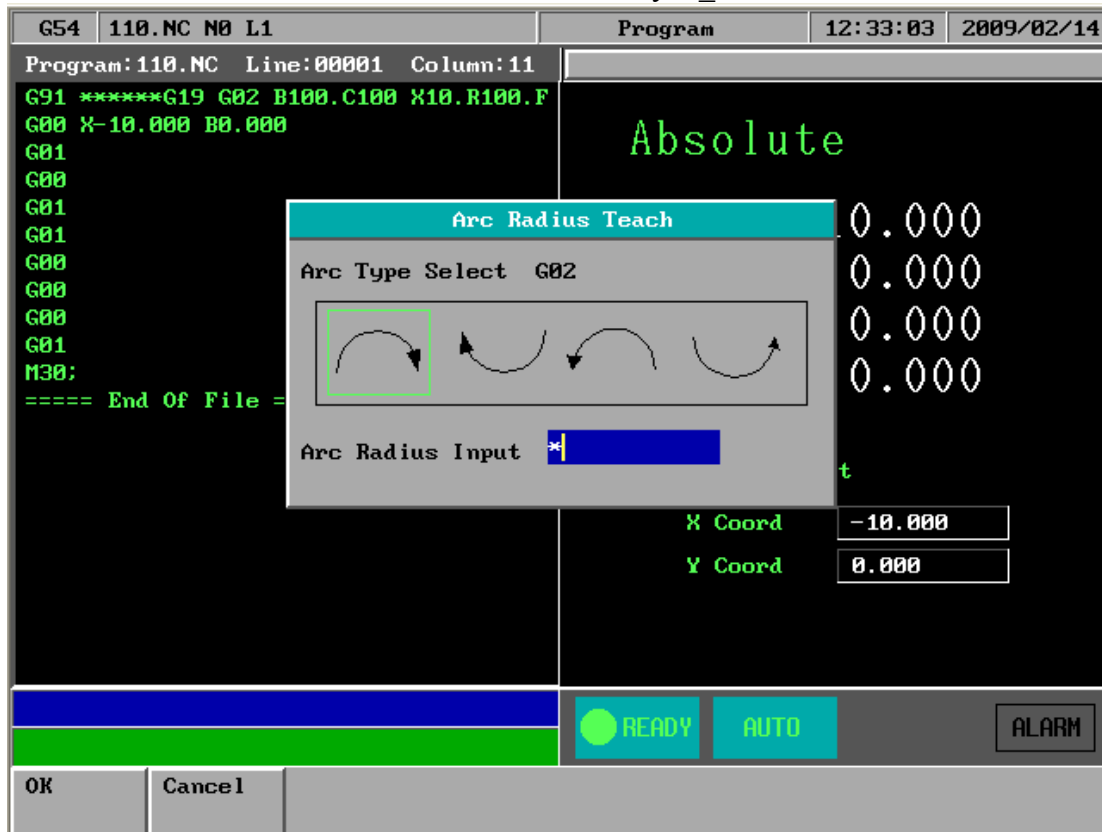
- 1st time press this key “arc teach “ ,CNC auto put current value to Arc middle point
- 2nd time press this key “arc teach “ ,CNC auto calculate G02 or G03 ,and filled the complete code to NC files

1.3.2.7.4 F4 : Cancel Middle

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

1.3.2.7.5 F5 : Radius Teach

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




1.3.2.7.6 F1 : Delete Line

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

Operation : Under Teach press  and then F1.


1.3.2.7.7 F2 : 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.

Operation : Under Teach press  and then F2.

1.3.2.7.8 F3 : Point Teach

Function: Add current absolute coordinate to NC files.

Operation : Under Teach press  and then F3.

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

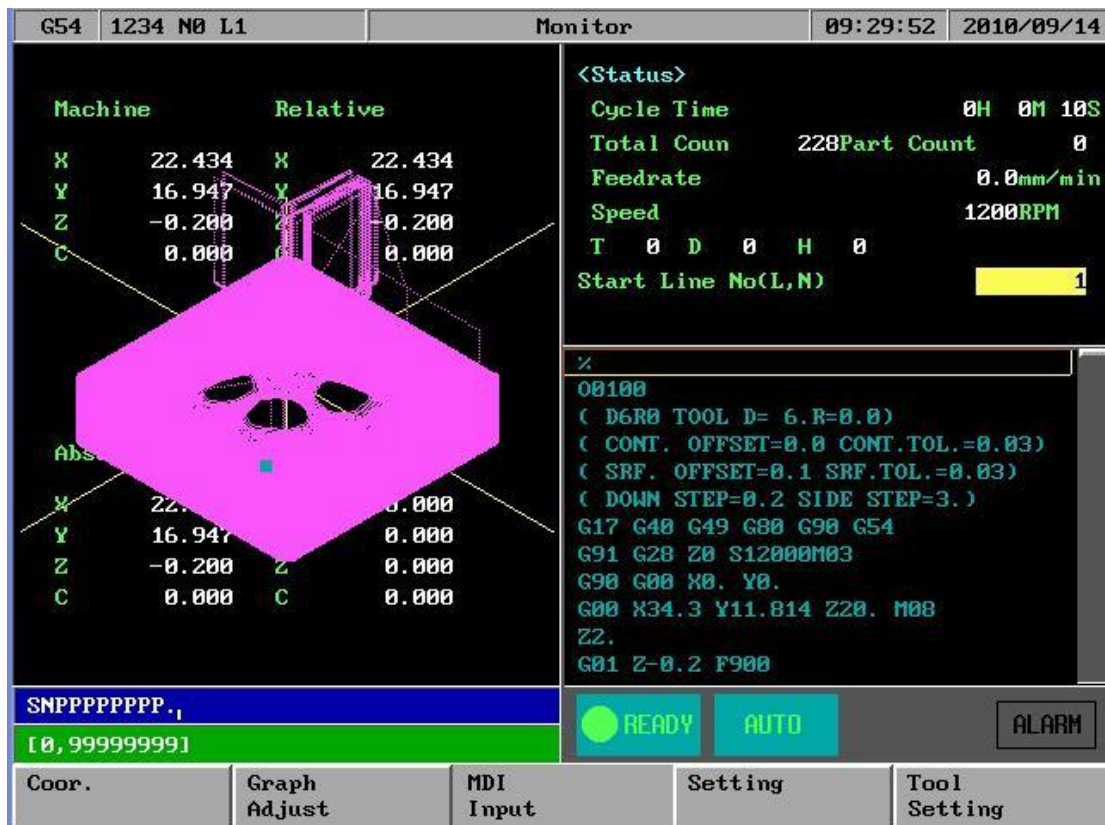
This selection displays machining speed, time, manual data input (MDI) and some machine information such as coordinate, range or program at the run time. Press F4 under the main menu to select this function.

Key Selections :

