SYNTEC

11 Series Mill Controller Operation Manual

Date: 2015/11/13

Version: 1.4

Contents

1 FUNCTION KEY AND SYSTEM CONFIGURATION 4

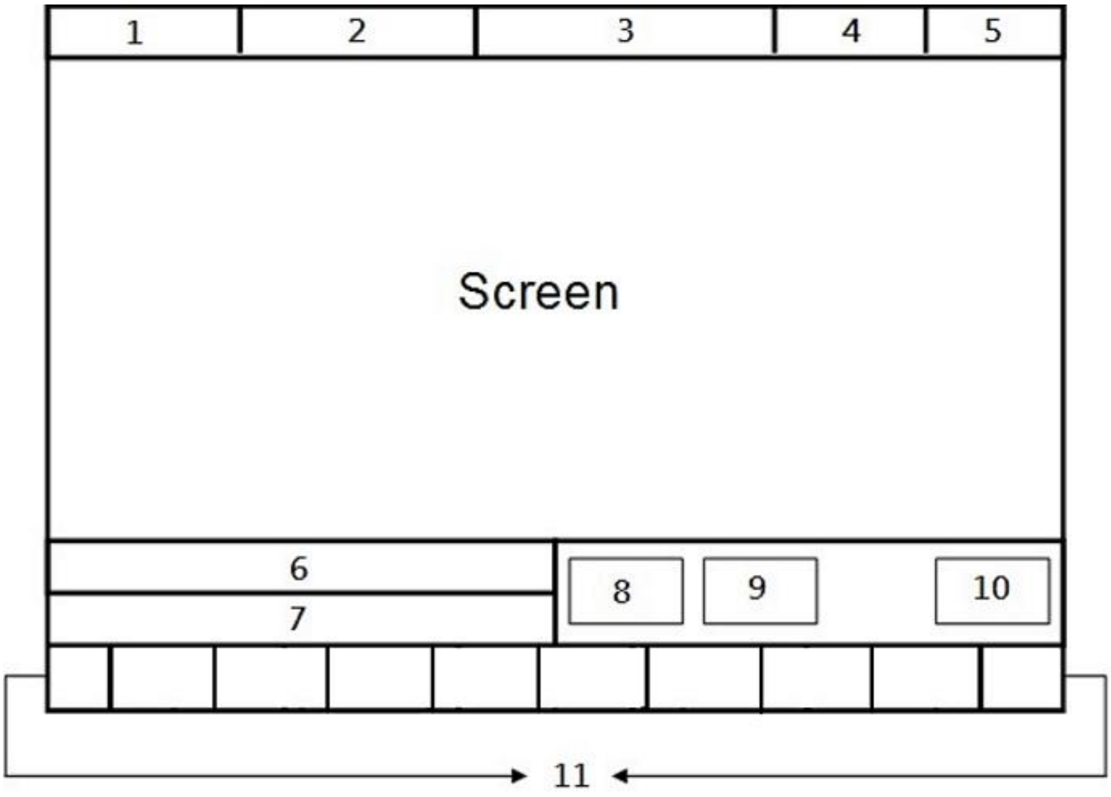
1.1 Main Screen Sections 4
1.2 CNC System Configuration 5
1.3 Coordinate 6
  1.3.1 Explanation of Function 7
  1.3.2 Switch Coordinate 8
  1.3.3 Half Coordinate 8
  1.3.4 Clear Rel. Coord. 9
  1.3.5 Clear All Rel. Coord. 9
1.4 Program 10
  1.4.1 Execute 11
  1.4.2 Delete Line 11
  1.4.3 Search/Replace 12
  1.4.4 Can Cycle 14
  1.4.5 Block Copy 16
  1.4.6 Teach 20
  1.4.7 Simulation 25
  1.4.8 File Manager 33
1.5 Offset/Setting 42
  1.5.1 Workpiece Cord. 43
  1.5.2 Tool Set 52
  1.5.3 Tool Tip Measure 55
  1.5.4 User Parameter Setting 57
1.6 Monitor 58
  1.6.1 Monitor Area of Machining Information 59
  1.6.2 Open File to Edit 64
  1.6.3 Simulation Switch 64
  1.6.4 MDI Input 64
  1.6.5 Parameter Set 65
  1.6.6 Tool Wear Set 65
  1.6.7 Start MPG Coordinate 66
  1.6.8 Work Record 66
  1.6.9 Clear Acum Cycle Time 66
  1.6.10 Graph Adjust 66
1.7 Maintain 67
  1.7.1 Alarm 67

1.7.2 Network Setting 69
1.7.3 Fast Diagnostic....73
1.7.4 PLC Param Setting....76
1.7.5 System Setting....77
1.7.6 Backup System....79
1.7.7 About....79
2 MACHINE OPERATION PANEL 80
2.1 Operation Panel 80
2.2 Text Key Description 86
3 HOW TO OPERATE 11 SERIES CONTROLLER 89
3.1 System Status 89
3.1.1 Not Ready 89
3.1.2 Ready 89
3.1.3 Busy 89
3.1.4 Pause 89
3.1.5 B-stop 90
3.2 Machine Preparation....90
3.2.1 Manual Function 90
3.2.2 Machining Process 93
3.3 Workpiece Preparation 96
3.3.1 Workpiece Cord set....96
3.3.2 Middle Func. 97
3.3.3 Tool Prepare 104
3.4 Program Preparation and Execute Machining....118
3.4.1 Specifying Machining Program 118
3.4.2 Simu. Setting....120
3.4.3 Machining Test 121
3.4.4 Machining Monitor 124
3.4.5 Alarm Processing 126
3.4.6 Network Setting 128
3.4.7 PC Setting 129
3.5 File Transfer 145
3.5.1 File Import....145
3.5.2 File Export....146
4 APPENDIX....147
4.1 Release Note....147

4.2 Contact Window..147

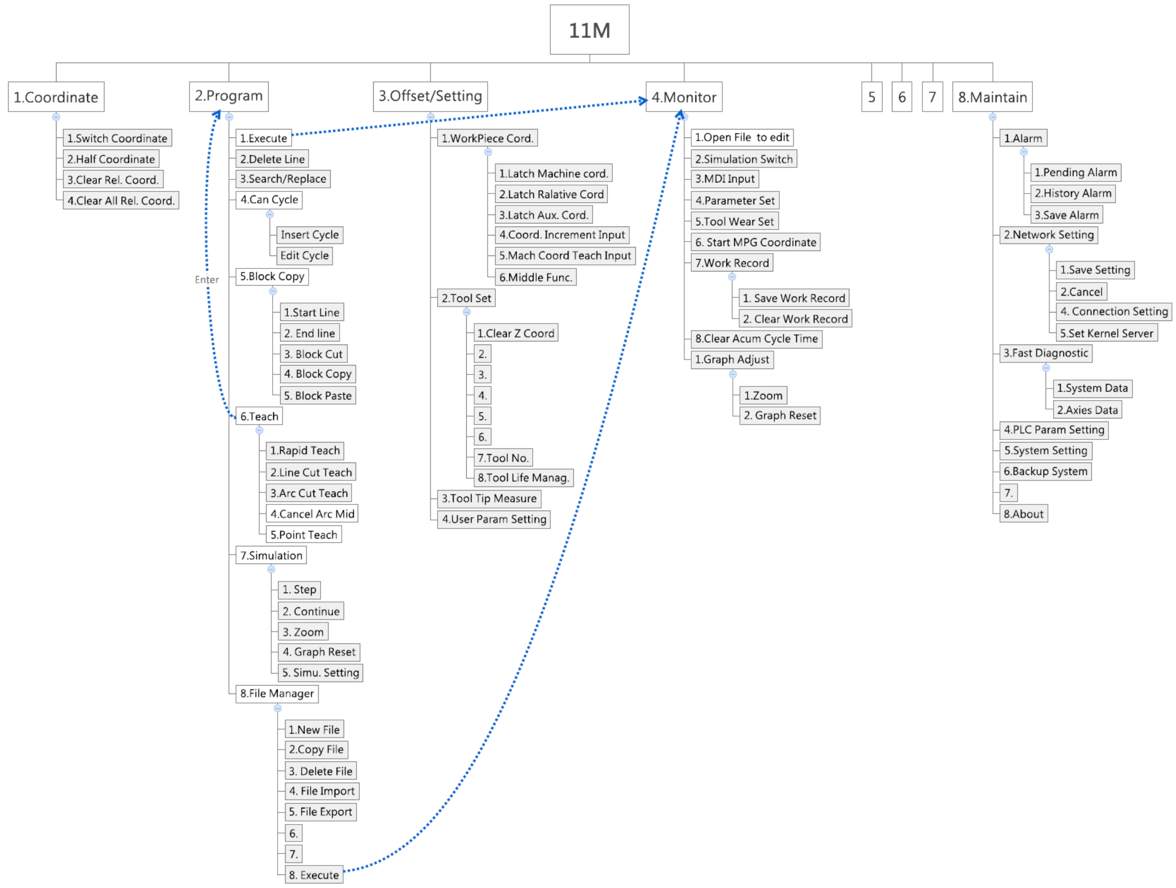
1 Function Key and System Configuration

1.1 Main Screen Sections

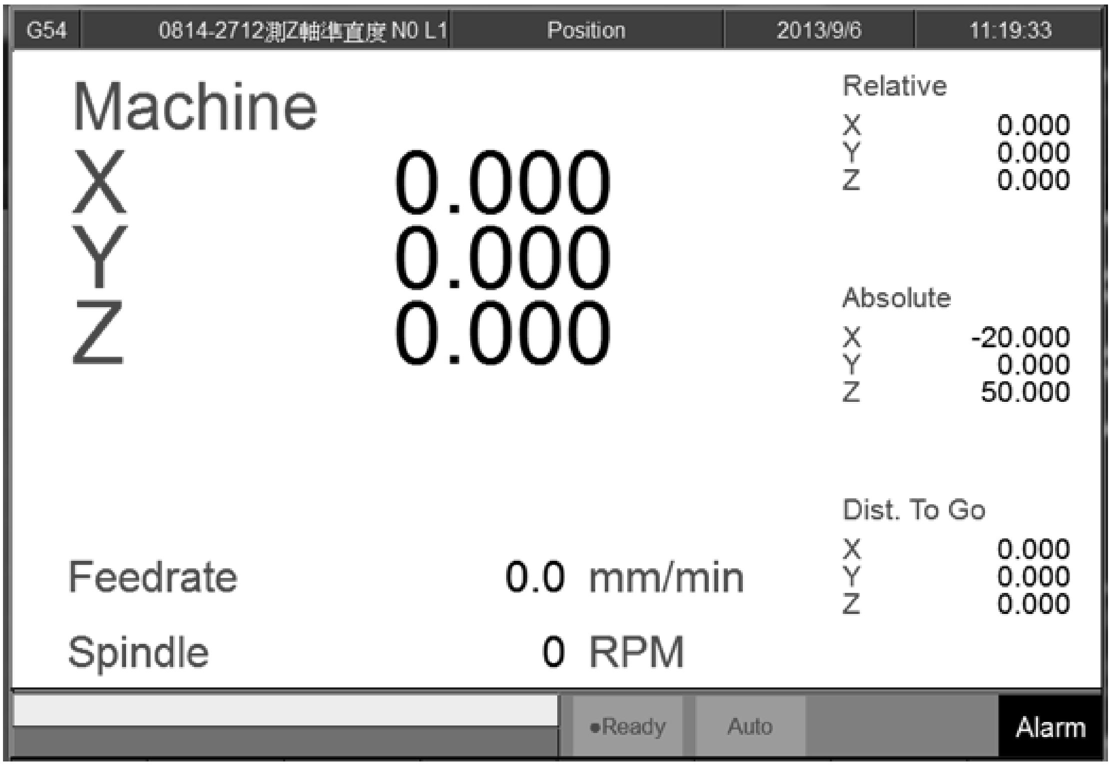


- Meanings for Sections on the Display:
    1. Present coordinate system
    2. Working file name and machining command line
    3. Title of interface(Current interface)
    4. Date
    5. Time
    6. Data input
    7. Display(Hint)
    8. Status
    9. Mode
    10. Alarm
    11. Function Key

1.2 CNC System Configuration


11M
1.Coordinate
1.Switch Coordinate
2.Half Coordinate
3.Clear Rel. Coord.
4.Clear All Rel. Coord.
2.Program
1.Execute
2.Delete Line
3.Search/Replace
4.Can Cycle
Insert Cycle
Edit Cycle
Enter
5.Block Copy
1.Start Line
2. End line
3. Block Cut
4. Block Copy
5. Block Paste
6.Teach
1.Rapid Teach
2.Line Cut Teach
3.Arc Cut Teach
4.Cancel Arc Mid
5.Point Teach
7.Simulation
1. Step
2. Continue
3. Zoom
4. Graph Reset
5. Simu. Setting
8.File Manager
1.New File
2.Copy File
3. Delete File
4. File Import
5. File Export
6.
7.
8. Execute
3.Offset/Setting
1.WorkPiece Cord.
1.Latch Machine cord.
2.Latch Ralative Cord
3.Latch Aux. Cord.
4.Coord. Increment Input
5.Mach Coord Teach Input
6.Middle Func.
2.Tool Set
1.Clear Z Coord
2.
3.
4.
5.
6.
7.Tool No.
8.Tool Life Manag.
3.Tool Tip Measure
4.User Param Setting
4.Monitor
1.Open File to edit
2.Simulation Switch
3.MDI Input
4.Parameter Set
5.Tool Wear Set
6. Start MPG Coordinate
7.Work Record
1. Save Work Record
2. Clear Work Record
8.Clear Acum Cycle Time
1.Graph Adjust
1.Zoom
2. Graph Reset
5
6
7
8.Maintain
1.Alarm
1.Pending Alarm
2.History Alarm
3.Save Alarm
2.Network Setting
1.Save Setting
2.Cancel
4. Connection Setting
5.Set Kernel Server
3.Fast Diagnostic
1.System Data
2.Axies Data
4.PLC Param Setting
5.System Setting
6.Backup System
7.
8.About

1.3 Coordinate



0814-2712測Z軸垂直度 N0 L1

2013/9/6 11:19:33

Machine
X 0.000
Y 0.000
Z 0.000

Relative
X 0.000
Y 0.000
Z 0.000

Absolute
X -20.000
Y 0.000
Z 50.000

Dist. To Go
X 0.000
Y 0.000
Z 0.000

Feedrate 0.0 mm/min

Spindle 0 RPM

- Command
  - ✧ F1 Coordinate
- Function
  - ✧ Switch current coordinate system on the screen.
  - ✧ Display the frequently use machining information.
  - ✧ Use the function key【POS】, can switch to the current page quickly.

    PS: By pressing 【POS】 Key, you may jump from other page to this page interface.

1.3.1 Explanation of Function

1.3.1.1 Coordinate Display

- ✧ Current screen can display 4 kind of coordinate system.
- ✧ Whenever users press F1「Switch Coordinate」function key, the coordinate on the screen will switch between four different kinds of coordinates.

1.3.1.2 F(Feedrate)

- ✧ User input Feedrate (mm/min).
- ✧ Actual Feedrate of cutting tool (mm/min).
- ✧ Percentages of Feedrate.

1.3.1.3 S(Rotating Speed of Spindle)

- ✧ User input Spindle speed (RPM).
- ✧ Actual speed of spindle (mm/min).
- ✧ Percentages of Spindle speed.

1.3.1.4 Run Time

- ✧ Machining Duration.

1.3.1.5 Part counter

- ✧ Number of parts that had been finished.

1.3.1.6 T(Tool No.)

- ✧ Current Tool no. and Tool compensation no.

1.3.2 Switch Coordinate

- Command
  - ✧ F1 Coordinate→F1 switch coordinate
- Function
  - ✧ Whenever users press F1「Switch Coordinate」function key, the coordinate display on the screen will switch between four different kinds of coordinates.

1.3.3 Half Coordinate

- Command
  - ✧ F1 Coordinate→F2 Half Coordinate
- Function
  - ✧ Relative coordinate divided by 2.
  - ✧ Combined with 「Clear Rel. Coord.」function , this function will calculate the middle point of the object.
- Operation Method
  - ✧ Key in the axis that you want to calculate and then press 「Half Coordinate」.
- Example
  - ✧ Current Rel. Coord. of X axis is 10.000.
  - ✧ Key in 「X」,and then press 「half coordinate」.
  - ✧ Current Rel. Coord. of X axis Will become 5.000.

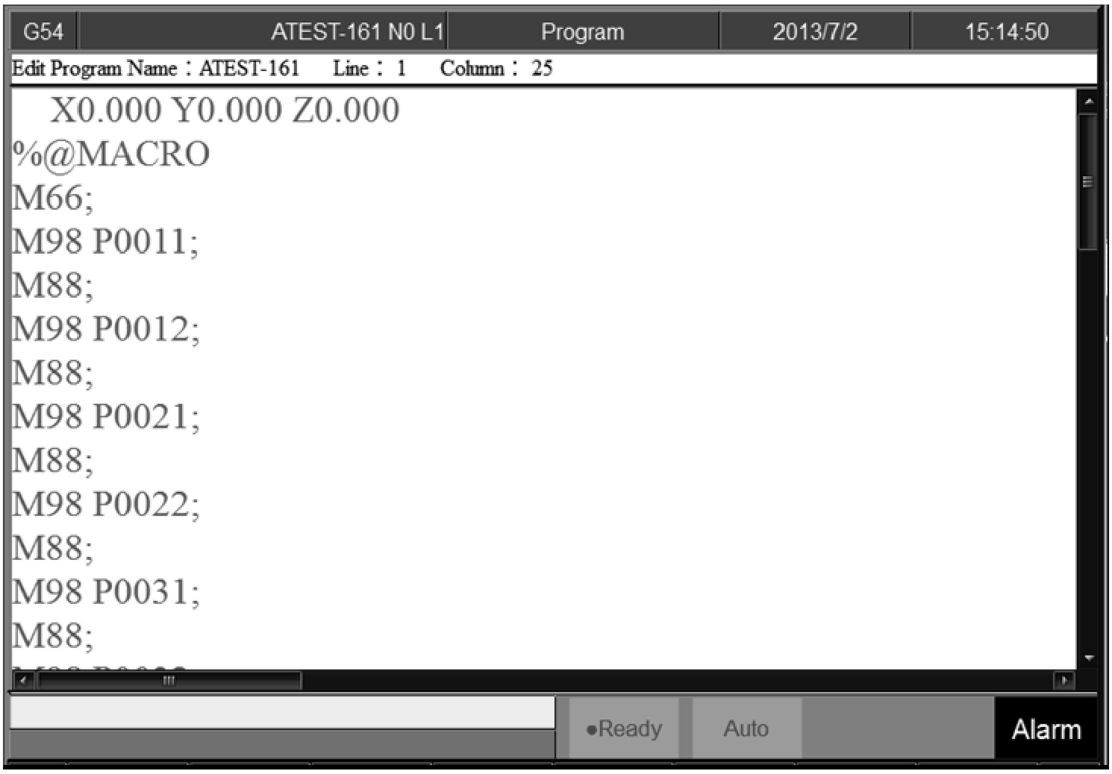
1.3.4 Clear Rel. Coord.