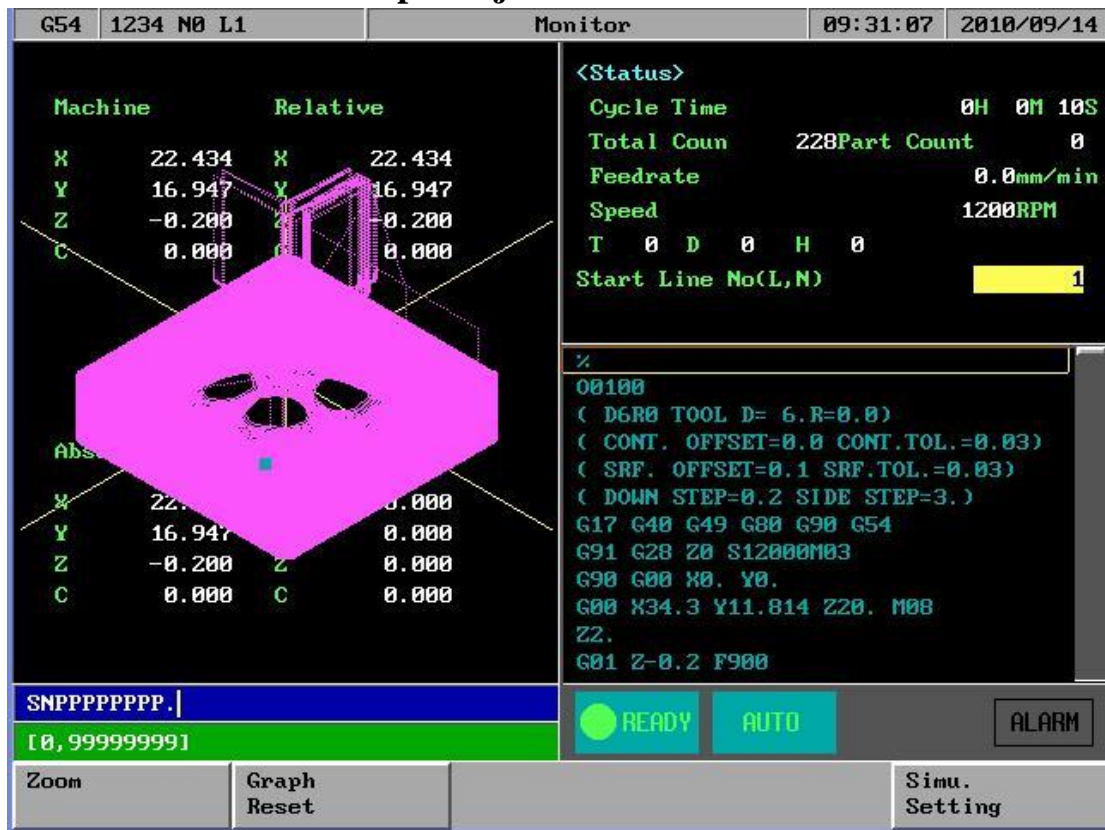
1.3.4.1 F1 : Coord

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

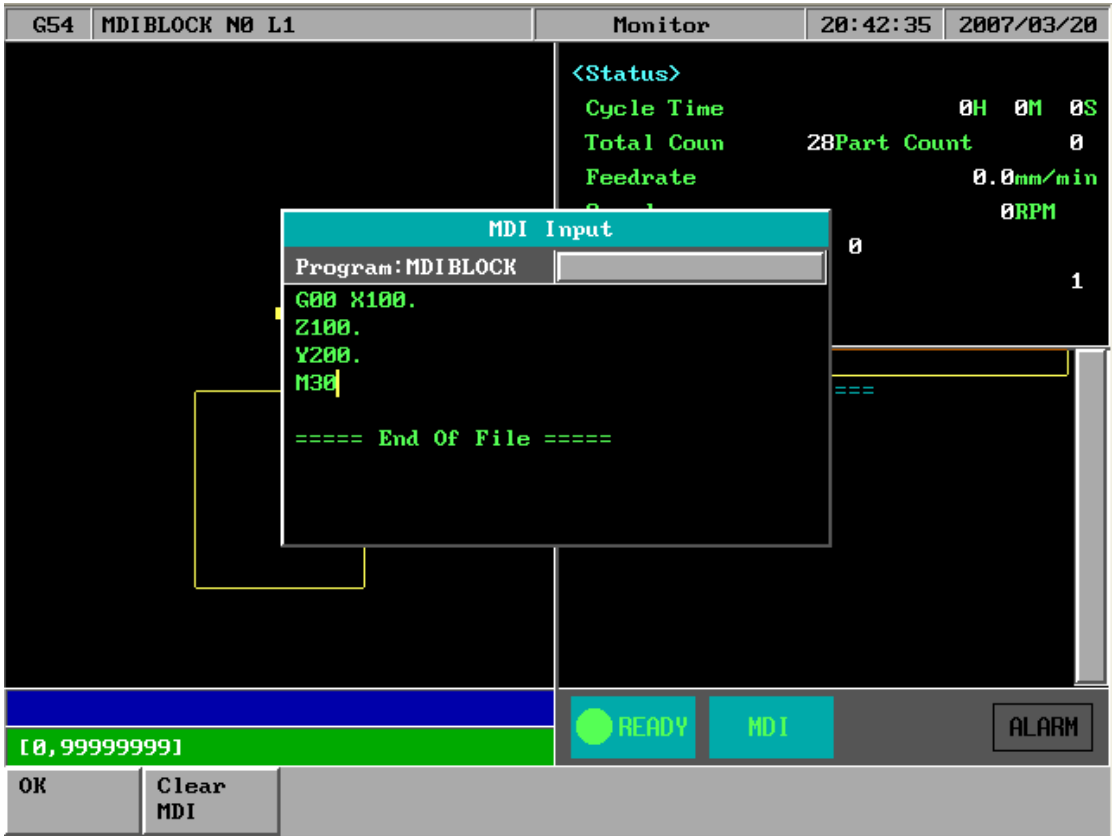
Function: To set simulation parameter

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 1234 N0 L1		Monitor		09:33:09 2010/09/14	
Machine X 22.434 Y 16.947 Z -0.200 C 0.000			Relative X 22.434 Y 16.947 Z -0.200 C 0.000		
			Input Mode(A:ABS:I:INC) (X..Z:TeachIn) ABS		
			<Tool Offset>		
	Radius	Rad	Wear	Length	Len
01	0.000	0.000	0.000	0.000	0.000
02	0.000	0.000	0.000	0.000	0.000
03	0.000	0.000	0.000	0.000	0.000
04	0.000	0.000	0.000	0.000	0.000
05	0.000	0.000	0.000	0.000	0.000
06	0.000	0.000	0.000	0.000	0.000
07	0.000	0.000	0.000	0.000	0.000
08	0.000	0.000	0.000	0.000	0.000
09	0.000	0.000	0.000	0.000	0.000
10	0.000	0.000	0.000	0.000	0.000
11	0.000	0.000	0.000	0.000	0.000
12	0.000	0.000	0.000	0.000	0.000
SNPPPPPPP.			<input checked="" type="radio"/> READY <input type="radio"/> AUTO <input type="button" value="ALARM"/>		
Clear X Relative		Clear Y Relative		Clear Z Relative	
				Tool No Data	

1.3.4.6 F5 : Work Record

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

Operation: Under

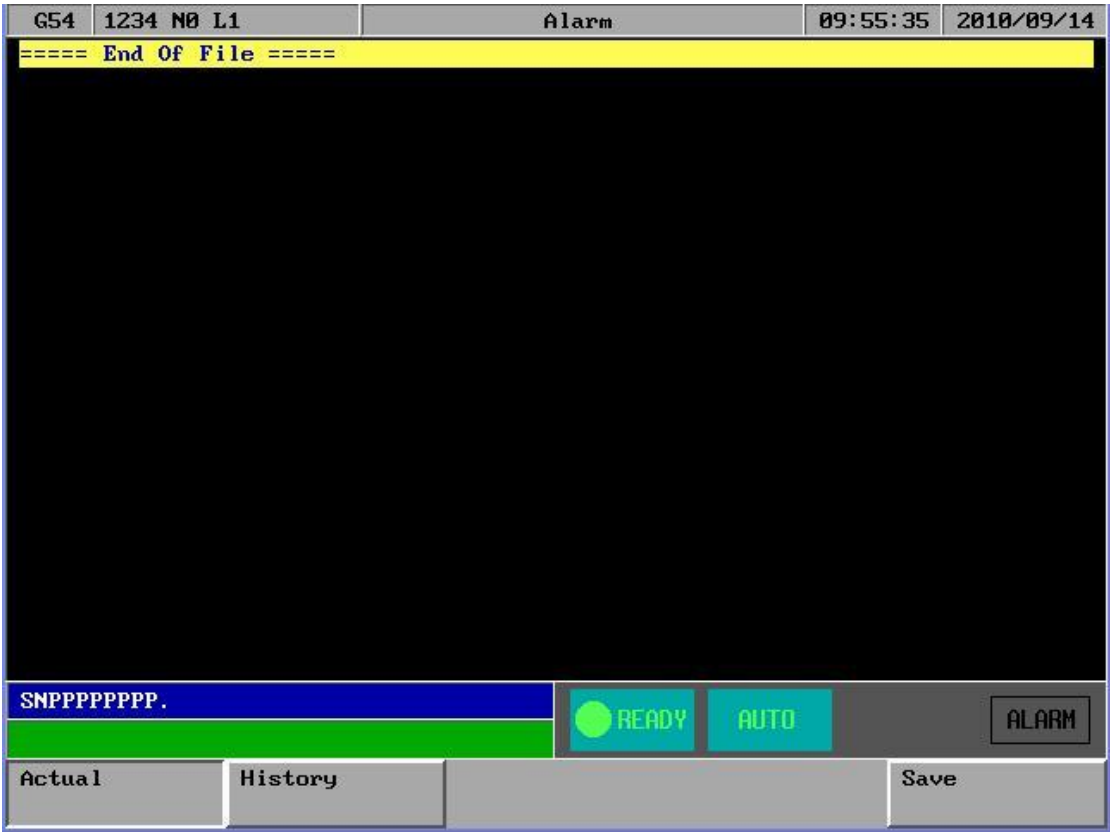
Mo

[illegible]

nitor submenu, press  and then F5.

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.

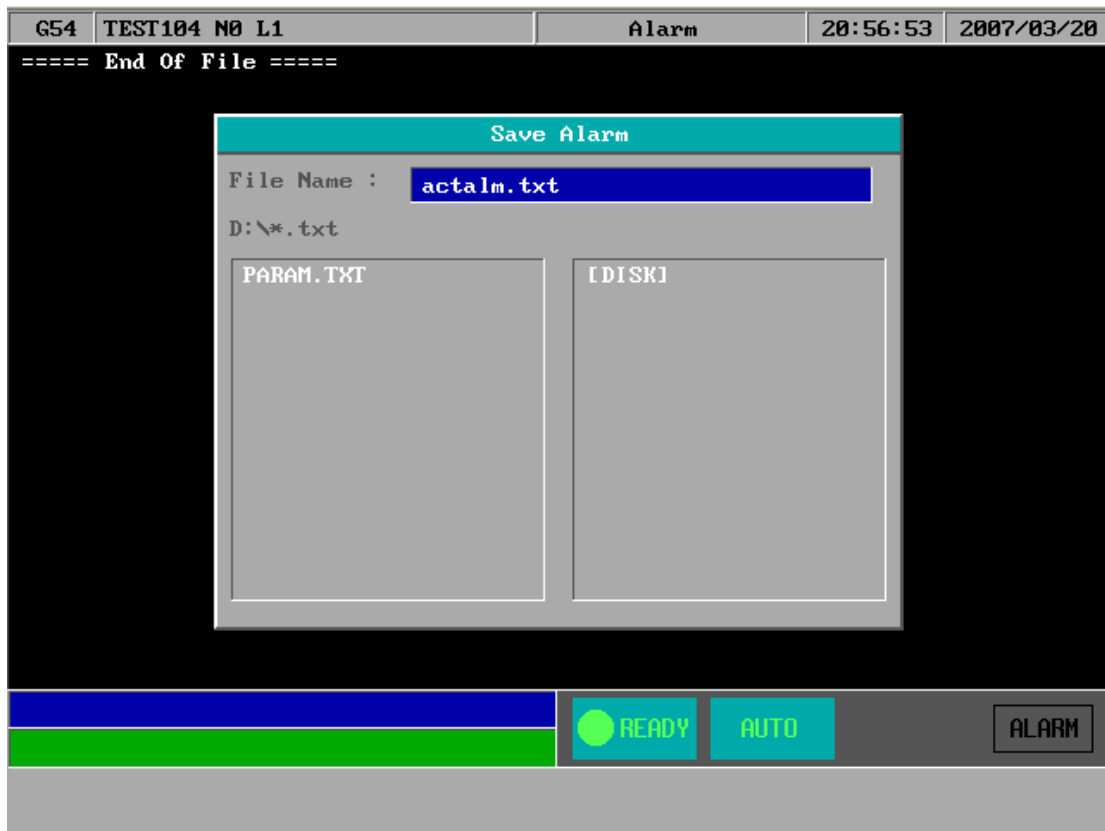
1.3.5.2 F2 : History

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.



1.3.7 F2 : 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  and then F2 to access this function as shown in the following figure.

G54	1234 N0 L1	Diagnosis	09:59:03	2010/09/14
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	FF FF 00 FF FF 00 FF FF 00 00	010	00 00 00 00 00 FF 00 FF FF 00	
020	00 00 FF 00 00 00 00 00 00 00	030	FF FF 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	
		READY AUTO		ALARM
PLC State	System Data	Global Var.	Coord Var.	System Manager

Key Selections:

1.3.7.1 F1 : PLC State

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

G54	1234 N0 L1	Diagnosis																10:02:41	2010/09/14		
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	FF	FF	00	FF	FF	00	FF	FF	00	00	010	00	00	00	00	00	FF	00	FF	FF	00
020	00	00	FF	00	00	00	00	00	00	00	030	FF	FF	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

READY

AUTO

ALARM

PLC I Bit	PLC O Bit	PLC C Bit	PLC S Bit	PLC A Bit
--------------	--------------	--------------	--------------	--------------

1.3.7.2 F2 : System Data

Function :

G54	1234 N0 L1	Diagnosis						10:03:25	2010/09/14
NO	DATA	NO	DATA	NO	DATA	NO	DATA	NO	DATA
0000	18886	0016	4444	0032	0	0048	40000	0064	0
0001	437283	0017	4444	0033	0	0049	32000	0065	0
0002	217466	0018	3333	0034	0	0050	0	0066	0
0003	729854	0019	11111	0035	0	0051	0	0067	0
0004	4883	0020	0	0036	0	0052	17075600	0068	0
0005	9766	0021	0	0037	0	0053	0	0069	0
0006	151	0022	169	0038	-1	0054	0	0070	0
0007	7060184	0023	97	0039	-1	0055	0	0071	0
0008	0	0024	22434	0040	22434	0056	0	0072	22434
0009	0	0025	16946	0041	16947	0057	0	0073	16947
0010	0	0026	-200	0042	-200	0058	0	0074	-200
0011	0	0027	0	0043	0	0059	0	0075	0
0012	1000	0028	0	0044	1200	0060	0	0076	0
0013	0	0029	17075975	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	010.42

PLC State	System Data	Global Var.	Coord Var.	System Manager

1.3.7.3 F3 : Global Variable

G54	1234 N0 L1	Diagnosis						10:03:25	2010/09/14
N0	DATA	N0	DATA	N0	DATA	N0	DATA	N0	DATA
0000	18886	0016	4444	0032	0	0048	40000	0064	0
0001	437283	0017	4444	0033	0	0049	32000	0065	0
0002	217466	0018	3333	0034	0	0050	0	0066	0
0003	729854	0019	11111	0035	0	0051	0	0067	0
0004	4883	0020	0	0036	0	0052	17075600	0068	0
0005	9766	0021	0	0037	0	0053	0	0069	0
0006	151	0022	169	0038	-1	0054	0	0070	0
0007	7060184	0023	97	0039	-1	0055	0	0071	0
0008	0	0024	22434	0040	22434	0056	0	0072	22434
0009	0	0025	16946	0041	16947	0057	0	0073	16947
0010	0	0026	-200	0042	-200	0058	0	0074	-200
0011	0	0027	0	0043	0	0059	0	0075	0
0012	1000	0028	0	0044	1200	0060	0	0076	0
0013	0	0029	17075975	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	U10.42

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

1.3.7.4 F4 : Coord Variable

[illegible]