- Command
  - ✧ F1 Coordinate→F3 Clear Rel. Coord.
- Function
  - ✧ Set the Relative Coordinate to zero.
- Operation Method
  - ✧ Key in the axis that you want to calculate and then press 「Clear Rel. Coord.」
- Example
  - ✧ Current Rel.Coord.of X axis is 10.000.
  - ✧ Key in 「X」,and then press 「Clear Rel. Coord.」.
  - ✧ Current Rel. Coord. of X axis Will be set to 0.000.

1.3.5 Clear All Rel. Coord.

- Command
  - ✧ F1 Coordinate→F4 Clear All Rel. Coord.
- Function
  - ✧ Clear all Relative Coordinate.
- Example
  - ✧ Current X axis of relative coordinate is 10.000, Y axis is 5.000.
  - ✧ Press「Clear All Rel. Coord.」function key.
  - ✧ Relative Coordinate of X and Y will be set to 0.000.

1.4 Program


G54
ATEST-161 N0 L1
Program
2013/7/2
15:14:50
•Ready
Auto
Alarm

Edit Program Name：ATEST-161 Line： 1 Column： 25

```
   X0.000 Y0.000 Z0.000
%@MACRO
M66;
M98 P0011;
M88;
M98 P0012;
M88;
M98 P0021;
M88;
M98 P0022;
M88;
M98 P0031;
M88;
```

- Command
  - ✧ F2 Program
- Function
  - ✧ This function provides users program management and editing functions.
- Operation Method
  - ✧ Users can use【↑】【↓】【←】【→】 on the key pad to move the cursor to anywhere on the screen for editing purpose.
  - ✧ With 【Page Up】【Page Down】to switch the pages.
  - ✧ With【Home】【End】can let the cursor jump between the top and end of the line.

✧ With the function key【Prog/File】can quickly switch between「Program」and「File Manager」.

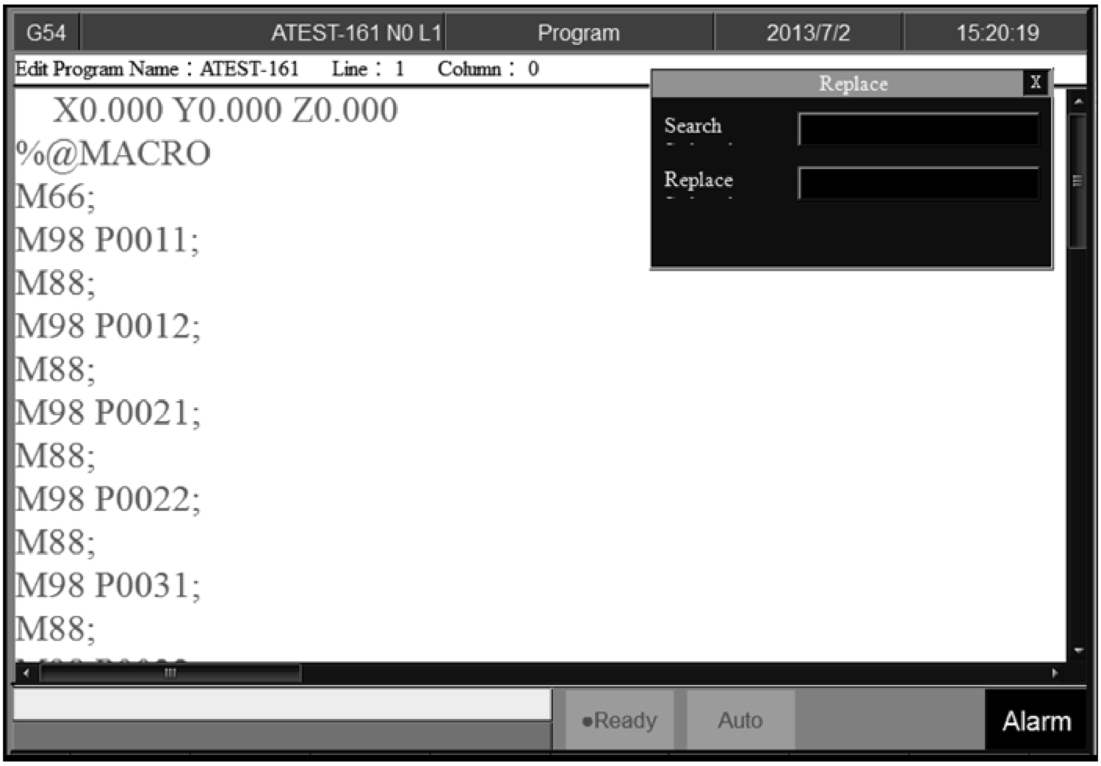
1.4.1 Execute

- Command
  - ✧ F2 Program→F1 Execute
- Function
  - ✧ Execute current program and also change the screen to 「Monitor」 page.
- Note
  - ✧ This function will be disabled during machining.

1.4.2 Delete Line

- Command
  - ✧ F2 Program→F2 Delete Line
- Function
  - ✧ Delete a line where the cursor is located.

1.4.3 Search/Replace


G54
ATEST-161 N0 L1
Program
2013/7/2
15:20:19
Replace
X
Search
Replace
•Ready
Auto
Alarm

Edit Program Name：ATEST-161 Line：1 Column：0

```
   X0.000 Y0.000 Z0.000
%@MACRO
M66;
M98 P0011;
M88;
M98 P0012;
M88;
M98 P0021;
M88;
M98 P0022;
M88;
M98 P0031;
M88;
```

- Command
  - ✧ F2 Program→F3 Search/Replace
- Function
  - ✧ Quick search for every occurrence of a specific word or phrase and automatically replace text.
- Operation Method
  - ✧ Press 「Search/Replace」function key, then Replace box will appear, just enter the text that you want to search and replace for.

1.4.3.1 Find Next

- Command
  - ✧ F2 Program→F3 Search/Replace→F1 Find Next
- Function
  - ✧ Find next.

1.4.3.2 Replace

- Command
  - ✧ F2 Program→F3 Search/Replace→F2 Replace
- Function
  - ✧ Replace with input text
- Operation Method
  - ✧ Press F2「Replace」to replace highlight string with new string.
  - ✧ If you want to skip the current highlighted string, press F1「Find Next」.

1.4.3.3 Replace All

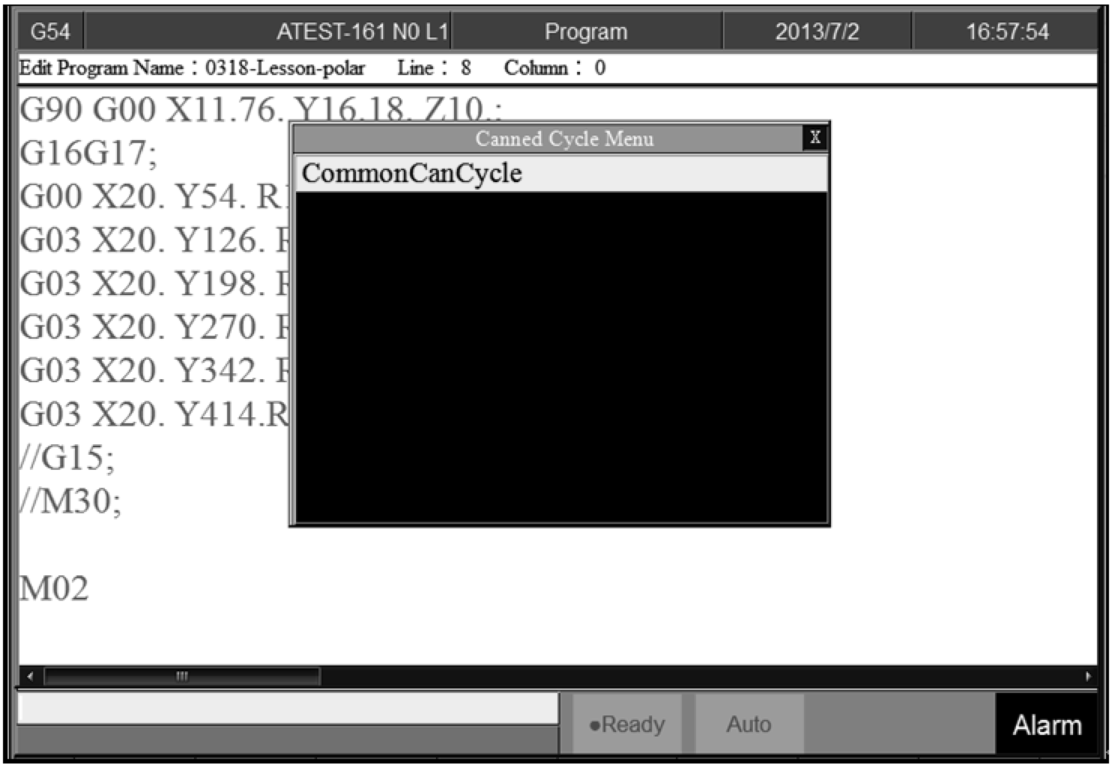
- Command
  - ✧ F2 Program→F3 Search/Replace→F3 Replace All
- Function
  - ✧ Replace all search text with input text.

1.4.3.4 Modify Setting

- Command
  - ✧ F2 Program→F3 Search/Replace→F4 Modify Setting
- Function
  - ✧ Reset 「Search/Replace」 string.

- Operation Method
  - ✧ Press F4「Modify Setting」to reset the「Search/Replace」content.

1.4.4 Can Cycle



G54
ATEST-161 N0 L1
Program
2013/7/2
16:57:54

Edit Program Name：0318-Lesson-polar Line：8 Column：0

```
G90 G00 X11.76. Y16.18. Z10.;
G16G17;
G00 X20. Y54. R
G03 X20. Y126.
G03 X20. Y198.
G03 X20. Y270.
G03 X20. Y342.
G03 X20. Y414.R
//G15;
//M30;

M02
```

Canned Cycle Menu
X
CommonCanCycle
•Ready
Auto
Alarm

- Command
  - ✧ F2 Program→F4 Can Cycle
- Function
  - ✧ Because SYNTEC system provides many kind of G code and different G code has each function. When editing the program, this function can help user to edit G code easily.

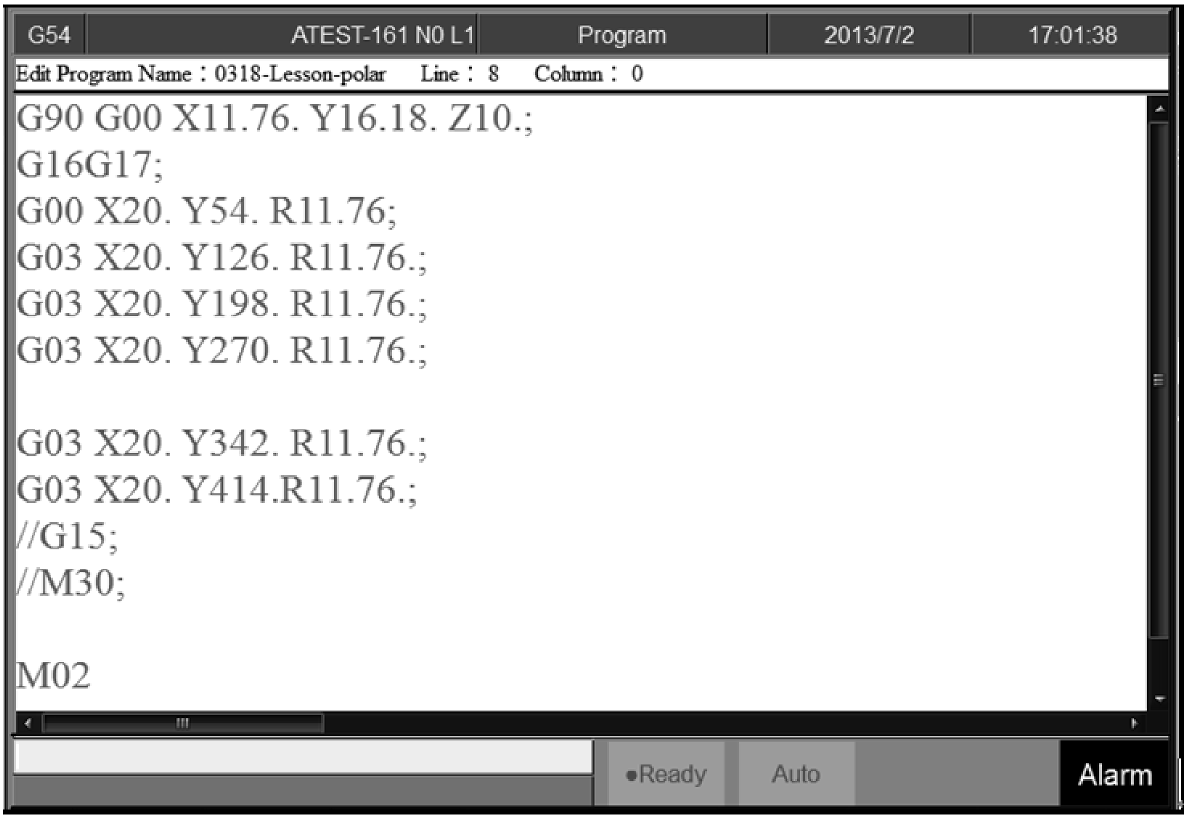
1.4.4.1 Insert Cycle

- Command
  - ✧ F2 Program→F4 Can Cycle→F1 Insert Cycle
- Function
  - ✧ Insert the required G code.
- Operation Method
  - ✧ Under the program edit mode, move the cursor to the desired location and press 「Insert Cycle」.
  - ✧ Follow the instruction and press 「OK」, the desired G code will insert into the next line of current cursor.

1.4.4.2 Edit Cycle

- Command
  - ✧ F2 Program→F4 Can Cycle→F2 Edit Cycle
- Function
  - ✧ Edit the current cycle.
- Operation Method
  - ✧ Move the cursor to the cycle and press 「Edit Cycle」,a modify page will show up. Modify the contents and press「OK」, the contents of the current cycle will change.

1.4.5 Block Copy


G54
ATEST-161 N0 L1
Program
2013/7/2
17:01:38
•Ready
Auto
Alarm

Edit Program Name：0318-Lesson-polar Line：8 Column：0

```
G90 G00 X11.76. Y16.18. Z10.;
G16G17;
G00 X20. Y54. R11.76;
G03 X20. Y126. R11.76.;
G03 X20. Y198. R11.76.;
G03 X20. Y270. R11.76.;

G03 X20. Y342. R11.76.;
G03 X20. Y414.R11.76.;
//G15;
//M30;

M02
```

- Path
  - ✧ F2 Program→F5 Block Copy
- Function
  - ✧ Select, cut, copy and paste more than one line of programs.

1.4.5.1 Start Line

- Command
  - ✧ F2 Program→F5 Block Copy→F1 Start Line
- Function
  - ✧ Define the start line of block.

1.4.5.2 End Line

- Command
  - ✧ F2 Program→F5 Block Copy→F2 End Line
- Function
  - ✧ Define the end line of block.

1.4.5.3 Block Cut

- Command
  - ✧ F2 Program→F5 Block Copy→F3 Block Cut
- Function
  - ✧ Cut the block that had been selected.

1.4.5.4 Block Copy

- Command
  - ✧ F2 Program→F5 Block Copy→F4 Block Copy
- Function
  - ✧ Copy the block that had been selected.

1.4.5.5 Block Paste

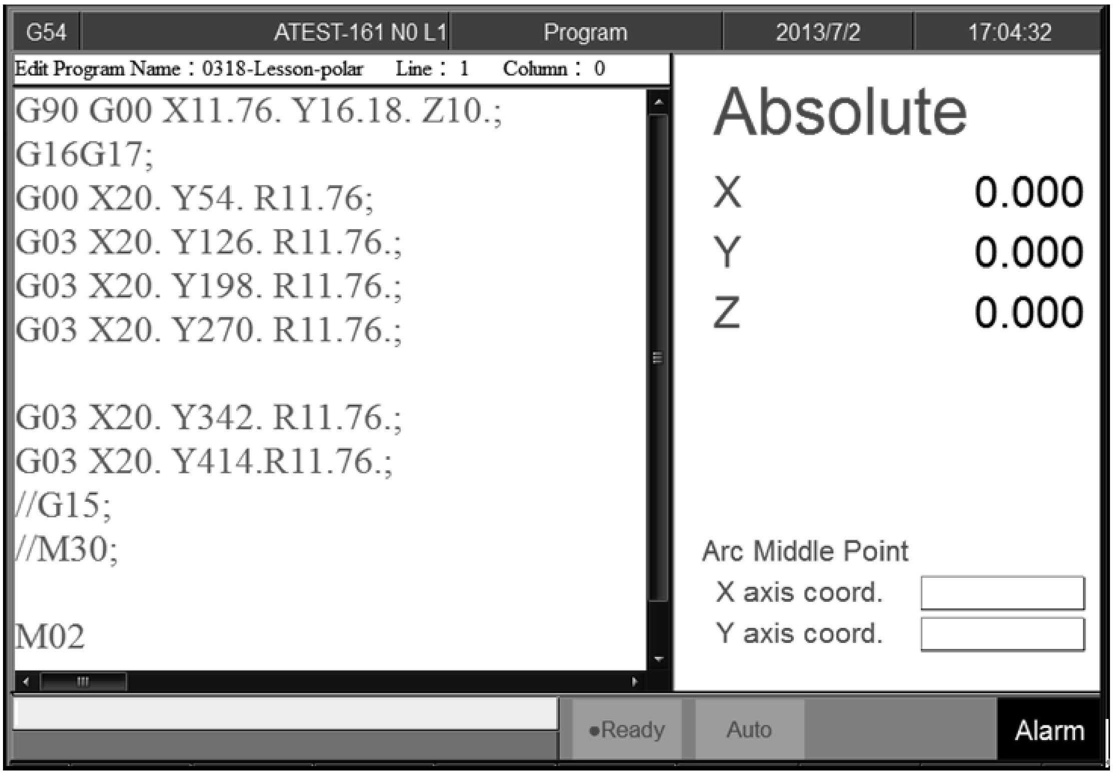
- Command
  - ✧ F2 Program→F5 Block Copy→F5 Block Paste
- Function

- ✧ Paste the block that had been 「Block Cut」and「Block Copy」.