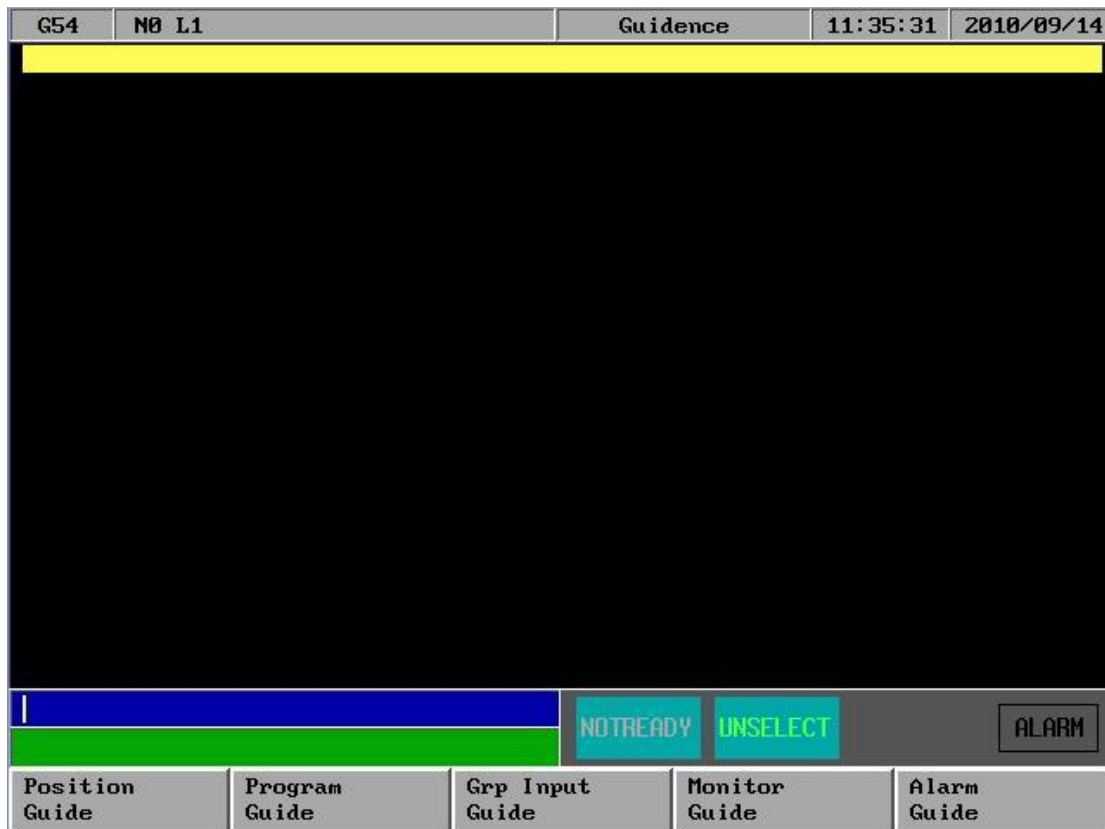
G54	1234 N0 L1	Diagnosis	10:13:14	2010/09/14
-----	------------	-----------	----------	------------

[illegible]

1.3.8 F3 : 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  then F3 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

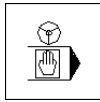
2.1.9 MDI mode single block execute

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



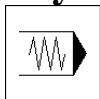
Description : User can use this function to check NC file

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

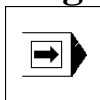


Description : User can use this function to check NC file

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

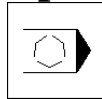


Description : User can use this function to check NC file

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

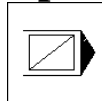
2.1.13 Option Stop



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

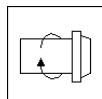
2.1.14 Option Skip



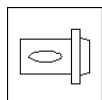
Description : User can use this function to decide NC file ‘/’ is skip or not
Operation :

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

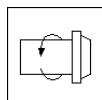
2.1.15 Spindle control



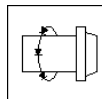
Spindle CW rotate



Spindle stop

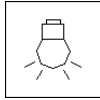


Spindle CCW rotate



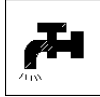
Spindle low speed: When spindle is rotate, press this key,
spindle would rotate with low speed

2.1.16 Working led



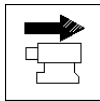
ON/OFF working led

2.1.17 Working Liquid

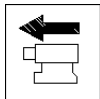


Flush working liquid

2.1.18 Aux table backward



2.1.19 Aux table forward



2.2 Text key description:

A	~	Z	:	English 26 character key
0	~	9	:	numerical key
		DEL	:	delete a character
		INS	:	Insert /replace mode switch
		SHIFT	:	for select keyboard the other textkey
		SPACE	:	for select keyboard the other textkey
		BACK SPACE	:	"Backspace" delete a character
		RESET	:	"RESET " abort the CNC status ,so please be careful to use this key
		ENTER	:	to input current data to input box
		HELP	:	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):

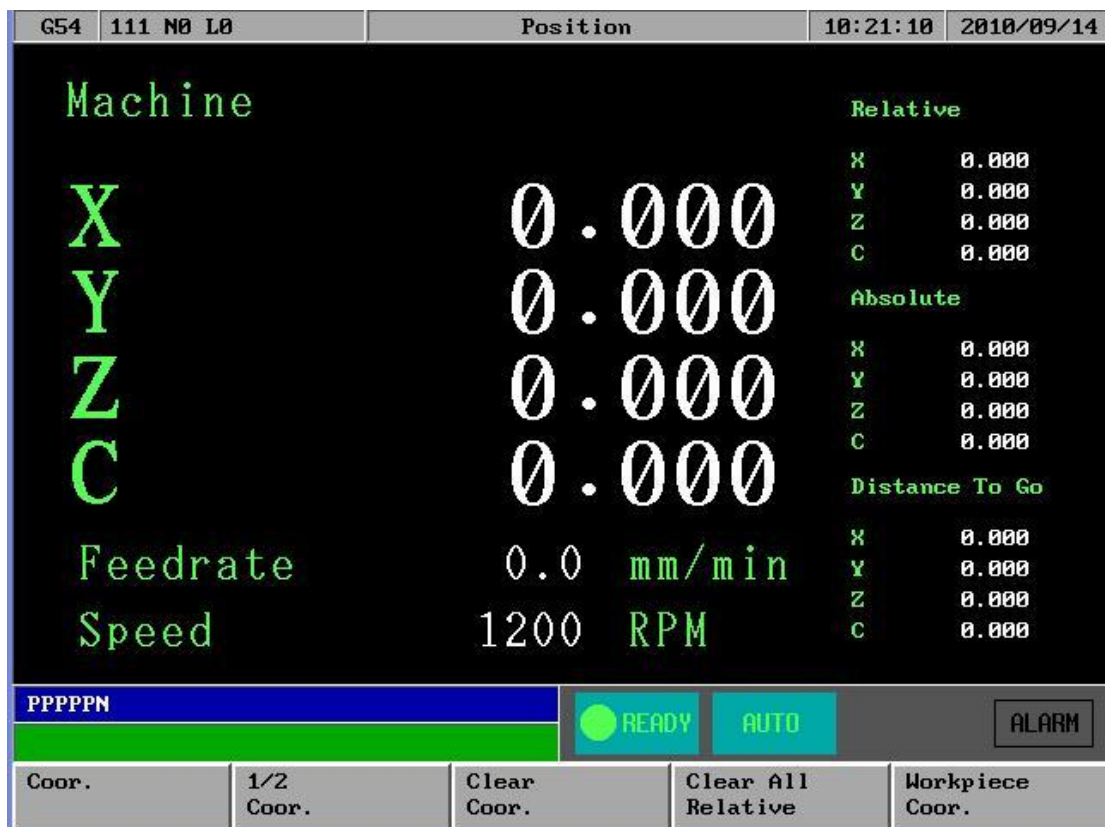
1. Release emergency stop button ,CNC status “NOT READY” change to “READY ”
2. Mode select switch rotate to MPG INC JOG mode
3. Select movement axis
4. Select movement distance (*1,*10,*100)
5. Rotate MPG ,table would move .

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




4. Press F1 “New file” ,to open a new file
5. Press F2”copy file” ,to copy current file to target file
6. Press F3 “delete file“ ,to delete current file
7. Press F4 “Import” ,to import a new file from floppy disk
8. Press F5 “Export “ ,to export file to floppy disk
9. Press  then F1 “RS232 Import” ,to import a new file from RS232
10. Press  then F2 “RS232 Export “ ,to export file to RS232
11. Press  then F3 “Network Import” ,to import a new file from network

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 1234 N0 L1		Monitor		09:33:09 2010/09/14																																																																																												
Machine X 22.434 Y 16.947 Z -0.200 C 0.000			Relative X 22.434 Y 16.947 Z -0.200 C 0.000																																																																																													
			Input Mode(A:ABS:I:INC) (X..Z:TeachIn) ABS																																																																																													
Abs X 22.434 Y 16.947 Z -0.200 C 0.000			<Tool Offset>																																																																																													
			<table border="1"> <thead> <tr> <th></th> <th>Radius</th> <th>Rad</th> <th>Wear</th> <th>Length</th> <th>Len</th> <th>Wear</th> </tr> </thead> <tbody> <tr><td>01</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>02</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>03</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>04</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>05</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>06</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>07</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>08</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>09</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>10</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>11</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> <tr><td>12</td><td>0.000</td><td></td><td>0.000</td><td>0.000</td><td></td><td>0.000</td></tr> </tbody> </table>				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
	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																																																																																										
SNPPPPPPP.			<div> <div>● READY</div> <div>AUTO</div> <div>ALARM</div> </div>																																																																																													
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 coor.”

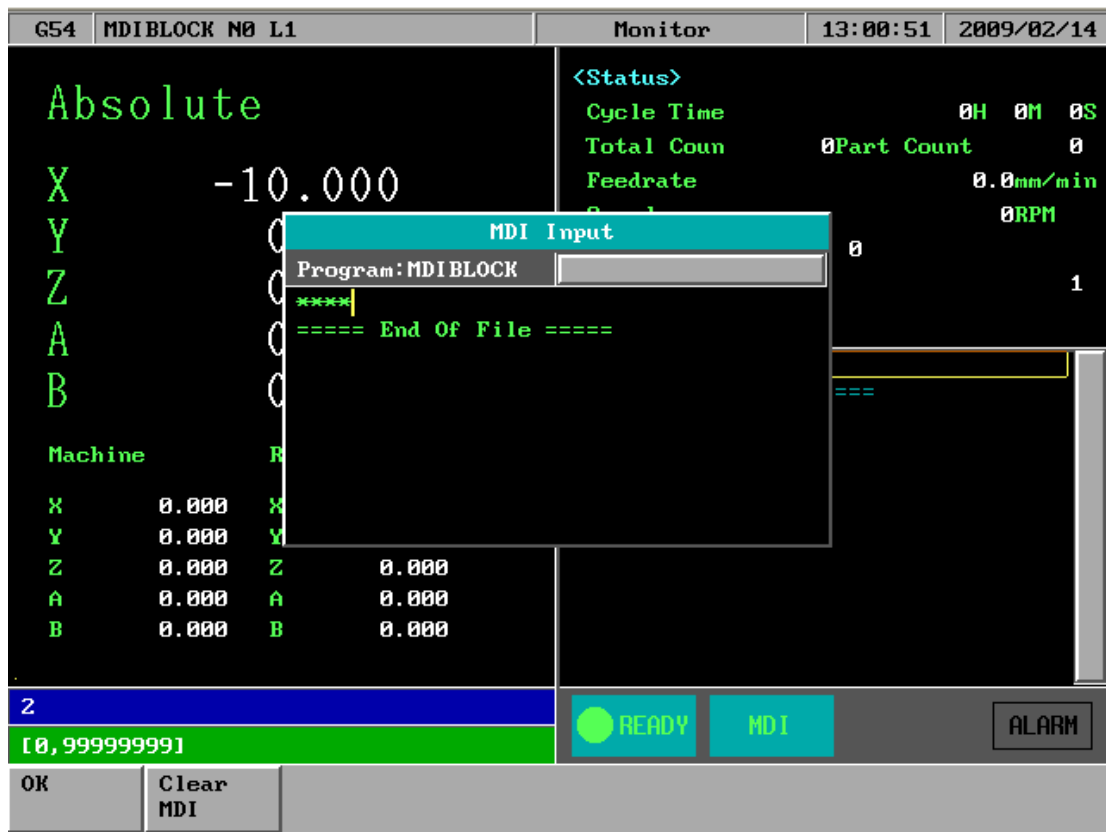
G54		AA N0 L1		Position		16:28:10		2010/09/13	
External Shift		G54		G55		Machine			
X	0.000	X	0.000	X	0.000	X	0.000		
Y	0.000	Y	0.000	Y	0.000	Y	0.000		
Z	0.000	Z	0.000	Z	0.000	Z	0.000		
C	0.000	C	0.000	C	0.000	C	0.000		
G56		G57		G58		Relative			
X	0.000	X	0.000	X	0.000	X	0.000		
Y	0.000	Y	0.000	Y	0.000	Y	0.000		
Z	0.000	Z	0.000	Z	0.000	Z	0.000		
C	0.000	C	0.000	C	0.000	C	0.000		
						Aux. Coordinate			
						X	0.000		
						Y	0.000		
						Z	0.000		
					READY	AUTO		ALARM	
Coord. Latched		Rel. Latched		Aux. Latched				Middle Func.	

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”

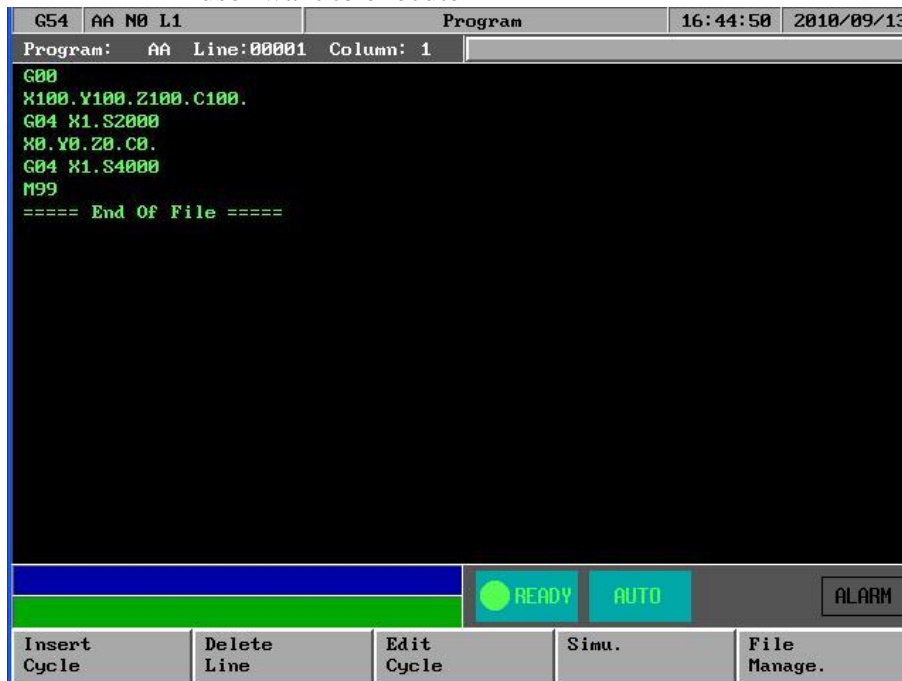


4. The screen display the MDI input window
5. Key in MDI data at input bar and press “ENTER”
6. Press 2nd operation panel “START” to execute the current block
7. If current block SYNTAX is correct ,the data in the window would be disappear .