- Operation Method
  - ✧ Move the cursor to the desire line and press「Start Line」and function key「End Line」enable.
  - ✧ Press【↑】【↓】【Page Up】【Page Down】 to select desired area.
  - ✧ Press「End Line」,block between「Start Line」and 「End Line」 will be selected.
    1. →Function key 「End Line」disable.
    2. →Function key 「Block Copy」enable.
    3. →Function key 「Block Cut」enable.
    4. →If 「Block Cut」being use, the whole block that had been highlight will be cut off.
    5. →Function key 「Block Copy」/「Block Cut」disable.
    6. →Function key 「Block Paste」enable.
  - ✧ Move the cursor to the desire location and press 「Block Paste」, the content that had been cut or copy will paste at the cursor location.
  - ✧ If 「Block Copy」is used, the block that had been selected will not disappear.
- Note
  - ✧ If 「Block Cut」is use, and do not paste the content immediately, the cut program will disappear.

✧ The contents of「Block Cut」can be pasted for only one time but the contents of 「Block Copy」 can be pasted for many times.

1.4.6 Teach


G54
ATEST-161 N0 L1
Program
2013/7/2
17:04:32
Arc Middle Point
X axis coord.
Y axis coord.
•Ready
Auto
Alarm

Edit Program Name：0318-Lesson-polar Line：1 Column：0

```
G90 G00 X11.76. Y16.18. Z10.;
G16G17;
G00 X20. Y54. R11.76;
G03 X20. Y126. R11.76.;
G03 X20. Y198. R11.76.;
G03 X20. Y270. R11.76.;

G03 X20. Y342. R11.76.;
G03 X20. Y414.R11.76.;
//G15;
//M30;

M02
```

```
Absolute
X        0.000
Y        0.000
Z        0.000
```

- Command

  ✧ F2 Program→F6 Teach

- Function

  ✧ Move the machine table with『MPG』/『JOG』/『INJOG』 to the destination and use 『Teach』 function, input the current absolute coordinate value to NC program.

  ✧ Omit the manual input problem.

1.4.6.1 Rapid Teach

- Command

  ✧ F2 Program→F6 Teach→F1 Rapid Teach

- Function
  - ✧ Add the current absolute coordinate as the value of 「G00 Rapid Traverse」 function in current program.

1.4.6.2 Line Cut Teach

- Command
  - ✧ F2 Program→F6 Teach→F2 Line Cut Teach
- Function
  - ✧ Add the current absolute coordinate as the value of 「G01 Linear Cutting」 function in current program.

1.4.6.3 Arc Cut Teach

- Command
  - ✧ F2 Program→F6 Teach→F3 Arc Cut Teach
- Function
  - ✧ Add current absolute coordinate as the input value of 「G02/G03 Circular Cutting」 function in current program.
- Operation Method
  - ✧ Move the worktable to the arc center and press 「Arc Cut Teach」, current absolute coordinate will be define as the arc center.
  - ✧ Move the worktable to the ending of the arc and press 「Arc Cut Teach」, current absolute coordinate will be define as the ending of the arc. Controller will automatically calculate the relation between middle and end point and determine whether to use G02 or G03.

The calculation result will be regarded as the input value of 「G02/G03 Circular Cutting」function.

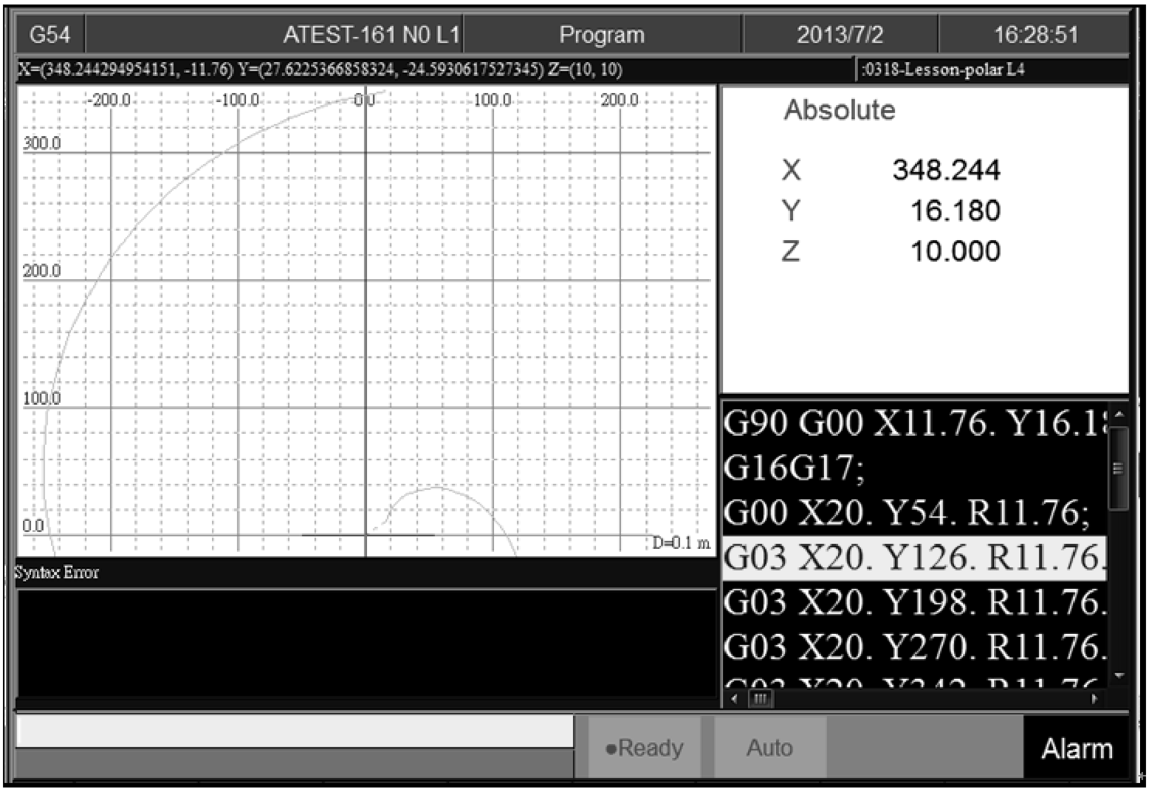
1.4.6.4 Cancel Arc Middle

- Command
  - ✧ F2 Program→F6 Teach→F4 Cancel Arc Middle
- Function
  - ✧ Clear the arc middle that had been set.
  - ✧ If the arc middle are not being set, this function will not enable.

1.4.6.5 Point Teach

- Command
  - ✧ F2 Program→F6 Teach→F5 Point Teach
- Function
  - ✧ Move the worktable to the arc center and press 「Point Teach」, current absolute coordinate will be input into current cursor location.

1.4.7 Simulation


G54
ATEST-161 N0 L1
Program
2013/7/2
16:28:51
-200.0
-100.0
0.0
100.0
200.0
300.0
200.0
100.0
0.0
D=0.1 m
●Ready
Auto
Alarm

X=(348.244294954151, -11.76) Y=(27.6225366858324, -24.5930617527345) Z=(10, 10)

:0318-Lesson-polar L4

Absolute
X 348.244
Y 16.180
Z 10.000

```
G90 G00 X11.76. Y16.1
G16G17;
G00 X20. Y54. R11.76;
G03 X20. Y126. R11.76
G03 X20. Y198. R11.76.
G03 X20. Y270. R11.76.
```

Syntax Error

- Command
  - ✧ F2 Program→F7 Simulation
- Function
  - ✧ Program simulation for the actual machining route.
  - ✧ Capability of debug.
  - ✧ Default display range will be the span of the full program.
  - ✧ Simulation setting can be modified by F5「simulate Setting」..

1.4.7.1 Step

- Command
  - ✧ F2 Program→F7 Simulation→F1 Step
- Function
  - ✧ Simulate the program block by block.
  - ✧ Monitor the variation of the coordinate for single block.

1.4.7.2 Continue

- Command
  - ✧ F2 Program→F7 Simulation→F2 Continue
- Function
  - ✧ System will scan all of the programs and then do the simulation.

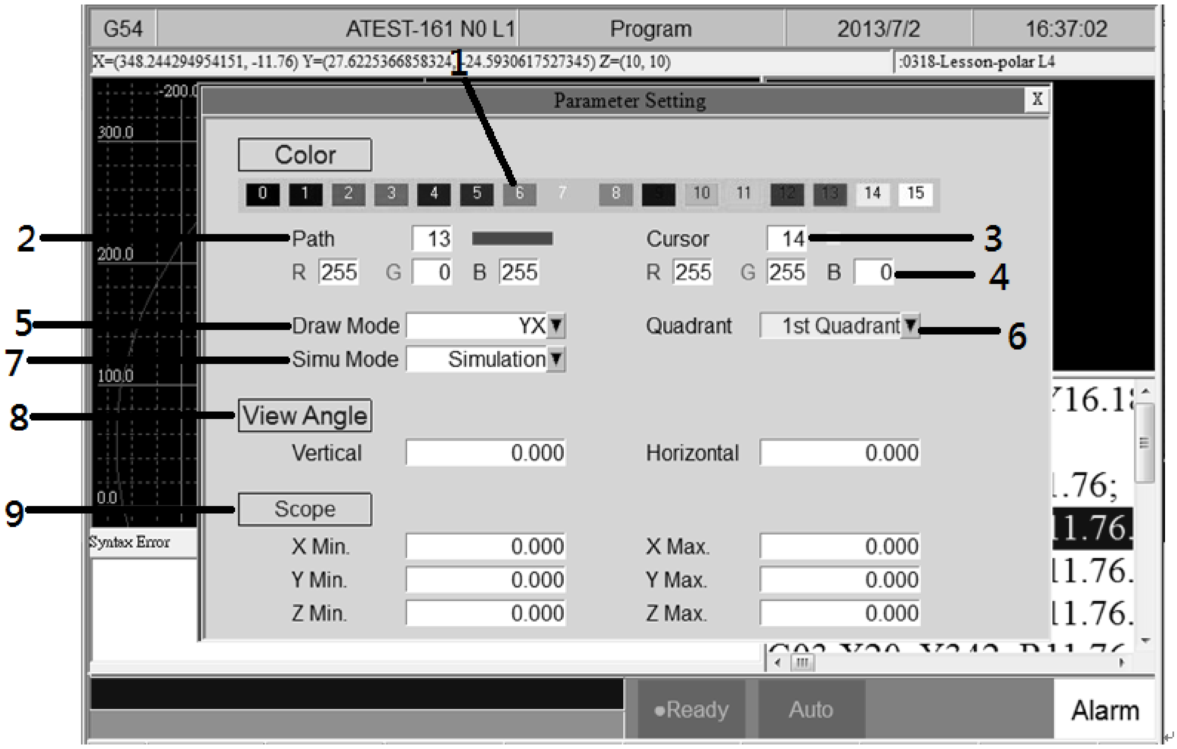
1.4.7.3 Zoom

- Command
  - ✧ F2 Program→F7 Simulation→F3 Zoom
- Function
  - ✧ Zoom in/out the simulation window.
- Operation Method
  - ✧ Press F3「Zoom」,there will be a block show up. Use【↑】【↓】【←】【→】can move the window up, down, left and right.
  - ✧ Use【Page Up】【Page Down】to change the zooming area.
  - ✧ Press【ENTER】to check the result.

1.4.7.4 Graph Reset

- Command
  - ✧ F2 Program→F7 Simulation→F4 Graph Reset
- Function
  - ✧ Reset the simulation result to default.

1.4.7.5 Simu. Setting



G54 | ATEST-161 N0 L1 | Program | 2013/7/2 | 16:37:02

X=(348.244294954151, -11.76) Y=(27.6225366858324, 24.5930617527345) Z=(10, 10) | :0318-Lesson-polar L4

Parameter Setting

Path 13 R 255 G 0 B 255

Cursor 14 R 255 G 255 B 0

Draw Mode YX

Quadrant 1st Quadrant

Simu Mode Simulation

Vertical 0.000

Horizontal 0.000

X Min. 0.000

X Max. 0.000

Y Min. 0.000

Y Max. 0.000

Z Min. 0.000

Z Max. 0.000

Syntax Error

1
2
3
4
5
6
7
8
9
-200.0
300.0
200.0
100.0
0.0
X
Color
0
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
View Angle
Scope
•Ready
Auto
Alarm

- Command
  - ✧ F2 Program→F7 Simulation→F5 Simu. Setting
- Function
  - ✧ Setting the relative simulation item.
- Simulation parameter
  1. Color
     - ◆ Provide 16 different colors (Setting 0~15).
  2. Path Color
     - ◆ Simulation path color.

- Provide 16 different colors (Setting 0~15).

3. Cursor Color
    - Color of cursor point
    - Provide 16 different colors (Setting 0~15).
4. RGB Value
    - Except the 16 default color, user can define the color by them self.
5. Draw Mode
    - User can define the profile simulate plane.

- Plane can define are as below.

◆ XYZ	◆ YX
◆ XY	◆ ZY
◆ YZ	◆ XZ
◆ ZX	◆

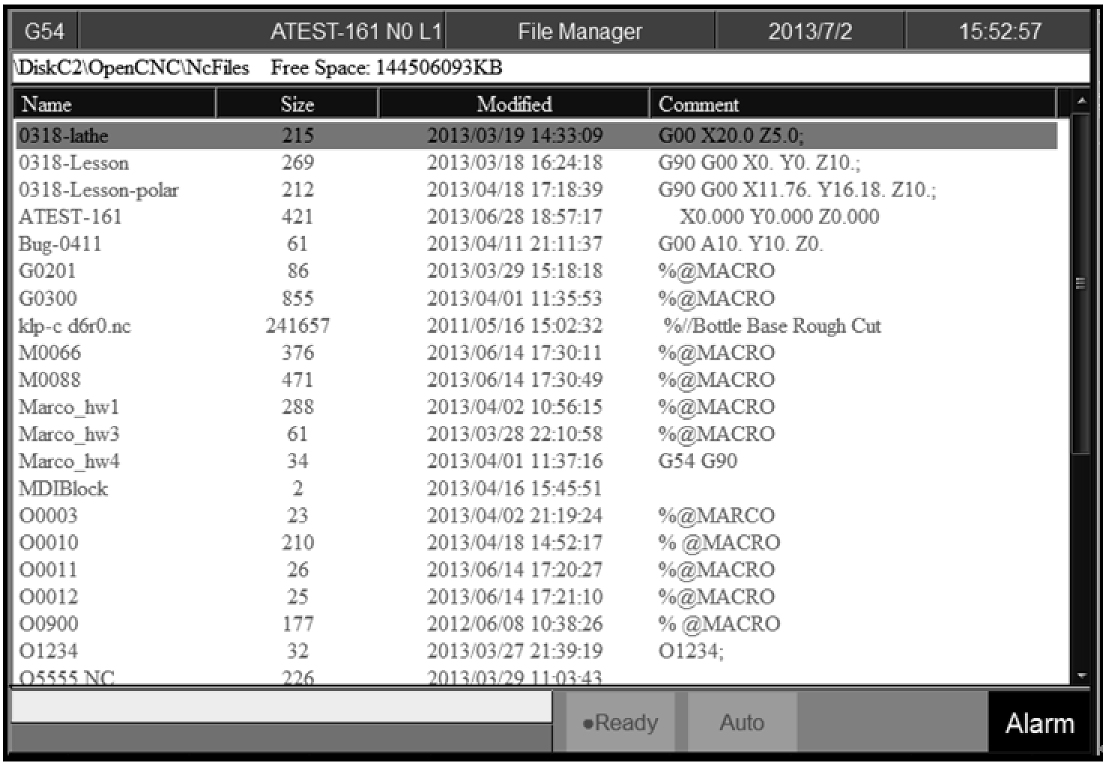
6. Setting quadrant
    - User can define the quadrant of simulate plane.
    - Quadrant can define are as below.
        - First
        - Second
        - Third
        - Fourth

7. Simulate Mode
    - ◆ Setting profile simulate method.
    - ● Simulation
        - ◆ When user go to the 『Monitor』 page, simulation will show up automatically.
        - ◆ System will scan the whole program and detect the simulation boundary, and then simulation will execute. No need to define the simulate boundary.
    - ● Direct Draw
        - ◆ When user go to the 『Monitor』page, cursor will show up but simulation will not execute automatically.
        - ◆ User need to define the simulation boundary first.
        - ◆ When the machining starts, cursor will follow up as well.
    - ● Not Simulation
        - ◆ Close the simulation function.
8. View Angle Setting
    - ◆ Under XYZ draw mode, by setting this parameter, simulation will show up with 3D simulation.
    - ◆ View angle can define are as below.
        - ◆ Vertical
        - ◆ Horizontal

9. Scope

- Scope can define are as below.
  - Minimum
    - X-Axis
    - Y-Axis
    - Z-Axis
  - Maximum
    - X-Axis
    - Y-Axis
    - Z-Axis

1.4.8 File Manager



\DiskC2\OpenCNC\NcFiles Free Space: 144506093KB

Name	Size	Modified	Comment
0318-lathe	215	2013/03/19 14:33:09	G00 X20.0 Z5.0;
0318-Lesson	269	2013/03/18 16:24:18	G90 G00 X0. Y0. Z10.;
0318-Lesson-polar	212	2013/04/18 17:18:39	G90 G00 X11.76. Y16.18. Z10.;
ATEST-161	421	2013/06/28 18:57:17	X0.000 Y0.000 Z0.000
Bug-0411	61	2013/04/11 21:11:37	G00 A10. Y10. Z0.
G0201	86	2013/03/29 15:18:18	%@MACRO
G0300	855	2013/04/01 11:35:53	%@MACRO
klp-c d6r0.nc	241657	2011/05/16 15:02:32	%//Bottle Base Rough Cut
M0066	376	2013/06/14 17:30:11	%@MACRO
M0088	471	2013/06/14 17:30:49	%@MACRO
Marco_hw1	288	2013/04/02 10:56:15	%@MACRO
Marco_hw3	61	2013/03/28 22:10:58	%@MACRO
Marco_hw4	34	2013/04/01 11:37:16	G54 G90
MDIBlock	2	2013/04/16 15:45:51	
O0003	23	2013/04/02 21:19:24	%@MARCO
O0010	210	2013/04/18 14:52:17	% @MACRO
O0011	26	2013/06/14 17:20:27	%@MACRO
O0012	25	2013/06/14 17:21:10	%@MACRO
O0900	177	2012/06/08 10:38:26	% @MACRO
O1234	32	2013/03/27 21:39:19	O1234;
O5555.NC	226	2013/03/29 11:03:43	

- Command
  - ✧ F2 Program→F8 File Manager
- Function
  - ✧ This function key can manage all of the NC files within the data storage device. The device can be setting with Pr3213.
- Operation Method
  - ✧ Use【↑】【↓】on the key pad to move the cursor to anywhere on the screen for editing purpose.
  - ✧ With 【Page Up】【Page Down】to switch the cursor between pages.
  - ✧ Press【ENTER】on the key pad, to assign the current cursor file as the execute file, screen will show up with

the program content and can enable to edit the program.

1.4.8.1 New File

- Command
  - ✧ F2 Program→F8 File Manager→F1 New File
- Function
  - ✧ Open a new file, that file will be the current edit file.
- Operation Method
  - ✧ Press「New File」function key, a dialog box will appear, enter the new file name and press 【ENTER】.
- Note
  - ✧ Default file name has no file extension. If user want to create a new file with file extension such as *.NC, just enter the extension (*.NC) as well.
  - ✧ The length of file name cannot be longer than 32 characters (include file extension)

1.4.8.2 Copy File

- Command
  - ✧ F2 Program→F8 File Manager→F2 Copy File
- Function
  - ✧ Copy the file that remarked by cursor.
- Operation Method
  - ✧ Use【↑】【↓】to move the cursor to the file that want to copy.

- ✧ Press「Copy File」 function key.
- ✧ A dialog box will appear, enter the new file name.

- Note
  - ✧ Default file name has no file extension. If user want to create a new file with file extension such as *.NC, just enter the extension (*.NC) as well.
  - ✧ The length of file name cannot be longer than 32 characters (include file extension)

1.4.8.3 Delete File

- Command
  - ✧ F2 Program→F8 File Manager→F3 Delete File
- Function
  - ✧ Delete file that remarked by cursor.
- Operation Method
  - ✧ Press F3「Delete File」, check box will show up in front of the NC file within the「File Manager」monitor page. Use【↑】【↓】to select the delete file.
- Sub-function Key
  - ✧ Select: Select file, can select more than one file and also can cancel the selection of one file.
  - ✧ Select All: Select all file.
  - ✧ Cancel Select: Deselect all files.
  - ✧ Delete File: Delete all of the selected files.
  - ✧ Delete All: Delete all file within data storage device.
- Note

✧ Current Programming and machining file cannot be deleted.

1.4.8.4 File Import

- Command
  - ✧ F2 Program→F8 File Manager→F4 File Import
- Function
  - ✧ Import outer file into controller
- Function Page Explanation
  - ✧ The upper block shows the outer devices selection with the following choice.
    - ■USBDisk
    - ■DiskA
    - ■Network
    - ■USBDisk2
  - ✧ Left column shows data structure of the outer device.
  - ✧ Right column shows data structure of the inner storage of the controller.
- Sub-function Explanation
  - ✧ Copy: Copy the remarked file from the outer device to the controller.
  - ✧ Select: Select or deselect each file. (Not available for folder)
  - ✧ Select All: Select all files.
  - ✧ Cancel Select: Deselect all files.
  - ✧ Device Change: Change outer device selection.
- Operation Method
  - ✧ Press F1「File Import」, a dialog box will appear.
  - ✧ Default outer device is USB Disk.

- ✧ If you want to change the outer device, press F5「Device Change」,switch the cursor to the desire device and press【Enter】, then the left column data structure will change and show the data structure of selected device.
- ✧ Use【↑】【↓】to select file.
- ✧ Move the cursor to the import file and press F2「Select」or 【Space】to remark file.
- ✧ After remarked all of the import file, press F1「Copy」then all of the remarked file will import into controller.

1.4.8.5 File Export

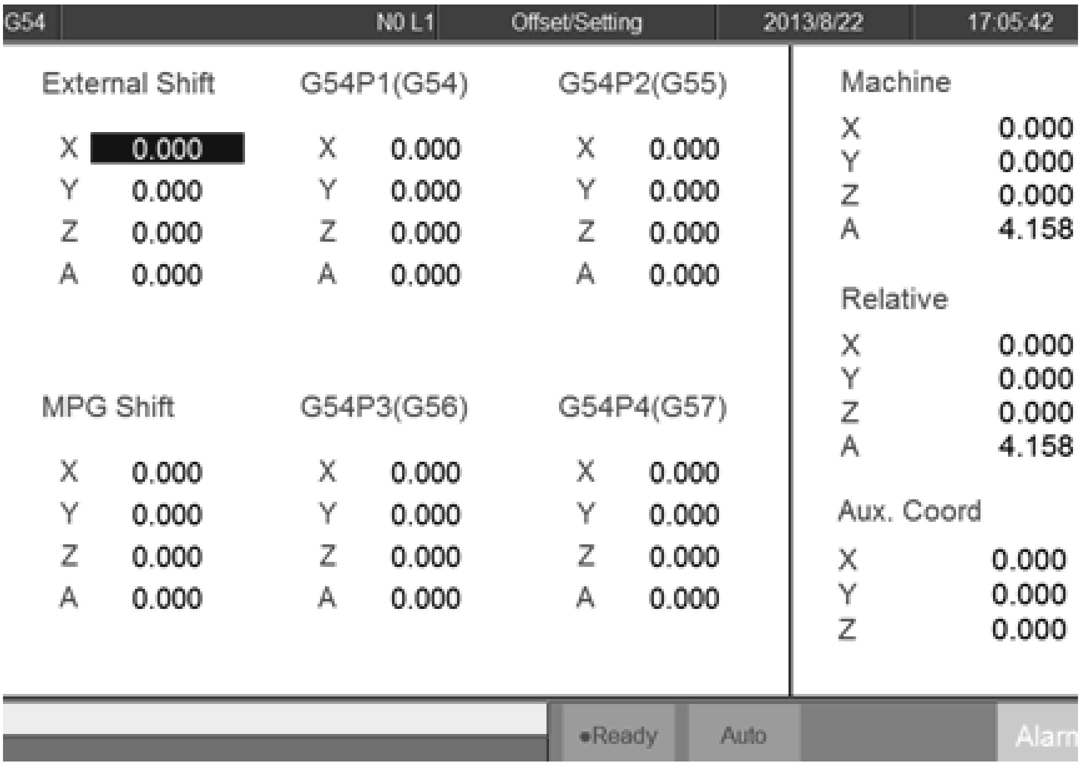
- Command
  - ✧ F2 Program→F8 File Manager→F5 File Export
- Function
  - ✧ Export file within controller to outer device.
- Function Page Explanation
  - ✧ The upper block shows the outer devices selection with the following choice.
    - ■USBDisk
    - ■DiskA
    - ■Network
    - ■USBDisk2
  - ✧ Left column shows data structure of the outer device.
  - ✧ Right column shows data structure of the inner storage of the controller.
- Sub-function Explanation
  - ✧ Copy: Copy the remarked file from the outer device to the controller.
  - ✧ Select: Select or deselect each file. (Not available for folder)
  - ✧ Select All: Select all files.
  - ✧ Cancel Select: Deselect all files.
  - ✧ Device Change: Change outer device selection.
- Operation Method
  - ✧ Press「File Export」, a dialog box will appear.
  - ✧ Default outer device is USBDisk.

- ✧ If you want to change the destination device, press F5「Device Change」, switch the cursor to the desire device and press【Enter】, then the below left column data structure will change, according to the selected device .
- ✧ Use【↑】【↓】to select file.
- ✧ Move the cursor to the export file and press F2「Select」or【Space】to remark file.
- ✧ After remarked all of the export file, press F1「Copy」then all of the remarked file will export from controller to outer device.

1.4.8.6 Execute

- Command
  - ✧ F2 Program→F8 File Manager→F8 Execute
- Function
  - ✧ Execute current program. and also change the screen to the 「monitor」 page.
- Operation Method
  - ✧ Use【↑】【↓】to select file, and then press「Execute」the selected file will be the executive file.
- Note
  - ✧ This function is invalid when machining.

1.5 Offset/Setting



G54 | N0 L1 | Offset/Setting | 2013/8/22 | 17:05:42

External Shift		G54P1(G54)		G54P2(G55)	
X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000
A	0.000	A	0.000	A	0.000

MPG Shift		G54P3(G56)		G54P4(G57)	
X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000
A	0.000	A	0.000	A	0.000

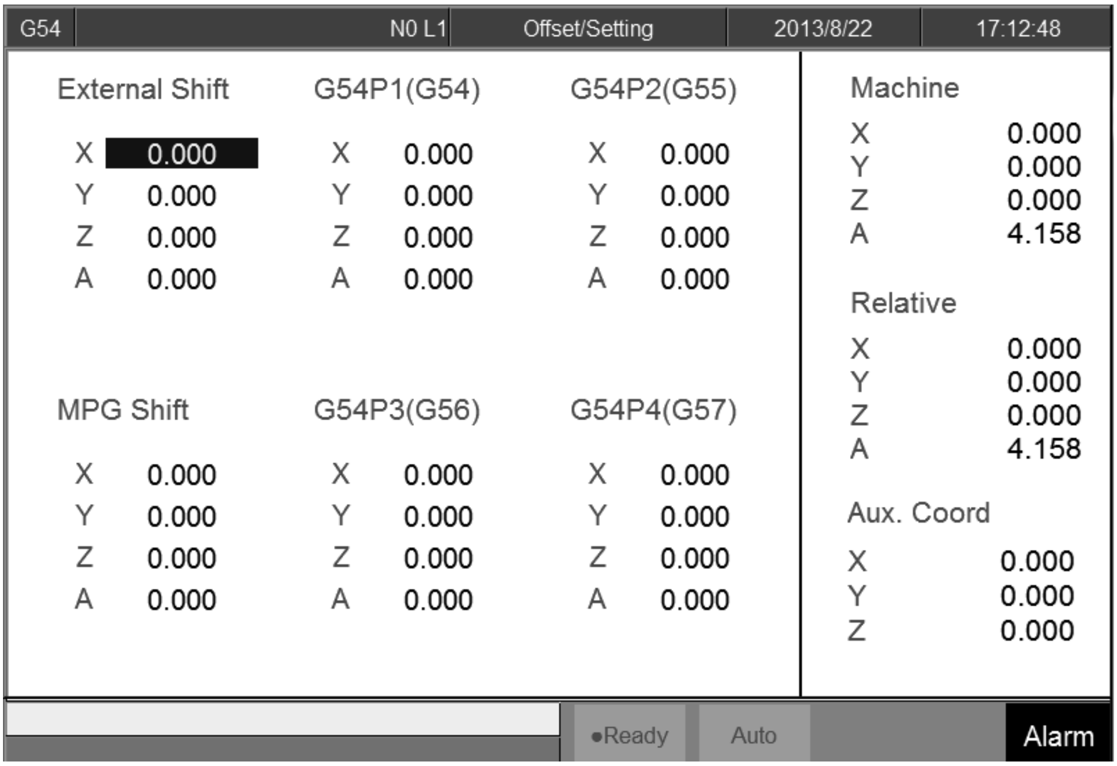
Machine	
X	0.000
Y	0.000
Z	0.000
A	4.158

Relative	
X	0.000
Y	0.000
Z	0.000
A	4.158

Aux. Coord	
X	0.000
Y	0.000
Z	0.000

- Command
  - ✧ F3 Offset/Setting
- Function
  - ✧ User can set up the coordinate system and tool offsets with this function.
  - ✧ With the function key 【Offset/Setting】can switch the page quickly to the Offset/Setting page.

1.5.1 Workpiece Cord.



	External Shift		G54P1(G54)		G54P2(G55)
X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000
A	0.000	A	0.000	A	0.000

	MPG Shift		G54P3(G56)		G54P4(G57)
X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000
A	0.000	A	0.000	A	0.000

	Machine
X	0.000
Y	0.000
Z	0.000
A	4.158

	Relative
X	0.000
Y	0.000
Z	0.000
A	4.158

	Aux. Coord
X	0.000
Y	0.000
Z	0.000

- Command
  - ✧ F3 Offset/Setting→F1 Workpiece Cord.
- Function
  - ✧ For workpiece coordinate setting purpose.
  - ✧ System default will be G54 if there are no specific declaration with G54~G59.10 in the NC code.
  - ✧ The External Shift will include into all the workpiece coordinate.
- Operation Method:
  - ✧ Move the cursor with【↑】【↓】【←】【→】.
  - ✧ Use【PageUp】【PageDown】to switch the pages.
- Note
  - ✧ After setting the workpiece coordinate, user need to check the tool length again.

1.5.1.1 Mach. Coord. teach

- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F1 Mach. Coord. Teach
- ✧ Function
  - ✧ Set current mechanical coordinate value into work piece coordinate.
- Operation Method
  - ✧ Move the worktable to the target location.
  - ✧ Move the cursor to the relate work piece coordinate and press 「Mach. Coord. Teach」
  - ✧ Original value of workpiece coordinate will be replaced by the corresponding mechanical coordinate.
- Example
  - ✧ Current mechanical coordinate of X axis is 5.000
  - ✧ Current value of X axis of G54 is 0.000
  - ✧ Move the cursor to G54 X axis.
  - ✧ Press 「Mach. Coord. Teach」, the value of X axis of G54 becomes 5.000

1.5.1.2 Rel. Coord. Teach

- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F2 Rel. Coord. Teach
- Function

  - ✧ Set current relative coordinate value into work piece coordinate.

- Operation Method
  - ✧ Move the worktable to the target location.
  - ✧ Move the cursor to the relate work piece coordinate and press 「Rel. Coord. Teach」
  - ✧ Original value of workpiece coordinate will be replaced by the corresponding relative coordinate.
- Example
  - ✧ Current relative coordinate of X axis is 5.000
  - ✧ Current value of X axis of G54 is 0.000
  - ✧ Move the cursor to G54 X axis.
  - ✧ Press「Rel. Coord. Teach」,the coordinate of G54 X axis become 5.000

1.5.1.3 Aux. Coord. Teach

- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F3 Aux. Coord. Teach
- Function
  - ✧ Set the current cursor located work piece coordinate value as the aux. coordinate value.
  - ✧ Aux. value appears after using middle function.
- Operation Method
  - ✧ Using middle function to calculate the aux. coordinate value.

- ✧ Move the cursor to the relate work piece coordinate and press 「Aux. Coord. Teach」
- ✧ Original value of workpiece coordinate will be replaced by the corresponding Aux. coordinate.

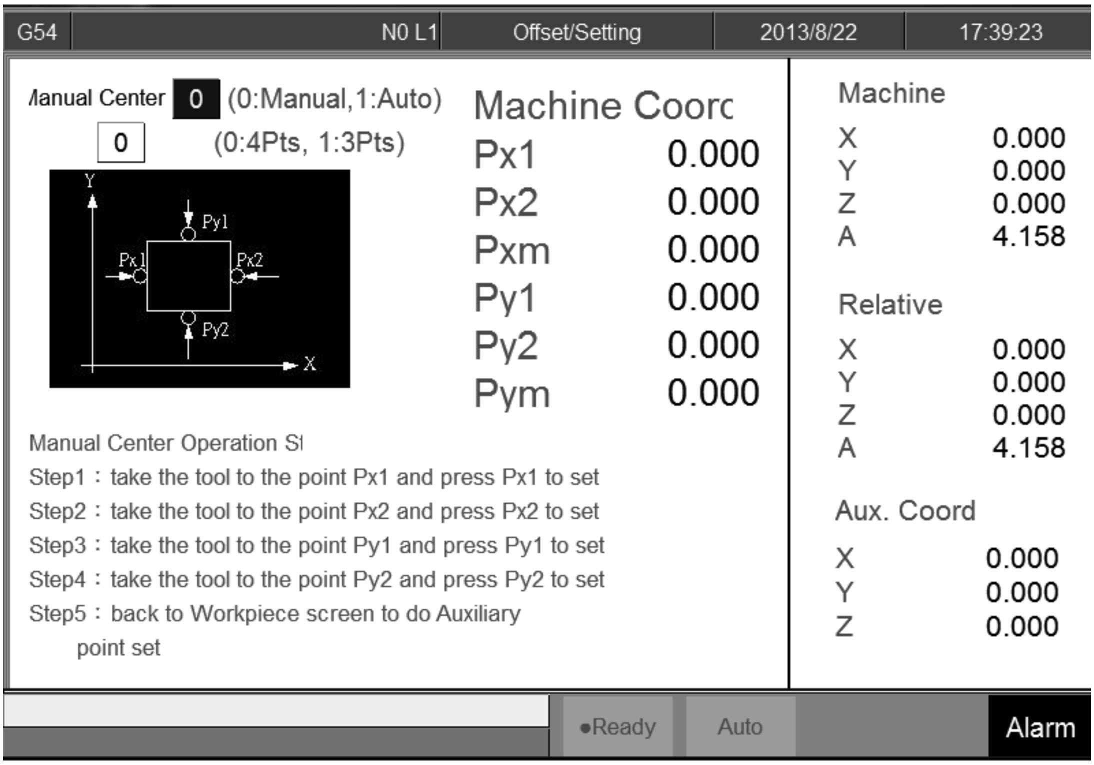
- ● Example
  - ✧ Current mechanical coordinate of X axis is 5.000
  - ✧ Current value of X axis of G54 is 0.000
  - ✧ Move the cursor to G54 X axis.
  - ✧ Press 「Aux. Coord. Teach」, the value of X axis of G54 become 5.000

1.5.1.4 Inc. Input

- ● Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F4 Inc. Input
- ● Function
  - ✧ Add the work piece coordinate value with the manual input value and restore into work piece coordinate again.
- ● Operation Method
  - ✧ Move worktable to the target location.
  - ✧ Input increment value.
  - ✧ Move the cursor to the work piece coordinate and press 「Inc. Input」
  - ✧ Work piece coordinate will be replaced by the cursor located coordinate +input value.
- ● Example

- ✧ Current mechanical coordinate of X axis is 5.000
- ✧ Move cursor to the workpiece coordinate G54 of X axis
- ✧ Key in 10.000
- ✧ Press 「Inc. Input」
- ✧ The coordinate of G54 X axis become 15.000

1.5.1.5 Middle Func.



G54 | N0 L1 | Offset/Setting | 2013/8/22 | 17:39:23

anual Center 0 (0:Manual,1:Auto)

0 (0:4Pts, 1:3Pts)

Machine Coorc

Px1 0.000

Px2 0.000

Pxm 0.000

Py1 0.000

Py2 0.000

Pym 0.000

Manual Center Operation St

Step1：take the tool to the point Px1 and press Px1 to set

Step2：take the tool to the point Px2 and press Px2 to set

Step3：take the tool to the point Py1 and press Py1 to set

Step4：take the tool to the point Py2 and press Py2 to set

Step5：back to Workpiece screen to do Auxiliary point set

Machine	
X	0.000
Y	0.000
Z	0.000
A	4.158
Relative	
X	0.000
Y	0.000
Z	0.000
A	4.158
Aux. Coord	
X	0.000
Y	0.000
Z	0.000

- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F5 Middle Func.
- Function
  - ✧ This function can help to correlate the middle point of the work piece.
  - ✧ Input the number that Middle Func caculate result to G54~G59 work piece coordinate.
  - ✧ Including manual and auto setting method.