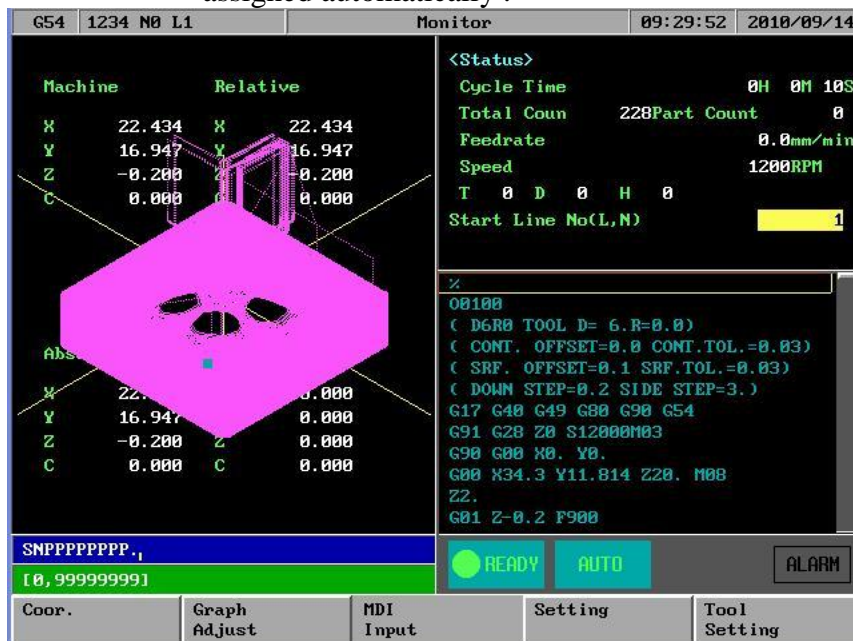
3.8 Assigned an executing NC file (AUTO)

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 .

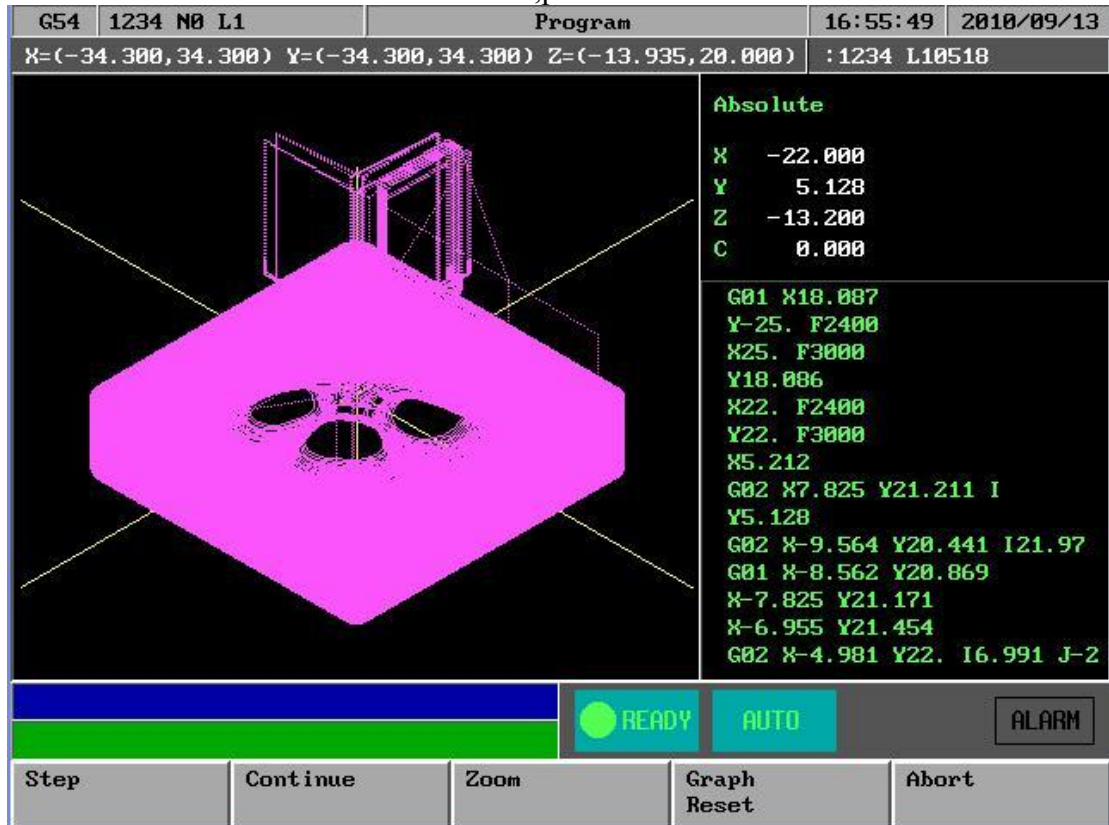


5. Please make sure CNC status is "READY", that is the only available status to assigned executing NC file .

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”

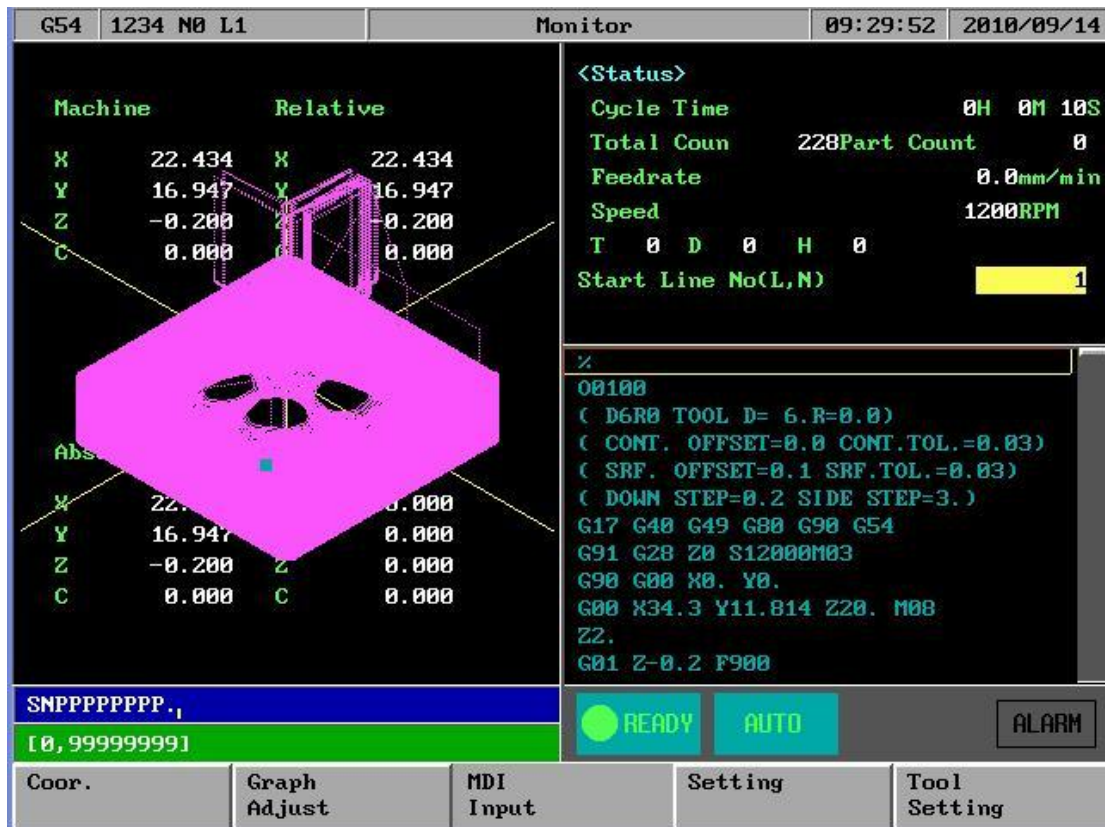


3. User can use “STEP” to check NC file step by step
4. Use “Continue” to simulation whole picture
5. Use “Zoom” check more detail
6. Use “Simulation setting” set simulation parameter
7. CNC status is “READY”, Press group function key “Monitor”
8. that is the available status to assigned executing NC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