1.5.1.5.1 Manual

- Function
  - ✧ Set the Middle Func. as 0
  - ✧ User controls the machine by MPG, and then moves the tip of 3D machine to the X&Y end point of the workpiece. This system will calculate the center point of the workpiece automatically.
- Operation method
  - ✧ Move the machine by MPG, touching Px1 point in tis figure and then press PX1 Set. The system will record the current mechanical coord. to Px1. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm and Aux. X position.
  - ✧ Move the machine by MPG, touching Px2 point in this figure and then press PX2 Set. The system will record the current mechanical coord. to Px2. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm and Aux. X position.
  - ✧ Move the machine by MPG, touching Py1 point in tis figure and then press PY1 Set. The system will record the current mechanical coord. to Py1. It will also compute the middle point of Py1 and Py2 and puts the result on Pym and Aux. Y position.
  - ✧ Move the machine by MPG, touching Py2 point in this figure and then press PY2 Set. The system will record the current mechanical coord. to Py2. It will also

compute the middle point of Py1 and Py2 and puts the result on Pym and Aux. Y position.

- ✧ Now, the values of Pxm and Pym are the middle point of the workpiece.
- ✧ In the WorkPiece Cord. Screen, move the cursor to the coord you want to set, press F3 「Aux. Coord. Teach」 insert the Aux. Coord value then this system will set the value according to the Aux Coord.

1.5.1.5.2 Auto

- Function
  - ✧ Set the Middle Func. as 1
  - ✧ Auto middle func. is different form Manual func. User only needs to enter the dimension of the workpiece and enter the boundary coordinate. Move the machine to the start point, system will find out the middle point automatically.
  - ✧ Auto middle func. is different form Manual func. User only needs to enter the dimension of the workpiece and enter the boundary coordinate. Move the machine to the start point, system will find out the middle point automatically.
- Parameter description
  - ✧ Length I: Workpiece X dimension length
  - ✧ Width J: Workpiece Y dimension length
  - ✧ Safe Distance H: This is the distance between start point P2 and the workpiece no matter X or Y direction.

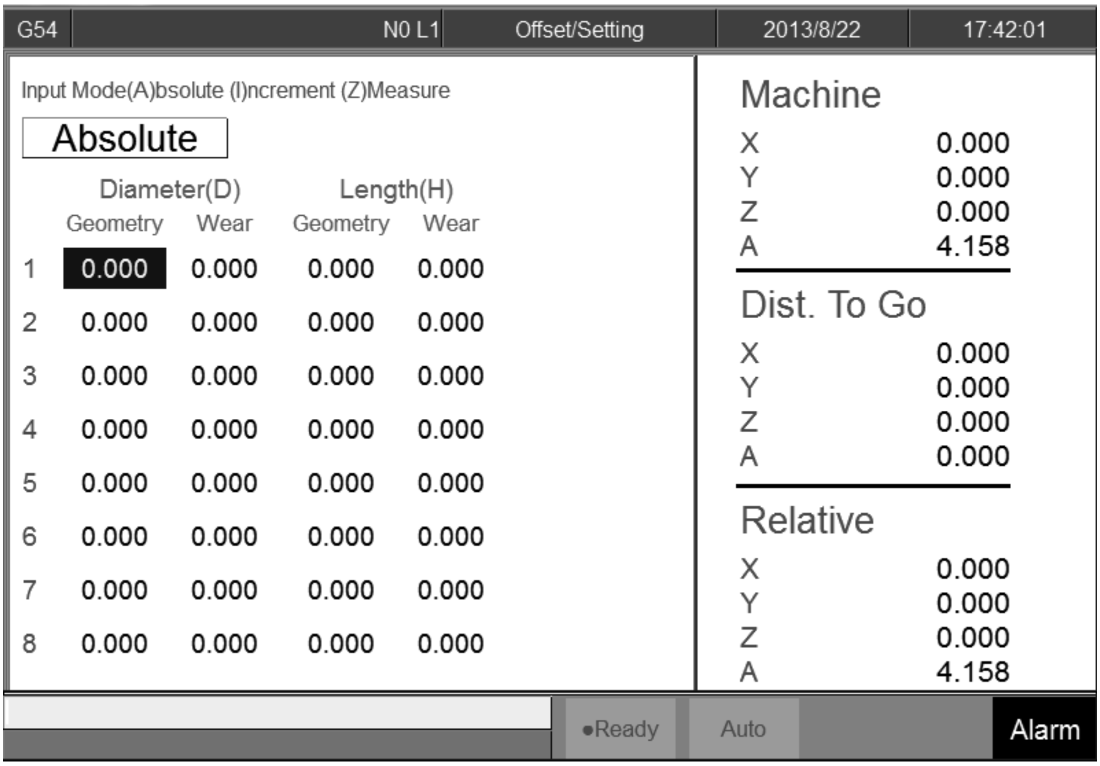
  - Feedrate F: Auto. center detect speed.
  - Z Coordinate: This is the safety distance prevent the contact between the tool and work piece while moving.
- Operation method
  - Move the tool to the Z height, the safe distance between tool and work piece. After that, press Z Coordinate Set, then system will record the present Z coordinate value as the safe distance.
  - Move the tool down to P2 under the work piece, the start point for auto center.
  - Press Auto center start, system will move the tool according to the setting data to contact with work piece and show the coordinate on the screen. It will also calculate the X Y center point of the work piece.
  - Go back to WorkPiece Cord. Move the cursor the workpiece coordinate position, press F3 「Aux. Coord. Teach」 insert the Aux. Cord value then this system will set the value according to the Aux Cord.

1.5.1.6 Mach. Coord Inc. Teach

- Command
  - F3 Offset/Setting→F1 Work Piece Cord.→F6 Mach. Coord Inc. Teach
- Function

  - ✧ By the Mach. Coord Inc. Teach, set the current cursor located work piece coordinate value as the new start coordinate value.
- Operation Method
  - ✧ Move the worktable to the destination.
  - ✧ Move the cursor to the relate work piece coordinate and press 「Mach. Coord Inc. Teach」
  - ✧ Current cursor located work piece coordinate will be replaced by the cursor coordinate+input number.
- Example
  - ✧ Current mechanical coordinate of X axis is 5.000
  - ✧ Current workpiece coordinate G54 of X axis
  - ✧ Key in 10,000
  - ✧ Press 「Mach. Coord Inc. Teach」
  - ✧ the coordinate of G54 X axis become 15.000

1.5.2 Tool Set



G54 | N0 L1 | Offset/Setting | 2013/8/22 | 17:42:01

Input Mode(A)bsolute (I)ncrement (Z)Measure

Absolute

	Diameter(D)		Length(H)	
	Geometry	Wear	Geometry	Wear
1	0.000	0.000	0.000	0.000
2	0.000	0.000	0.000	0.000
3	0.000	0.000	0.000	0.000
4	0.000	0.000	0.000	0.000
5	0.000	0.000	0.000	0.000
6	0.000	0.000	0.000	0.000
7	0.000	0.000	0.000	0.000
8	0.000	0.000	0.000	0.000

Machine

X	0.000
Y	0.000
Z	0.000
A	4.158

Dist. To Go

X	0.000
Y	0.000
Z	0.000
A	0.000

Relative

X	0.000
Y	0.000
Z	0.000
A	4.158

- Command
  - ✧ F3 Offset/Setting→F2 Tool Set
- Function
  - ✧ Switch to compensate setting page.
  - ✧ Actual G41/G42 compensation value = Geometry + Wear diameter
  - ✧ Actual G43/G44 compensation value = Geometry + Wear length
- Function of parameter
  - ✧ Geometry: G41/G42 tool radius Dn compensation setting(not diameter)
  - ✧ Wear of geometry: Tiny geometry modification of tool.
  - ✧ Length: G43/G44 tool lengths Hn compensation.
  - ✧ Length wear: Tiny length modification of tool.

- Operation method
  - ✧ With 【↑】【↓】【←】【→】key to move the cursor.
  - ✧ 【PageUp】【PageDown】 to switch the page.
  - ✧ Key in method: Absolute, Increment, Measure method
- Absolute method:
  - ■Press 「A」 and press 「Enter」
  - ■The value where the cursor is will be set as the input value.
- Increment method
  - ■Press 「I」 and then press 「Enter」.
  - ■The value where the cursor is will be set as input value + cursor value.
- Measure method
  - ■Press「Z」 and then press「Enter」, the value where the cursor is will be set as current Z axis relative coordinate value.
  - ■Press F2 Set Tool Mach. Coord., the value where the cursor is will be set as current Z axis machine coordinate value.(effective version from 2.2.0)
  - ■Press F3 Set Tool Rel. Coord., the value where the cursor is will be set as current Z axis relative coordinate value. (effective version from 2.2.0)
- Note
  - ✧ When the tool length had been set, related tool wear will become zero.

✧ This Setting is disabled during machining condition.

1.5.2.1 Clean Z relative coordinate

- Command
    - ✧ F3 Offset/Setting→F2 Tool Set.→F1 Clear Z Coord
- Function
    - ✧ Clean the relative value of Z coord.

1.5.2.2 Tool No.

- Command
    - ✧ F3 Offset/Setting→F2 Tool Set→F7Tool No.
- Function
    - ✧ Refer the chapter 3.3.3.5

1.5.2.3 Tool Life Manag.

- Command
    - ✧ F3 Offset/Setting→F2 Tool Set→F8 Tool Life Manag.
- Function
    - ✧ Refer the chapter 3.3.3.6

1.5.3 Tool Tip Measure

✧ F3 Offset/Setting→F3 Tool Tip Measure

- Function
    - ✧ When the machine is equip with tool detection equipment, via the parameter setting, can move the machine table to the detective location and execute the top tip measure.
    - ✧ According to different situation, here are three different method

      -Single tool, single workpiece

      -Single tool, multi workpiece

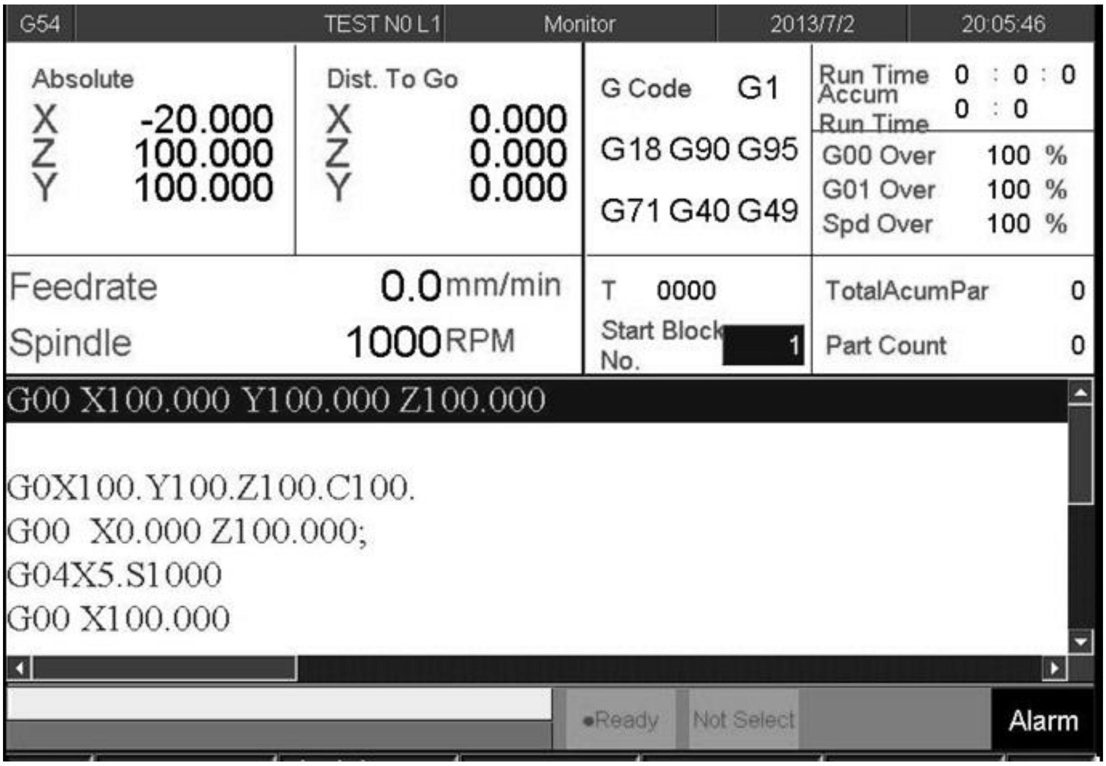
-multi tool, multi workpiece

-Refer the chapter 3.3.3.3.

1.5.4 User Parameter Setting

- Command
  - ✧ F3 Offset/Setting→F4 User Parameter Setting
- Function
  - ✧ SYNTEC controller provided user to set the related machining parameter by them self.
- Function of parameter
  - ✧ Detail explanation please reference「Mill Parameter Manual」.

1.6 Monitor


G54
TEST N0 L1
Monitor
2013/7/2
20:05:46
●Ready
Not Select
Alarm

Absolute
X -20.000
Z 100.000
Y 100.000

Dist. To Go
X 0.000
Z 0.000
Y 0.000

G Code G1
G18 G90 G95
G71 G40 G49

Run Time 0 : 0 : 0
Accum Run Time 0 : 0
G00 Over 100 %
G01 Over 100 %
Spd Over 100 %

Feedrate 0.0 mm/min
Spindle 1000 RPM

T 0000
Start Block No. 1

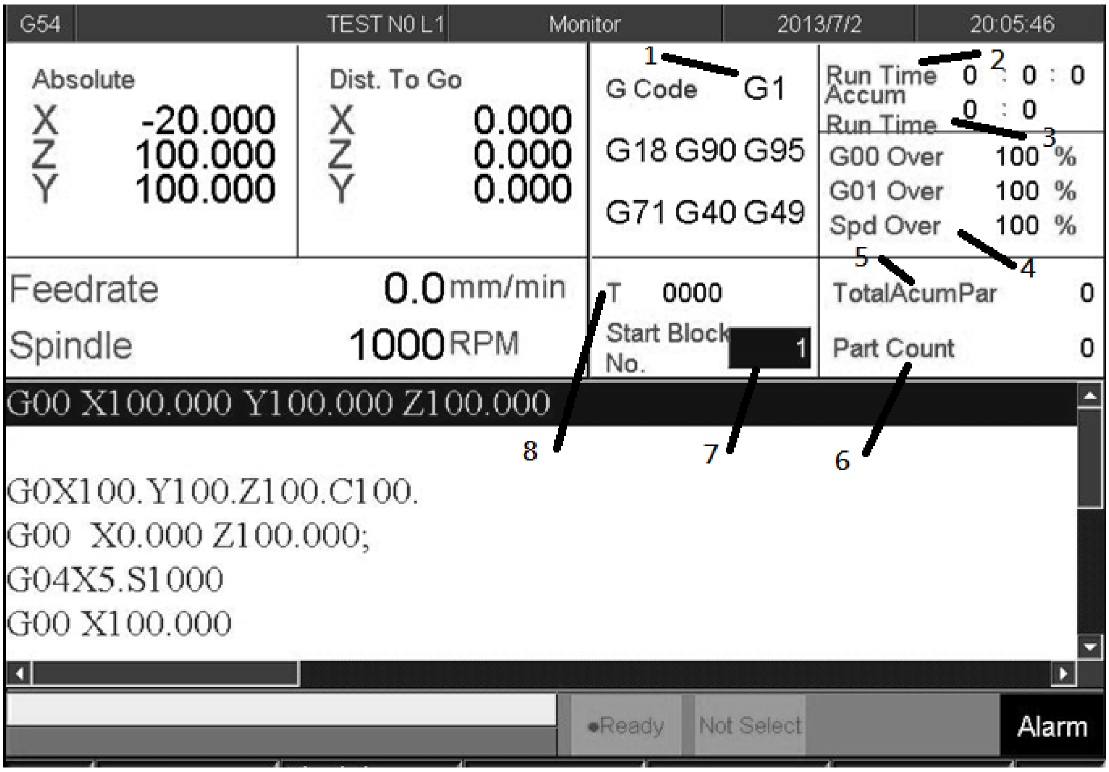
TotalAcumPar 0
Part Count 0

```
G00 X100.000 Y100.000 Z100.000

G0X100.Y100.Z100.C100.
G00  X0.000 Z100.000;
G04X5.S1000
G00 X100.000
```

- Command:
    - ✧ F4 Monitor
- Function
    - ✧ This page monitors key machining information during machining process.

1.6.1 Monitor Area of Machining Information



G54 | TEST N0 L1 | Monitor | 2013/7/2 | 20:05:46

Absolute
X -20.000
Z 100.000
Y 100.000

Dist. To Go
X 0.000
Z 0.000
Y 0.000

G Code G1
G18 G90 G95
G71 G40 G49

Run Time 0 : 0 : 0
Accum Run Time 0 : 0
G00 Over 100 %
G01 Over 100 %
Spd Over 100 %

Feedrate 0.0mm/min
Spindle 1000RPM

T 0000
Start Block No. 1

TotalAcumPar 0
Part Count 0

```
G00 X100.000 Y100.000 Z100.000

G0X100.Y100.Z100.C100.
G00  X0.000 Z100.000;
G04X5.S1000
G00 X100.000
```

1.6.1.1 Machine Status Monitor

- Machine information
- Absolute coordinate
- Distance to go
- Feed rate
- Spindle

1.6.1.2 Program Monitor Block

✧ This block will display current machining program

✧ Yellow bar indicate to the current running block.

1.6.1.3 Machining Information

- Function

✧ It is overlap with 「Process Setting」.

✧ Press 「Parameter Set」 can switch display information.

- Description
  1. G Code
     - It will show the G code under machining
  2. Run Time Accum
     - Processing time for this workpiece now
  3. Run Time
     - Total machining process time
  4. Percentage ratio
     - G00 percentage
     - G01 percentage
     - Spindle speed percentage
  5. Total Acum Par
     - Total work pieces that had been finished.
     - System won’t do any initialized action automatically.
     - If you want to do the initialization by manual, press 「Parameter Set」 switch to 「Part count」, set the 「Total Acum Par」 as 0.Part Count
  6. Part Count
     - Count no. will begin from zero when the program is running.
     - Total work pieces number machined by CNC
  7. Start Block
     - We can set the start block of machining process.
     - n: Set the start line number as n. (Ex. 20)
     - L+n: Set the start line number as n. (Ex.L20)
     - N+n: Search N+n located line number and then assign that line as the start line (Ex. N3).
     - T+n: Search T+n located line number and assign it as the start line (Ex. T01).

- If line number is out of max line number, then it will assign to the last line.
- Please refer to 3.4.3.4 break point initialization about start block go back.

8. Tool Data
    - T
    - 4 numbers
    - The first two code are the tool no..
    - The last two code are the tool compensate no.

1.6.1.4 Display Area of Machining Setting

- Description

✧ This area is overlap with 「Machining Information」, press F4「Parameter Set」 to change the displays.

- Explanation of Display:

1. Interrupt Line No.

✧ Display the last interrupted serial number (N)

2. Interrupt Colum No.

✧ Display the last interrupted line number (L)

3. Spindle speed

✧ Speed of spindle.

✧ It is allow to setting when system is busy. Moreover, it will be enabled immediately

- Feed rate

✧ Set the speed of the feed rate. ◦

✧ It is allow to setting when system is busy, but the value will be updated after completely executing processing block.

- Total AcumPar

✧ Total work pieces number machined by CNC

System cannot automatically reset this value to zero

- Part count

✧ Setting current work pieces no.

- ✧ Count no. will begin from zero when the running program is change.
- ✧ When CNC executes M code defined by parameter 3804, part count would be added 1 and run time will be reset to 0. When required part number is reached, system will change to halt status.

- Required part
  - ✧ Set the upper limit of part count number.
  - ✧ Once part count number is reached, an alarm will be pop up and system will change to halt status.

1.6.1.5 Simulation Area

- Description
  - ✧ Display the tool trajectory of current program.
  - ✧ Related setting, please see F2-program→ F5-simulaiton→ F5- Simu. Setting.
  - ✧ Use F2「Simulation Switch」 to change the display content

1.6.2 Open File to Edit

- Command:
  - ✧ F4 Monitor→F1 Open File to Edit
- Function
  - ✧ Load and edit the current machining program, also switch to “F2-program” interface.
- Note:
  - ✧ Once system is on running state, edit function will be disabled.

1.6.3 Simulation Switch

- Command:
  - ✧ F4 Monitor→F2 Simulation Switch
- Description
  - ✧ Display or hide simulation display
  - ✧ 「Graph Adjust」 will enable under 「Simulation Switch」 conditions.

1.6.4 MDI Input

- Command:
  - ✧ F4 Monitor→F3 MDI Input
- Description
  - ✧ Manual Data Input. Using for simple NC program or testing purpose.
- Operation:
  - ✧ Select MDI mode
  - ✧ MDI function is enabled after finishing HOME search action.

- ✧ Press F3 「MDI」, edit the program.
- ✧ Press F1 (OK) to confirm the input command.
- ✧ The command line will show up on the right upper corner of screen.
- ✧ Press 【CYCLE START】to execute the command.

- ● Note:
  - ✧ This function is enables under MDI mode.

1.6.5 Parameter Set

- ● Command:
  - ✧ F4 Monitor→F4 Parameter Set
- ● Function
  - ✧ Switch the screen between 「Machining Setting」 and 「machining information」.

1.6.6 Tool Wear Set

- ● Command:
  - ✧ F4 Monitor→F5 Tool Wear Set
  - ● Description
    - ✧ Display the tool wear setting interface, user can setting tool wear here.
    - ✧ Actual Tool length= Tool length +Tool wear
  - ● Parameter Setting
    - ✧ Tool Wear Set: Tiny tool length modification.
  - ● Note
    - ✧ If the tool length is setting by measure method tool wear setting will become 0 after the tool length is set.

1.6.7 Start MPG Coordinate

1. Command:
    - F4 Monitor→ F6 Start MPG Coordinate
2. Description
    - Detail descriptions please refer to 3.4.3.3.

1.6.8 Work Record

- Command:
    - ✧ F4 Monitor→ F7 Work Record
- Description
    - ✧ Check current machining record and export to external storage device.

1.6.9 Clear Acum Cycle Time

- Command:
    - ✧ F4 Monitor→ F8 Clear Acum Cycle Time
- Function
    - ✧ Clear the accumulative time

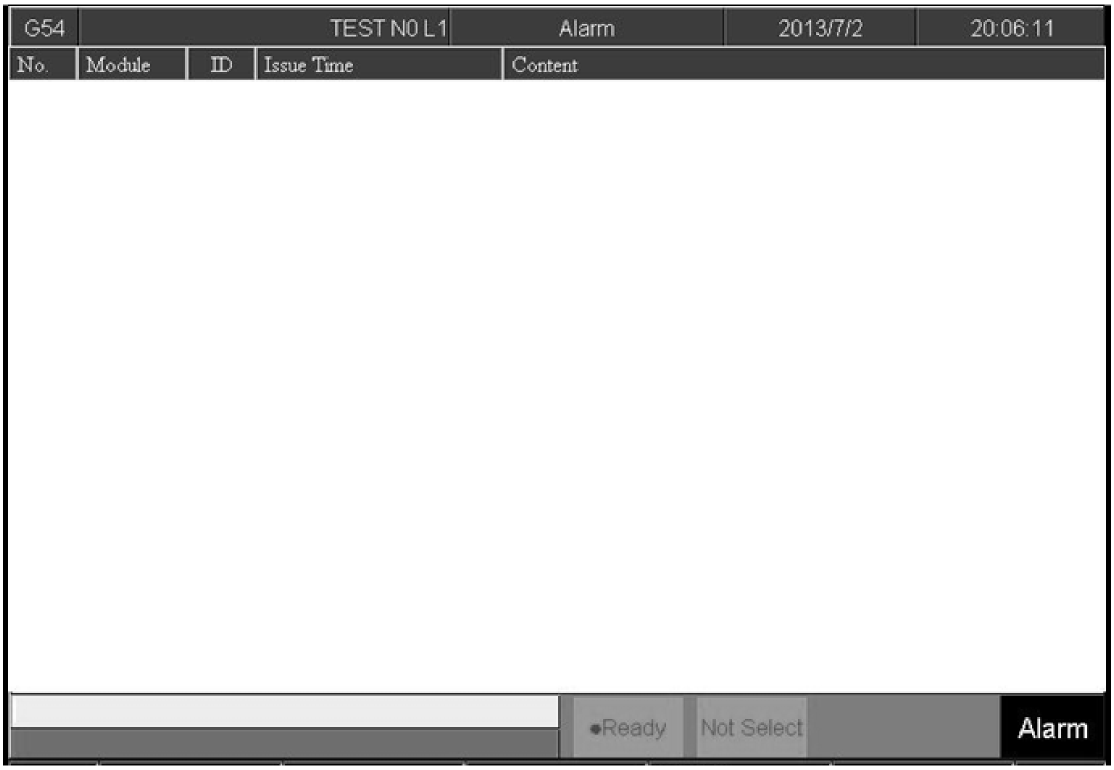
1.6.10 Graph Adjust

- Command:
    - ✧ F4 Monitor→ Next→F1 Graph Adjust
- Description
    - ✧ Zoom in/out simulation graph, this function will enable under 「Simulation Switch」 is open.
- Operation
    - ✧ Please refer to 1.4.7 「Simulation」.

1.7 Maintain

- Command:
  - ✧ F8 Maintain
- Description
  - ✧ Screen displays alarm, network setting, fast diagnostic, PLC param setting, system setting

1.7.1 Alarm


G54
TEST N0 L1
Alarm
2013/7/2
20:06:11
No.
Module
ID
Issue Time
Content
•Ready
Not Select
Alarm

- Command:
  - ✧ F8 Maintain→F1 Alarm
- Description
  - ✧ Display alarm messages on the screen.

1.7.1.1 Pending Alarm

- Command:
  - ✧ F8 Maintain→F1 Alarm→ F1 Pending Alarm
- Description
  - ✧ Display current system alarm.

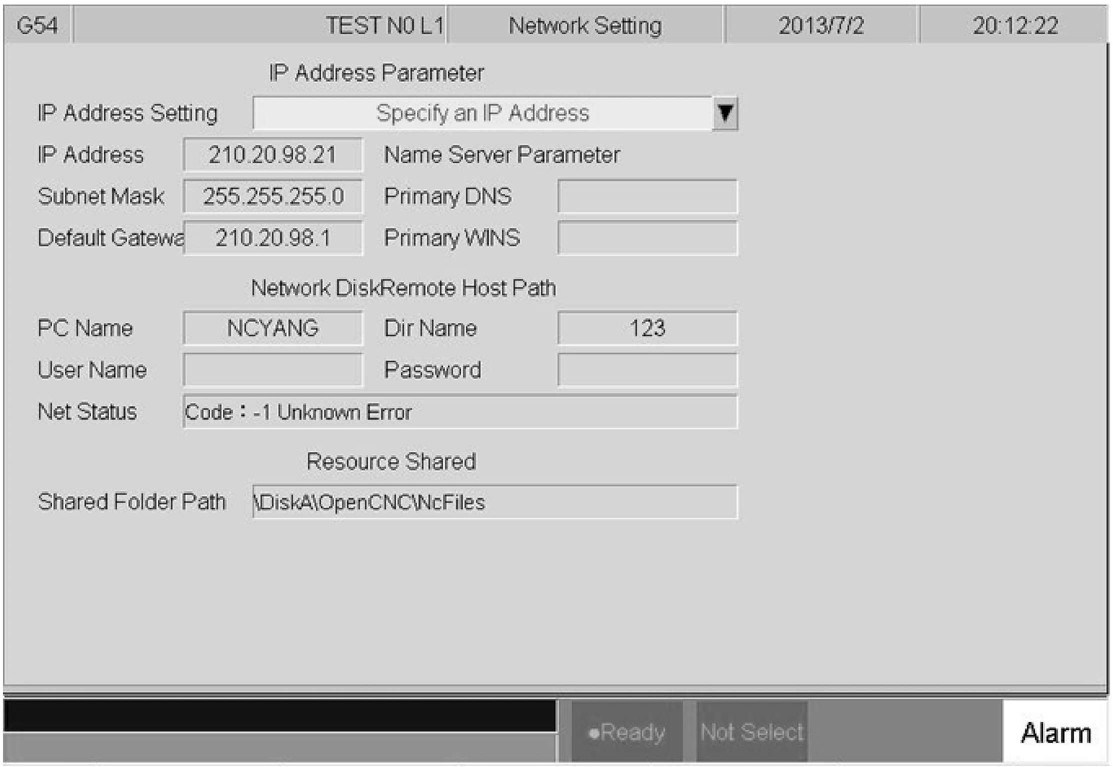
1.7.1.2 History Alarm

- Command:
  - ✧ F8-Maintain→F1 Alarm→ F2 History Alarm
- Description
  - ✧ Show all the alarm history of the system.
- Note:
  - ✧ Some alarm were not displayed here, ex: MACRO alarm

1.7.1.3 Save Alarm

- Command:
  - ✧ F8 Maintain→F1 Alarm→ F3 Save Alarm
- Description
  - ✧ Save Alarm History to external device according to the current display content.
  - ✧ Export file name are default :
  - ✧ Actual alarm: actalm.txt.
  - ✧ History: histalm.txt.

1.7.2 Network Setting



IP Address Parameter

IP Address Setting: Specify an IP Address

IP Address: 210.20.98.21

Subnet Mask: 255.255.255.0

Default Gatewa: 210.20.98.1

Name Server Parameter

Primary DNS:

Primary WINS:

Network DiskRemote Host Path

PC Name: NCYANG

Dir Name: 123

User Name:

Password:

Net Status: Code：-1 Unknown Error

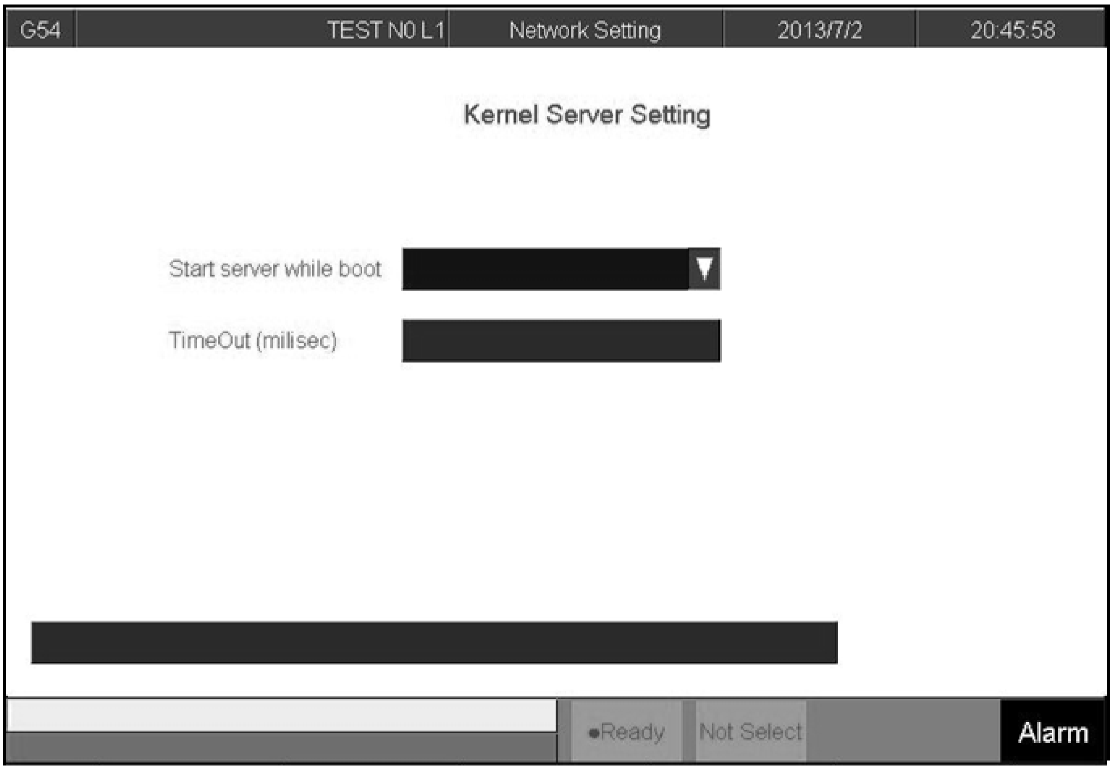
Resource Shared

Shared Folder Path: \DiskA\OpenCNC\NcFiles

- Command:
  - ✧ F8 Maintain→F2 Network Setting
- Description
  - ✧ System network setting
- Related information
  - ✧ IP address setting
  - ✧ Select 「Obtain an IP address automatically」 when network cable(with HUB) is used.
  - ✧ For jumper (without HUB), select “use the following IP address” and enter IP address(the last IP no. must different from controller setting) and Subnet mask(same with controller setting)
- IP Address
  - ✧ Enter IP address that can be used.

- Sunet Mask
  - Enter the IP address for subnet mask (the same with PC setting).
- PC name
  - Enter the same full computer name of PC.
- Dir name
  - Enter the sharing folder name (same with PC sharing folder)
  - User name and password
  - If the shared folder is not setting the user and password name, user do not need to enter user name, if yes, please enter the same user name and password.

1.7.2.1 Set Kernel Server

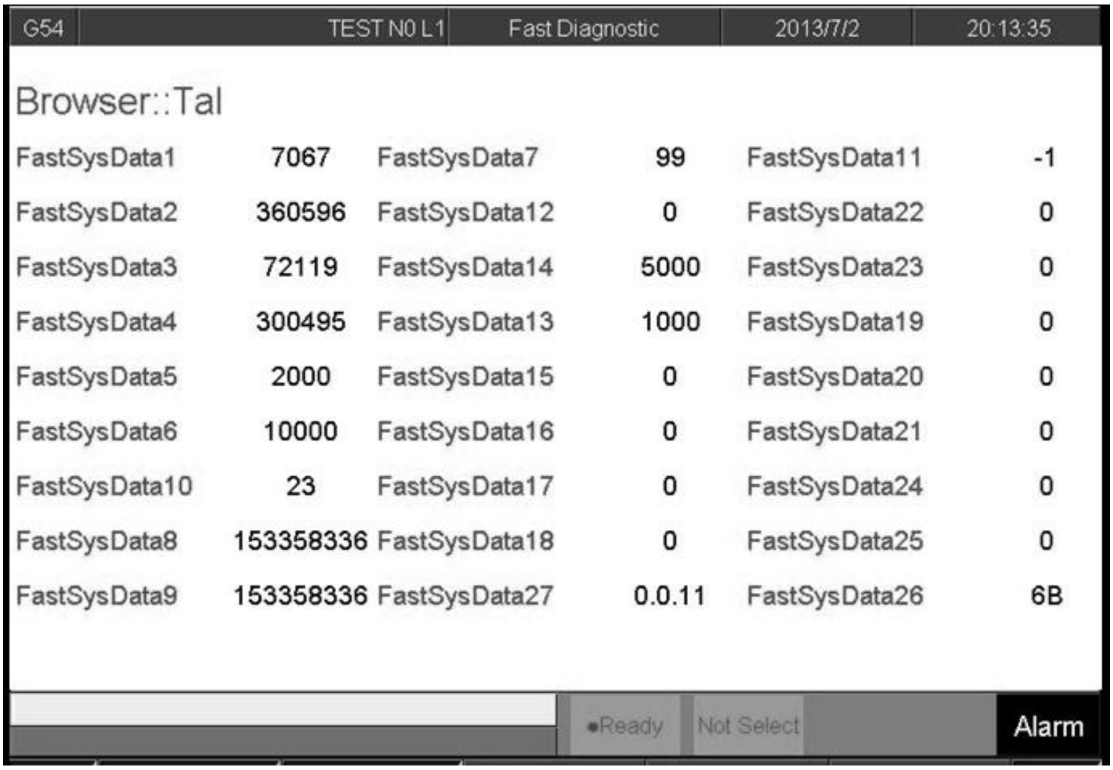

G54
TEST N0 L1
Network Setting
2013/7/2
20:45:58
Kernel Server Setting
Start server while boot
TimeOut (milisec)
•Ready
Not Select
Alarm

- Command:
  - ✧ F8 Maintain→F2 Network Setting→ F5 Set Kernel Server
- Description
  - ✧ Setting related function to kernel server
- Related infor.
  - ✧ Start server and kernel or not when power on.
  - ✧ Timeout(ms)
  - ✧ Set the acceptable time out when connecting to Kernel server unsuccessfully.