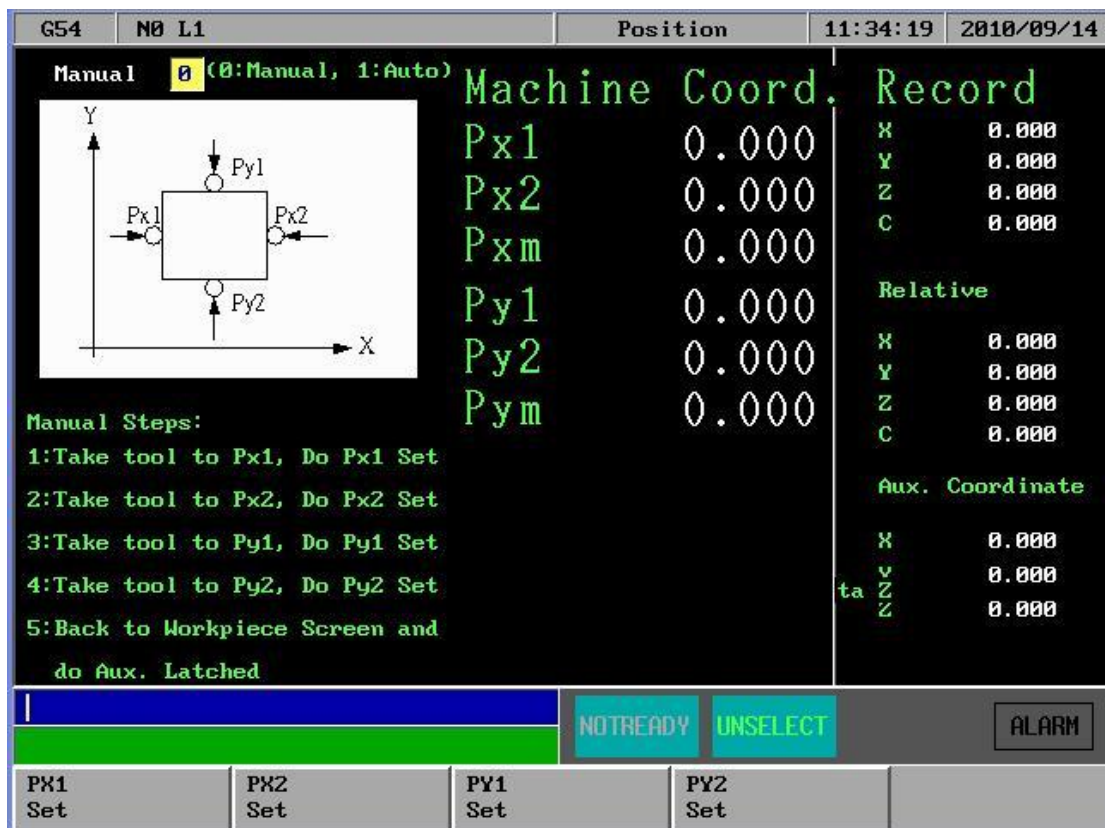
Here we supply the operation introduction of manual center and auto center as follows.

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.

G54 N0 L1		Position	11:37:05	2010/09/14																																																				
Auto 1 (0:Manual, 1:Auto)		<div style="display: flex; justify-content: space-between;"> <div style="width: 30%;"> </div> <div style="width: 65%;"> <h2 style="margin: 0;">Machine Coord. Record</h2> <table style="width: 100%; border-collapse: collapse;"> <tr> <td style="width: 30%;">Px1</td> <td style="width: 30%;">0.000</td> <td style="width: 10%;">X</td> <td style="width: 30%;">0.000</td> </tr> <tr> <td>Px2</td> <td>0.000</td> <td>Y</td> <td>0.000</td> </tr> <tr> <td>Pxm</td> <td>0.000</td> <td>Z</td> <td>0.000</td> </tr> <tr> <td></td> <td></td> <td>C</td> <td>0.000</td> </tr> <tr> <td colspan="4" style="padding-top: 10px;">Relative</td> </tr> <tr> <td>Py1</td> <td>0.000</td> <td>X</td> <td>0.000</td> </tr> <tr> <td>Py2</td> <td>0.000</td> <td>Y</td> <td>0.000</td> </tr> <tr> <td>Pym</td> <td>0.000</td> <td>Z</td> <td>0.000</td> </tr> <tr> <td></td> <td></td> <td>C</td> <td>0.000</td> </tr> <tr> <td colspan="4" style="padding-top: 10px;">Aux. Coordinate</td> </tr> <tr> <td></td> <td></td> <td>X</td> <td>0.000</td> </tr> <tr> <td></td> <td></td> <td>Y</td> <td>0.000</td> </tr> <tr> <td></td> <td></td> <td>Z</td> <td>0.000</td> </tr> </table> </div> </div>			Px1	0.000	X	0.000	Px2	0.000	Y	0.000	Pxm	0.000	Z	0.000			C	0.000	Relative				Py1	0.000	X	0.000	Py2	0.000	Y	0.000	Pym	0.000	Z	0.000			C	0.000	Aux. Coordinate						X	0.000			Y	0.000			Z	0.000
Px1	0.000	X	0.000																																																					
Px2	0.000	Y	0.000																																																					
Pxm	0.000	Z	0.000																																																					
		C	0.000																																																					
Relative																																																								
Py1	0.000	X	0.000																																																					
Py2	0.000	Y	0.000																																																					
Pym	0.000	Z	0.000																																																					
		C	0.000																																																					
Aux. Coordinate																																																								
		X	0.000																																																					
		Y	0.000																																																					
		Z	0.000																																																					
Auto Steps: 1:Input Touch Information 2:Take tool to P1 do Z Teach 3:Take tool to Auto Start P2 4:Press F1 to do Auto Center 5:Back to Workpiece Screen and do Aux. Latched		Input Material Information: Length I: Width J: SaveDist H Feedrate F: Z Coord P1																																																						
NOTREADY		UNSELECT		ALARM																																																				
Auto Mid Start	Auto Mid FeedHold																																																							

Operation :

1. To enter center submenu, press F1"Positon"=>F5"workpiece Coord"=>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 F5 "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.

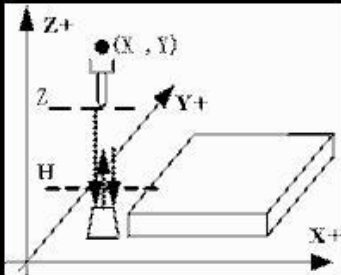
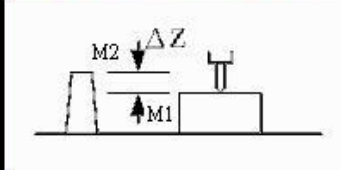
3.12 Z-axis Auto Tool

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"=>=>F1"Tool Tip Measure"

G54	N0 L1	Position	11:39:12	2010/09/14
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		
Machine X 0.000 Y 0.000 Z 0.000 C 0.000 Relative X 0.000 Y 0.000 Z 0.000 C 0.000 Aux. Coordinate X 0.000 Y 0.000 Z 0.000		NOTREADY UNSELECT ALARM		
0: Aux, 1: G54, 2: G55, 3: G56, 4: G57, 5: G				
Measure Start	Measure FeedHold	Delta Z Set	XY Ref. Teach	Mach. Z Teach