1.7.2.1.1 Start Server

- Command:
  - ✧ F8 Maintain→F2 Network setting→ F5 Set Kernel Server→ F1 Start Server
- Description
  - ✧ Start server immediately

1.7.3 Fast Diagnostic

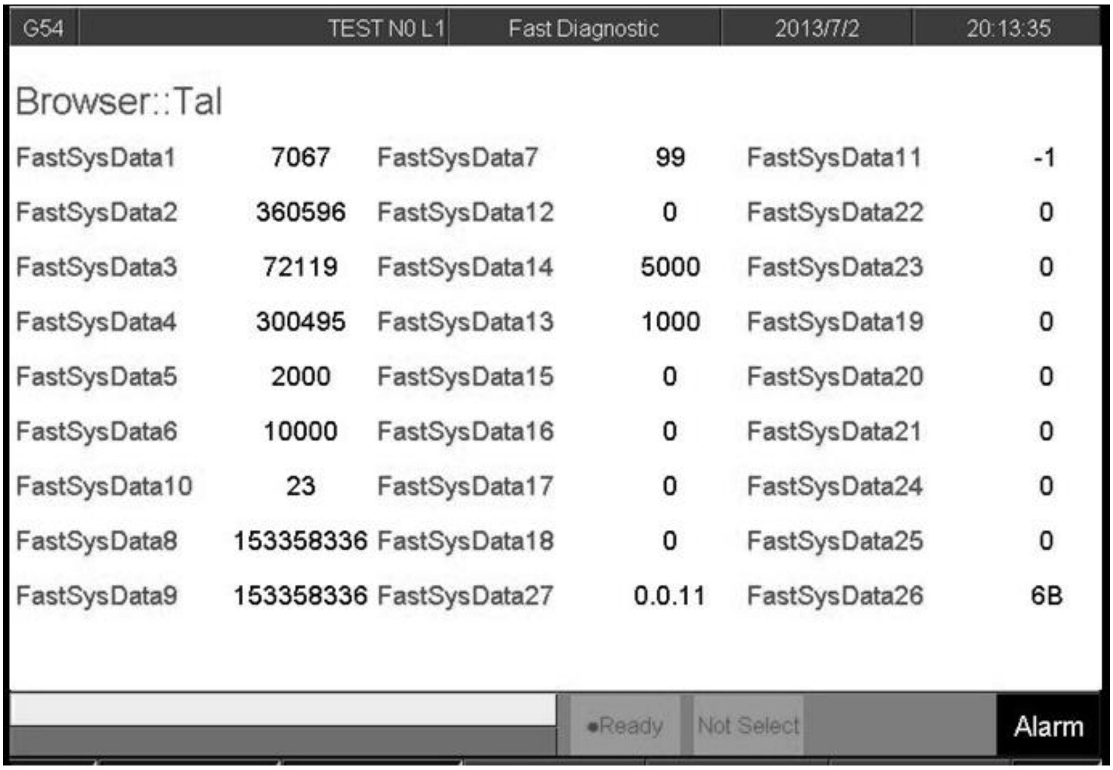


Browser::Tal

FastSysData1	7067	FastSysData7	99	FastSysData11	-1
FastSysData2	360596	FastSysData12	0	FastSysData22	0
FastSysData3	72119	FastSysData14	5000	FastSysData23	0
FastSysData4	300495	FastSysData13	1000	FastSysData19	0
FastSysData5	2000	FastSysData15	0	FastSysData20	0
FastSysData6	10000	FastSysData16	0	FastSysData21	0
FastSysData10	23	FastSysData17	0	FastSysData24	0
FastSysData8	153358336	FastSysData18	0	FastSysData25	0
FastSysData9	153358336	FastSysData27	0.0.11	FastSysData26	6B

- Command:
  - ✧ F8 Maintain→F3 Fast Diagnostic
- Description
  - ✧ Display simple diagnostic information of system and axes

1.7.3.1 System Data

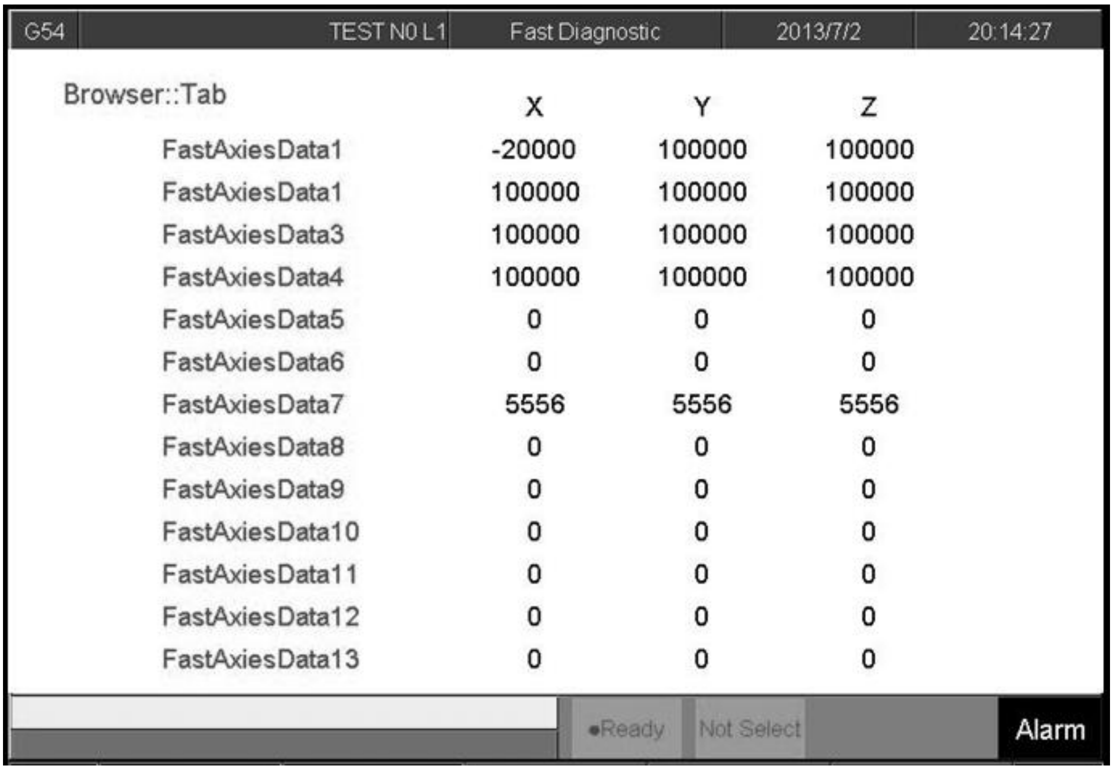


Browser::Tal

FastSysData1	7067	FastSysData7	99	FastSysData11	-1
FastSysData2	360596	FastSysData12	0	FastSysData22	0
FastSysData3	72119	FastSysData14	5000	FastSysData23	0
FastSysData4	300495	FastSysData13	1000	FastSysData19	0
FastSysData5	2000	FastSysData15	0	FastSysData20	0
FastSysData6	10000	FastSysData16	0	FastSysData21	0
FastSysData10	23	FastSysData17	0	FastSysData24	0
FastSysData8	153358336	FastSysData18	0	FastSysData25	0
FastSysData9	153358336	FastSysData27	0.0.11	FastSysData26	6B

- Command:
  - ✧ F8 Maintain→F3 Fast Diagnostic→ F1 System Data
- Description
  - ✧ Display simple diagnostic information of system

1.7.3.2 Axes Data

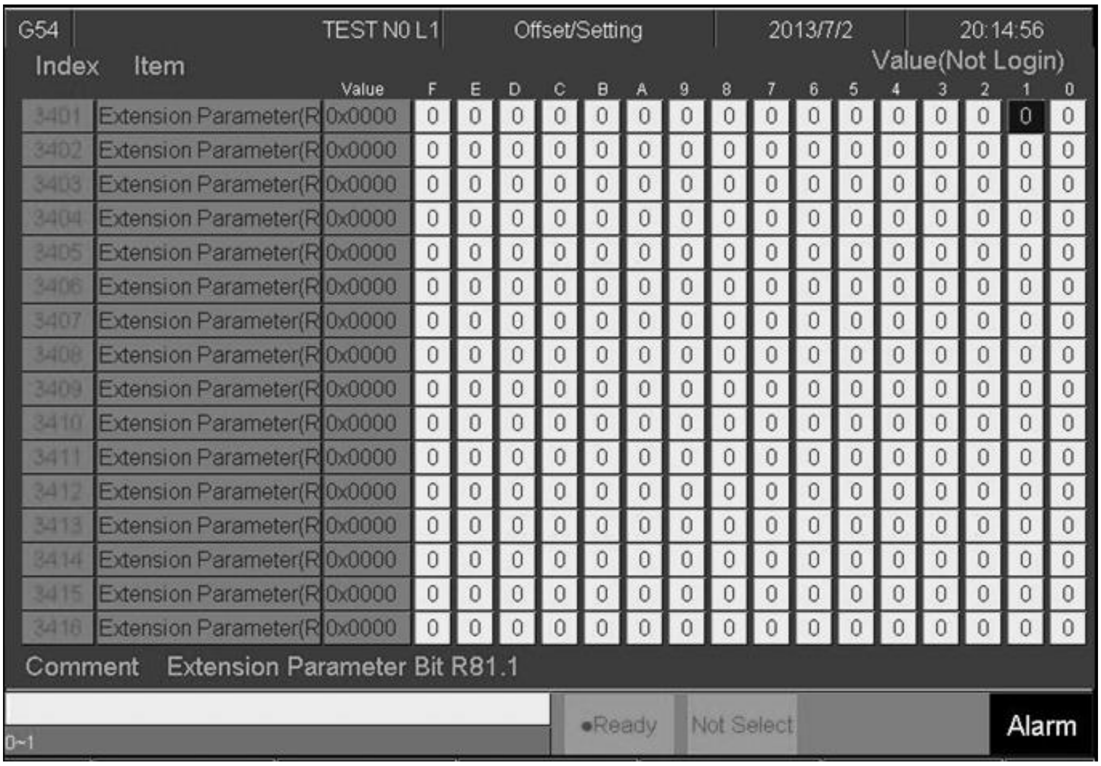

G54
TEST N0 L1
Fast Diagnostic
2013/7/2
20:14:27
•Ready
Not Select
Alarm

Browser::Tab

	X	Y	Z
FastAxiesData1	-20000	100000	100000
FastAxiesData1	100000	100000	100000
FastAxiesData3	100000	100000	100000
FastAxiesData4	100000	100000	100000
FastAxiesData5	0	0	0
FastAxiesData6	0	0	0
FastAxiesData7	5556	5556	5556
FastAxiesData8	0	0	0
FastAxiesData9	0	0	0
FastAxiesData10	0	0	0
FastAxiesData11	0	0	0
FastAxiesData12	0	0	0
FastAxiesData13	0	0	0

- Command:
  - ✧ F8 Maintain→F3 Fast Diagnostic→ F2 Axes Data
- Function
  - ✧ Display simple diagnostic information of Axes

1.7.4 PLC Param Setting



Index	Item	Value	F	E	D	C	B	A	9	8	7	6	5	4	3	2	1	0
3401	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3402	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3403	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3404	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3405	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3406	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3407	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3408	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3409	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3410	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3411	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3412	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3413	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3414	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3415	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0
3416	Extension Parameter(R	0x0000	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0	0

Comment Extension Parameter Bit R81.1

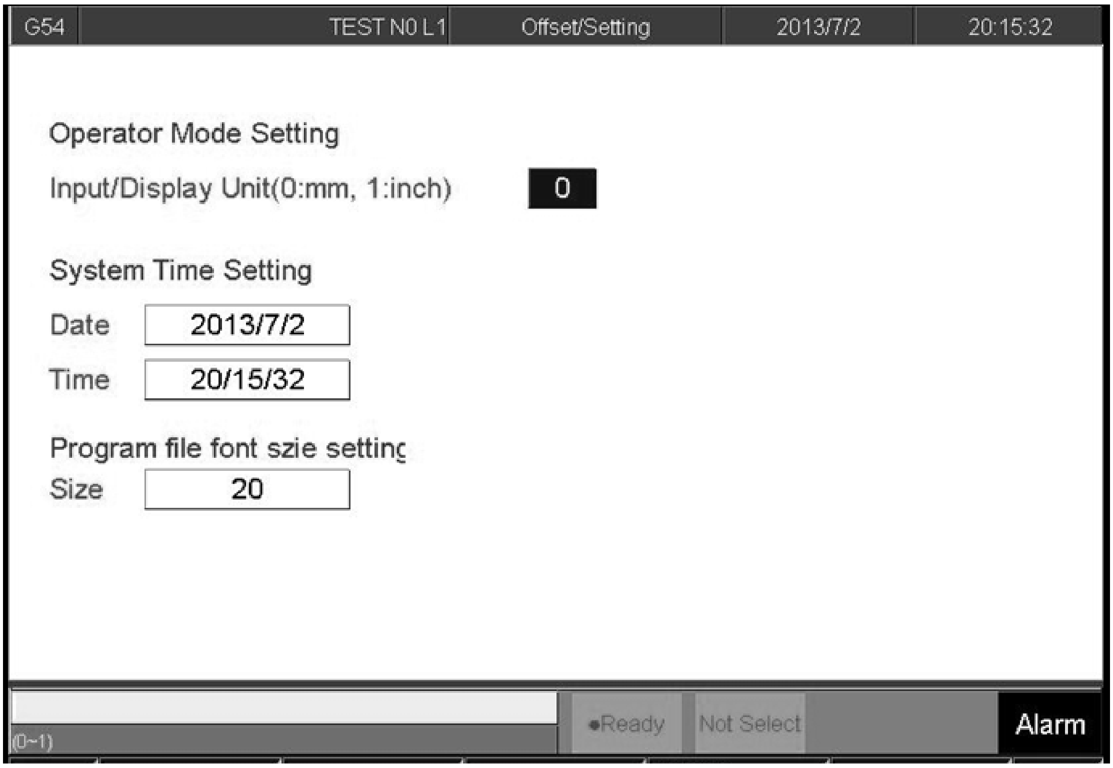
- Command:
  - ✧ F8 Maintain→ F4 PLC Param Setting
- Description
  - ✧ SYNTEC's controller provides R81 ~ R100, totally 20 sets of registers for machine manufacture use, each register have 16 Bits.
  - ✧ Machine manufacture can use those 20 sets of registers provide user to control the flag of PLC specific functions.
- Operation
  - ✧ Press【↑】【↓】【←】【→】 to move cursor and 【PageUp】【PageDown】go to next/previous page
  - ✧ Users are able to input [0] or [1] and also add comment for every bits.

✧ Corresponding file name for comment:

ParamExt_RBit_(L).xml

✧ (L)=COM/CHT/CHS/language

1.7.5 System Setting


G54
TEST N0 L1
Offset/Setting
2013/7/2
20:15:32
(0~1)
•Ready
Not Select
Alarm

Operator Mode Setting

Input/Display Unit(0:mm, 1:inch) 0

System Time Setting

Date 2013/7/2

Time 20/15/32

Program file font szie settinc

Size 20

- Command:

  ✧ F8 Maintain → F5 System Setting

- Description

  ✧ This page is used to set system environment

- Operation

  ✧ Press【↑】【↓】【←】【→】to move cursor and 【PageUp】【PageDown】go to next/previous page

1.7.5.1 Operator Mode Setting

✧ Setting system unit

■0：mm (Metric unit)

■1：Inch (British unit)

✧ Note: reboot to enable setting

1.7.5.2 System Time Setting

✧ Date: input format YYYY/MM/DD

- YYYY: year
- MM: month
- DD: day

✧ Time: input format HH/MM/SS

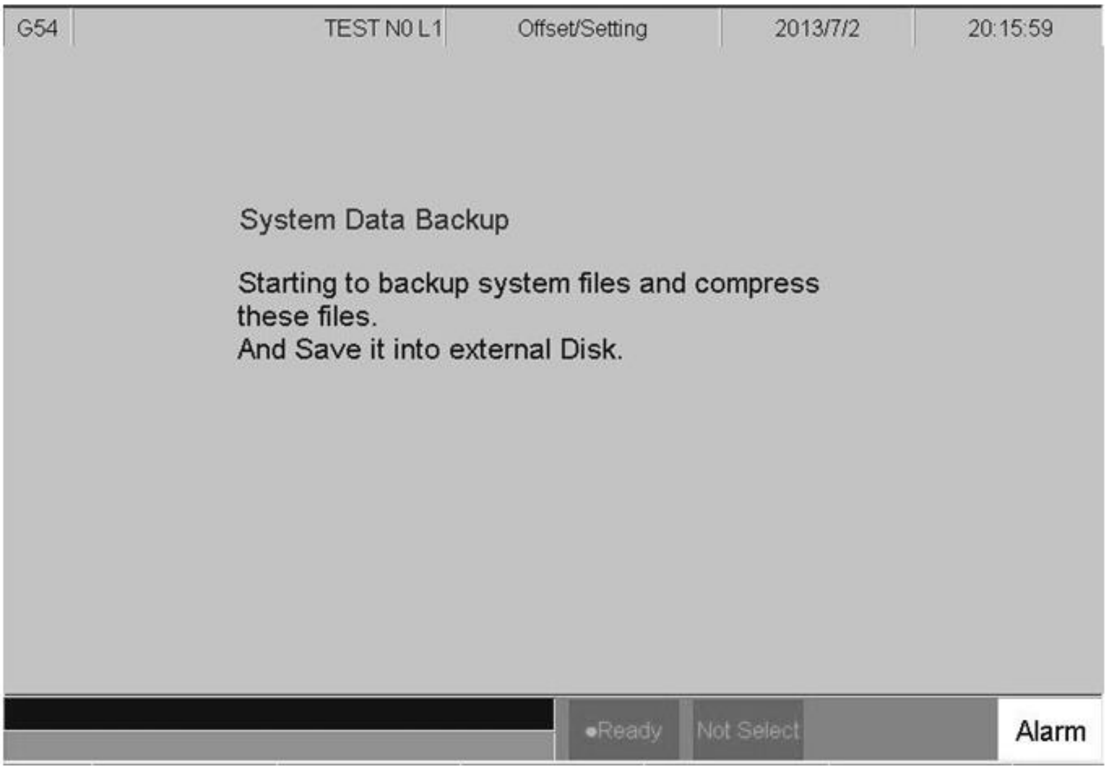
- HH: hours
- MM: minute
- SS: second

1.7.5.3 Program File Font Size Setting

✧ Setting display of font size in program

✧ Note: reboot to enable setting

1.7.6 Backup System


G54
TEST N0 L1
Offset/Setting
2013/7/2
20:15:59
•Ready
Not Select
Alarm

System Data Backup

Starting to backup system files and compress these files.
And Save it into external Disk.

- Command:
  - ✧ F8 Maintain → F6 Backup System
- Description
  - ✧ Compress backup system data and save it into external device.

1.7.7 About

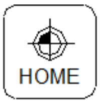
- Command:
  - ✧ F8 Maintain → F8 About
- Function
  - ✧ Provide controller software version

2 Machine Operation Panel

2.1 Operation Panel

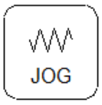
- ● POWER ON
  - ✧ Turn on controller's power
- ● POWER OFF
  - ✧ Turn off controller's power
- ● Emergency Stop
  - ✧ For safety reason of user and machine in case of unusual phenomenon, after this button is pressed, CNC would stop all motion, and all main power. Therefore safety of people and machine will be guaranteed.

■ **Home Mode**



● When CNC power is on, please implement home search

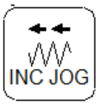
■ **JOG Mode (Rapid JOG)**

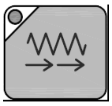


● Operators can use JOG to control axis movement with

「X+/X-/Y+/Y-/Z+/Z-」 on panel.

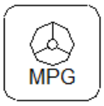
■ **Incremental JOG (IN JOG)**





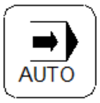
● Operators can use JOG to control axis movement.

■ **MPG Mode**



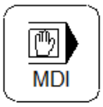
● User can use MPG (Manual Pulse Generator) mode to control axis movement.