Auto tool length measurement

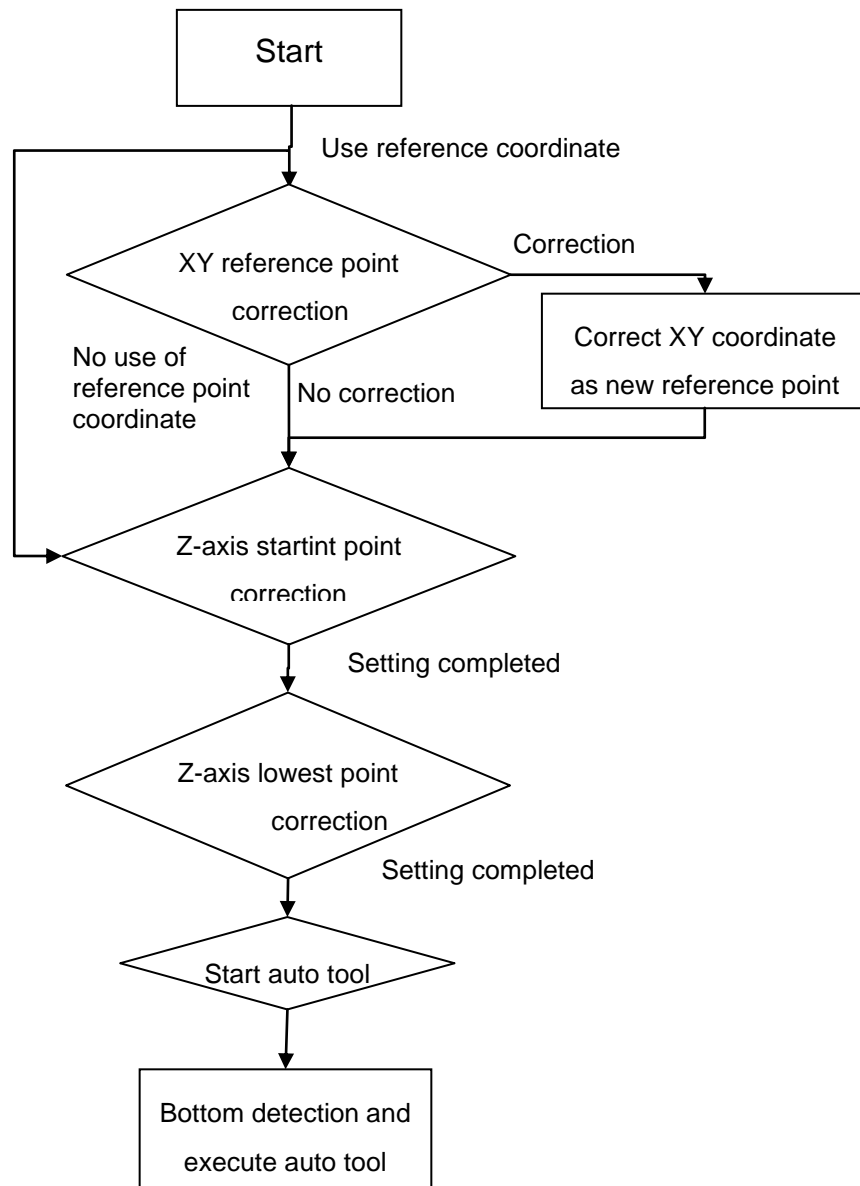
It needs to measure the length of the tool again when change another tool in order to compensate the correct tool length to the processing path.

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;

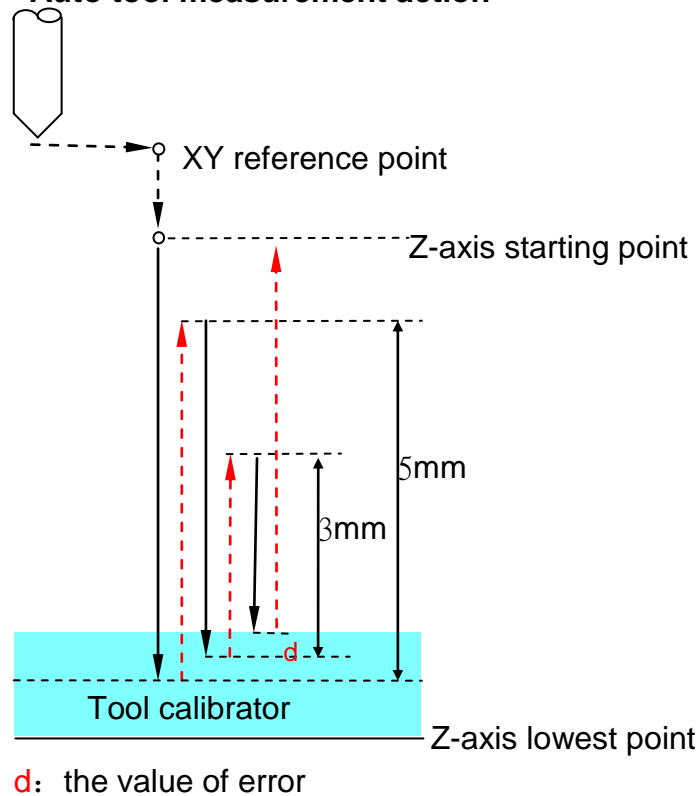


Delta z set:

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



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.

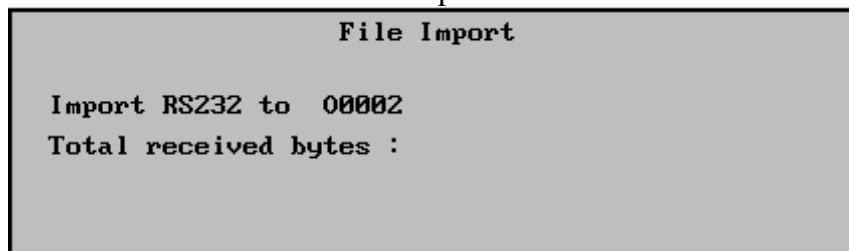
- I. Press [program] [file manage] to file manage submenu.
And than to press [=>] show up as follow:



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



- 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.



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.

G54	1234 N0 L1	Program		17:59:21	2010/09/13
Program storage free space 110540800 bytes					
1234	241629	03/22/07	11:17	pm	
4-06R0~1	2	11/03/06	10:23	am	
AA	74	11/02/06	01:19	pm	G00
FF	7	08/25/05	05:03	pm	Y100.
00001	68	12/02/05	02:25	pm	G00X50.Y50.Z50.
01000	258	12/02/05	02:25	pm	銑床測試程式 FOR G73
01010	263	12/02/05	02:25	pm	銑床測試程式 FOR G74
TEST	2	11/03/06	09:30	am	
RS232	DNC Program				
				 READY	AUTO
					
RS232 Import		RS232 Export		Network Import	
				Multi Import	
				Multi Export	

- 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)

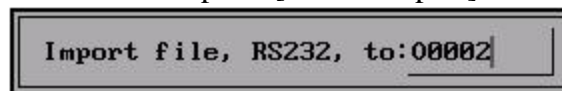
3. Step III : Turn on the controller power. The RS232 file input is ready. Operation has been stated as follows.

Operation has been stated as follows.

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



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



- 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.

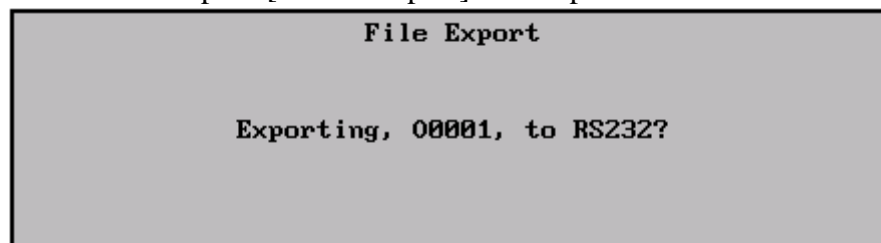


4. Step IV : Upper End of PC. Executing c:\cnc\cnc into the controller simulation software. Does RS232 file output. Operation has been stated as follows.

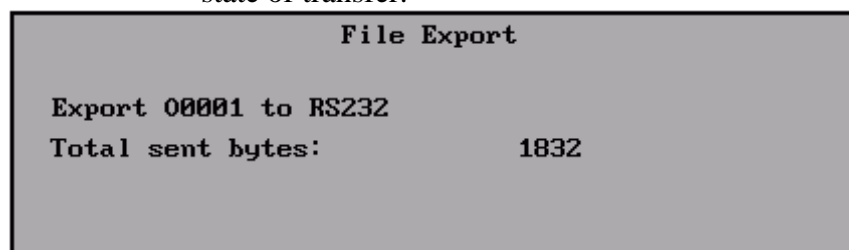
- I. Press [program] [file manage] to file manage submenu. Than to press [=>] key to appearing. And press up/down key select that want output of file. Such as O1000.



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



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



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.**