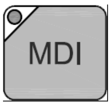
■ **AUTO Mode**



● Users use this function to execute machining NC file

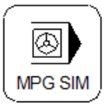
■ **MDI Mode**

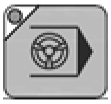




● Users use this function to execute a block without NC file

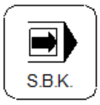
■ **MPG Simulation**





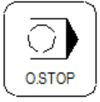
● Enable this function under 「AUTO」 and 「MDI」 Mode, the percentage of moving speed is depend on the MPG turning speed . This function used to check NC file.

■ **Single Block**



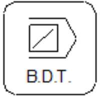
● Enable this function under 「AUTO」 and 「MDI」 Mode, after finished one block action hold on the 「B-STOP」 mode. To continue, users need to do the 「Cycle start」 action again. This function used to check NC file.

■ **Optional Stop**



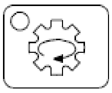
● Users can use this function to decide whether CNC is stop or not when encounter with 「M01」within NC (Numerical Control) program, while executing NC program.

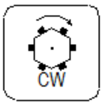
■ **Optional Skip**



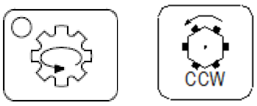
● Users can use this function to decide whether program skips or not when program is encountered with 「/」 sign in NC file.

■ Auto Tool Change in clock wise direction

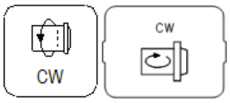




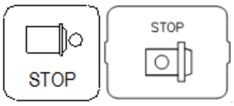
- **Auto Tool Change ATC in counter clock wise direction**



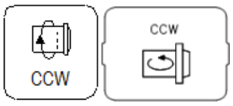
- **Spindle Control**



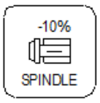
Spindle CW rotation



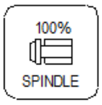
Spindle stop



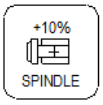
Spindle CCW rotation



Spindle rate deceleration: Spindle speed will decelerate 10%.

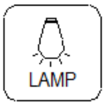

100%
SPINDLE

Spindle rate 100%: Spindle will rotate with 100%.



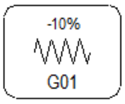
Spindle rate acceleration: Spindle speed will accelerate 10%.

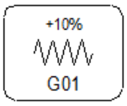
- **Working LED**

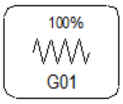


- Turn ON/OFF working LED

- **G01 Rank**

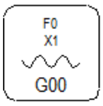


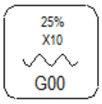


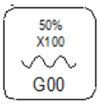


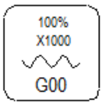
Decreasе G01 rate 10%   Increase G01rate10%   G01 rate100%

- ■ **G00 Rank**





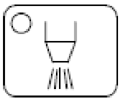



100%
X1000
G00

G00 rate x1 G00 rate x10 G00 rate x100 G00 rate x1000

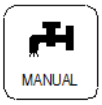
- ● Those function key can be regard as G00, MPG or INJOG rank.

- ■ **Blowing**



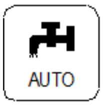
- ● ON/OFF blowing air.

- ■ **Cutting coolant system**



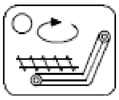
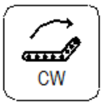
- ● Turn ON/OFF working liquid.

- ■ **Auto cutting coolant system**

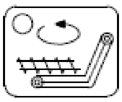
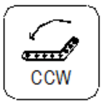


- ● This function will be enable, when encounter with relate coolant on/off M code in NC program.

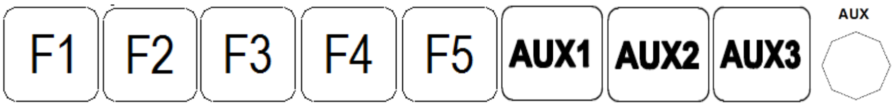
- ■ **Chip conveyor machine move in clockwise direction**

- ■ **Chip conveyor machine move in counter clockwise direction**

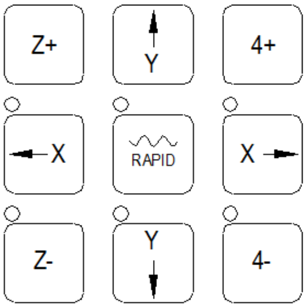



- ■ **Augment key(User define)**


F1
F2
F3
F4
F5
AUX1
AUX2
AUX3
AUX

- Machine builder augment key, combine with PLC ladder.

- **Direction of linear Axes &Rotation Axes & fast travel key**



- Move the axes with fast travel speed key when press the axes and fast travel key at the same time.

2.2 Text Key Description

A ～ Z : 26 English character keys

0 ～ 9 : Numerical keys

DEL : Delete key

INS : Insert /replace key

SHIFT : Switch keyboard to use subscript key on keyboard

SPACE : Space key

BACK SPACE : While working with text, use this key to delete characters to the left of the insertion point.

RESET : ”RESET “ abort the CNC status, only use this key if necessary

ENTER	: To input current data to input box
HELP	: Enable Help function/message on screen
/	: For optional skip key input
;	: End of block
.	: Decimal fraction
PROG	: Change to program edit page.
MONI	: Change to monitor page.
TEACH	: Change to customize teach page. (Disable function key with no customize action)
INSERT	: Insert word when edit program.
DELETE	: Delete word when edit program.
	: PAGE UP
	: PAGE DOWN

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

All above symbols are used for “Program Edit” mode.

↟ ↡ ： Switch cursor to Page Up /Page Down

↑ ← → ↓ ： Edit cursor control key

3 How to Operate 11 Series Controller

3.1 System Status

3.1.1 Not Ready

On Not Ready status, system cannot implement any operations
Conditions:

- ✧ Emergency stop button is pressed
- ✧ Serious alarm appears
- ✧ System is switched to "Not Ready" status and stop all operations to ensure safety of machine and human

SYNTEC's controller will display different status corresponding to different conditions as below:

3.1.2 Ready

On "Ready" status, system can implement all operations
Conditions:

- ✧ "Emergency stop" button is released and no alarm exists, system is switched "Not Ready" to "Ready" status.
- ✧ On "Busy"/ "Pause"/ "B-stop" status, if user presses "Reset" key; System is switched to "Ready" status.

3.1.3 Busy

System is operating program
Conditions:

- ✧ On"Busy"/ "Pause"/ "B-stop" status, if if system executes process; System will switched to "Busy" status.

3.1.4 Pause

In operational process, system pause the axis movement
Conditions:

- ✧ Once Feed hold button is pressed when system is on "Busy" status, System will be switched from "Busy" to "Pause" status

Note: On "Pause" status, Spindle still rotates

3.1.5 B-stop

Conditions:

- ✧ Machine program runs to M0 single block when system is on “Busy” status.
- ✧ Single block function is triggered when system is on “Busy” status.

Note: On “B-Stop” status, Spindle still rotates

3.2 Machine Preparation

3.2.1 Manual Function

SYNTEC controller provides 4 functions used to control axis movement including JOG, INC JOG, MPG and Rapid JOG

3.2.1.1 JOG

- Function:
  - ✧ Control the axis movement X, Y, Z according to selected direction
  - ✧ Can control more than one axis at the same time
- Condition:
  - ✧ System is on “Ready” status
  - ✧ JOG mode is selected
- Operation:
  - ✧ Press axis direction key (X+,X-,Y+,Y-,Z+…)
  - ✧ Hold the axis direction key to keep the axis moving uninterrupted
  - ✧ JOG speed can be adjusted by G01%

3.2.1.2 Rapid JOG

- Function:

  - ✧ Control axis movement X, Y, Z according to selected direction with G00 speed
  - ✧ Can control more than one axis at the same time
- Condition:
  - ✧ System is on “Ready” status
  - ✧ JOG mode is selected
- Operation:
  - ✧ Press axis direction key (X+,X-,Y+,Y-,Z+…) and rapid key “～ “at the same time ,machine will move with G00- rapid speed
  - ✧ Hold the axis direction key to keep the axis moving uninterrupted
  - ✧ Rapid JOG speed can be adjusted by G00%
- Note:
  - ✧ Rapid JOG is usually much faster than JOG, so when operating, please confirm the axis position to ensure human and machine safety.

1. INC JOG (incremental JOG)

- Function:
  - ✧ Control axis movement X, Y, Z according to selected direction with fixed distance(incremental distance)
- Condition:
  - ✧ System is on “Ready” status
  - ✧ INC JOG mode is selected
- Operation:
  - ✧ Press axis direction key (X+, X-,Y+,Y-,Z+…)
  - ✧ Cannot be constantly triggered like JOG or rapid JOG mode, INC JOG mode only moves once with a fixed distance when axis direction key is pressed once.
  - ✧ The fixed distance in INC JOG mode can be selected by percentage movements as below:
  - ✧ X1 ：Distance 0.001mm
  - ✧ X10 ：Distance 0.010mm
  - ✧ X100：Distance 0.100mm
  - ✧ Those percentage movements is shared with MPG mode

2. MPG

- Function:
  - ✧ Control axis movement X, Y, Z according to selected direction
- Condition:
  - ✧ System is on “Ready” status
  - ✧ MPG mode is selected
- Operation:
  - ✧ Press axis direction key (X+, X-,Y+,Y-,Z+…)
  - ✧ Cannot be constantly triggered like JOG or rapid JOG mode, INC JOG mode only moves once with a fixed distance when axis direction key is pressed once.
  - ✧ The moving distance when turning one track in MPG mode can be selected by percentage movements as below:
  - ✧ X1 ：Distance 0.001mm
  - ✧ X10 ：Distance 0.010mm
  - ✧ X100：Distance 0.100mm

3.2.2 Machining Process

3.2.2.1 AUTO

- Condition:
  - ✧ System is on “Ready” status
  - ✧ AUTO mode is selected
- Operation:
  - ✧ Press 「CYCLE START」 button

- ✧ System will machine the current machining program
- ✧ System status will be switched from “Ready” to “busy” and backs to “Ready” when machining is finished

3.2.2.2 Single Block

- Function:
    - ✧ Excute each single block in program
- Condition:
    - ✧ System is on “Ready” status
    - ✧ Single block mode is selected
- Operation:
    - ✧ Press 「CYCLE START」 button
    - ✧ System will excute process the current single block in program
    - ✧ System status will be switched from “Ready” to “busy” and backs to “Ready” when machining is finished

3.2.2.3 Home

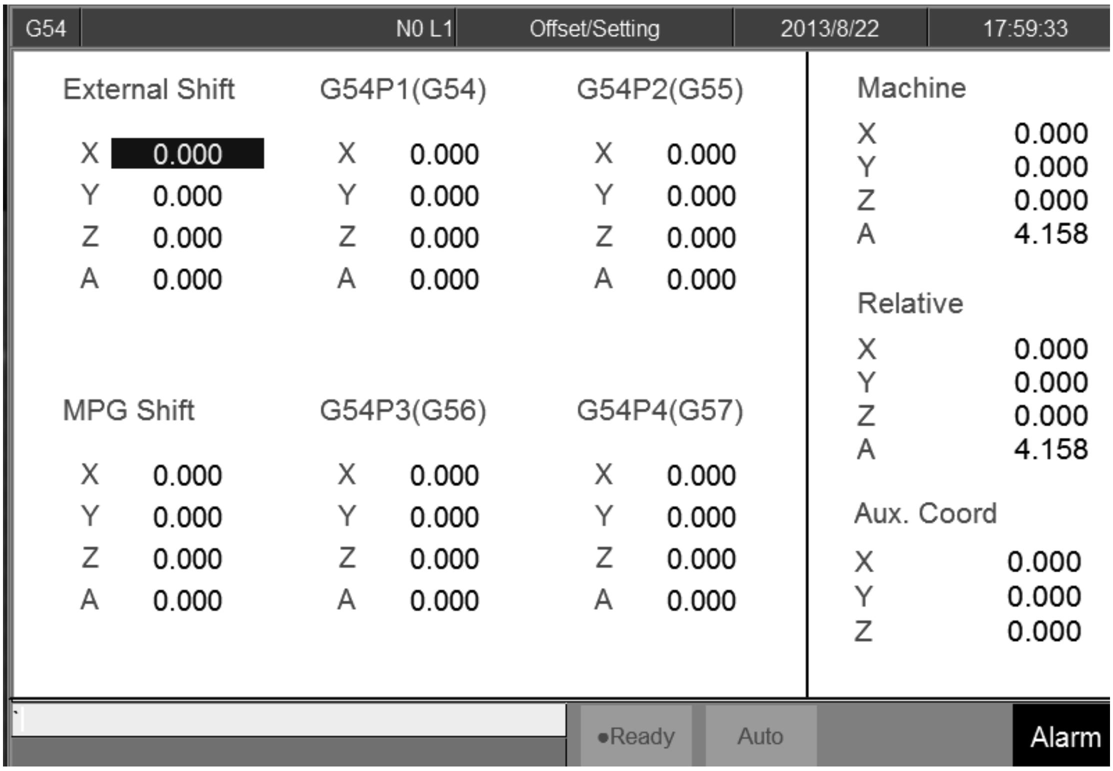
Because tool and workpiece coordinate setting is based on Machine zero point, it is necessary to make sure where is machine zero (HOME). Therefore, when CNC restarts, return to reference point (search HOME) is very important. Otherwise, SYNTEC CNC controller will not be allowed to execute AUTO NC files.

- Operation:
    - ✧ Release emergency stop button, CNC status will change “NOT READY” to “ READY ”
    - ✧ Select HOME mode

- ✧ Press axis direction key(X+,X-,Y+,Y-,Z+…) ,each axis would start HOMING
- ✧ Home direction is defaulted in the CNC parameter
- ✧ Home function can run 3 axes at the same time
- ✧ After HOMING, all machine coordinates will be zero.
- ✧ After HOMING, software stroke limit of each axis just is enable, so before HOMING, please do not run machine too fast

3.3 Workpiece Preparation

3.3.1 Workpiece Cord set



G54 | N0 L1 | Offset/Setting | 2013/8/22 | 17:59:33

	External Shift		G54P1(G54)		G54P2(G55)
X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000
A	0.000	A	0.000	A	0.000

	MPG Shift		G54P3(G56)		G54P4(G57)
X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000
A	0.000	A	0.000	A	0.000

Machine	
X	0.000
Y	0.000
Z	0.000
A	4.158

Relative	
X	0.000
Y	0.000
Z	0.000
A	4.158

Aux. Coord	
X	0.000
Y	0.000
Z	0.000

- Command:
  - ✧ F3 Offset/Setting → F1 Workpiece Cord Set
  - ✧ Switch the cursor to the position wants to set by using page up/down keys and arrow keys
  - ✧ Use ↑↓←→and ↑、↓ move the cursor to the coord you want to key in。
  - ✧ Can use the Mach. Coord. Teach to insert the value.
  - ✧ Can use the Rel. Coord. Teach Cord to insert the value.
  - ✧ Can use the Aux. Coord. Teach to insert the value

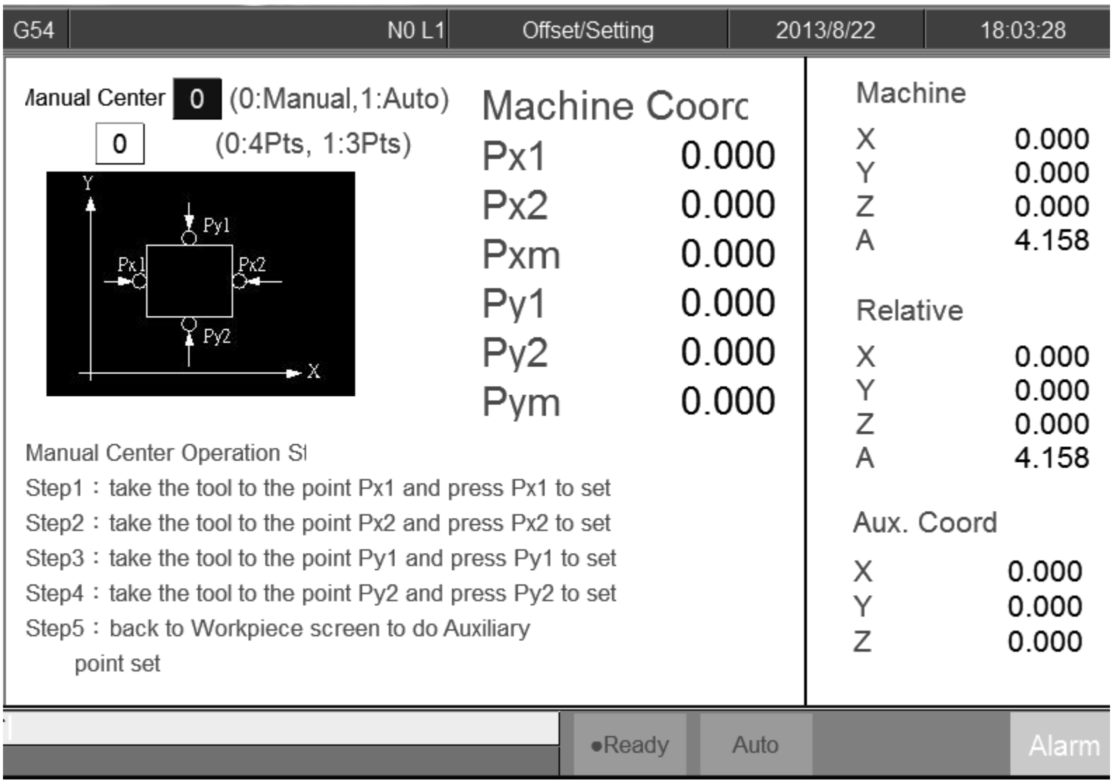
3.3.2 Middle Func.

- Function:

Middle function is used for finding middle point of the workpiece. Use the middle point as the start point. As a result, we need to touch the edge of the workpiece by too or detector. After controller gets the coordinate, it will calculate the middle point itself. We only need to set this point as the machining start point. Please confirm that your maching has automatic alignment tool and then you can use this function.

- Operation:

In the 8 botton system, user can use this function by press F3 Offset/Setting →F1 Workpeice Cord. →F6 Middle Func.



/anual Center 0 (0:Manual,1:Auto)
0 (0:4Pts, 1:3Pts)

Machine Coorc

Px1	0.000
Px2	0.000
Pxm	0.000
Py1	0.000
Py2	0.000
Pym	0.000

Machine

X	0.000
Y	0.000
Z	0.000
A	4.158

Relative

X	0.000
Y	0.000
Z	0.000
A	4.158

Aux. Coord

X	0.000
Y	0.000
Z	0.000

Manual Center Operation St
Step1：take the tool to the point Px1 and press Px1 to set
Step2：take the tool to the point Px2 and press Px2 to set
Step3：take the tool to the point Py1 and press Py1 to set
Step4：take the tool to the point Py2 and press Py2 to set
Step5：back to Workpiece screen to do Auxiliary point set

.

3.3.2.1 Manual 4Pts Center

- Function

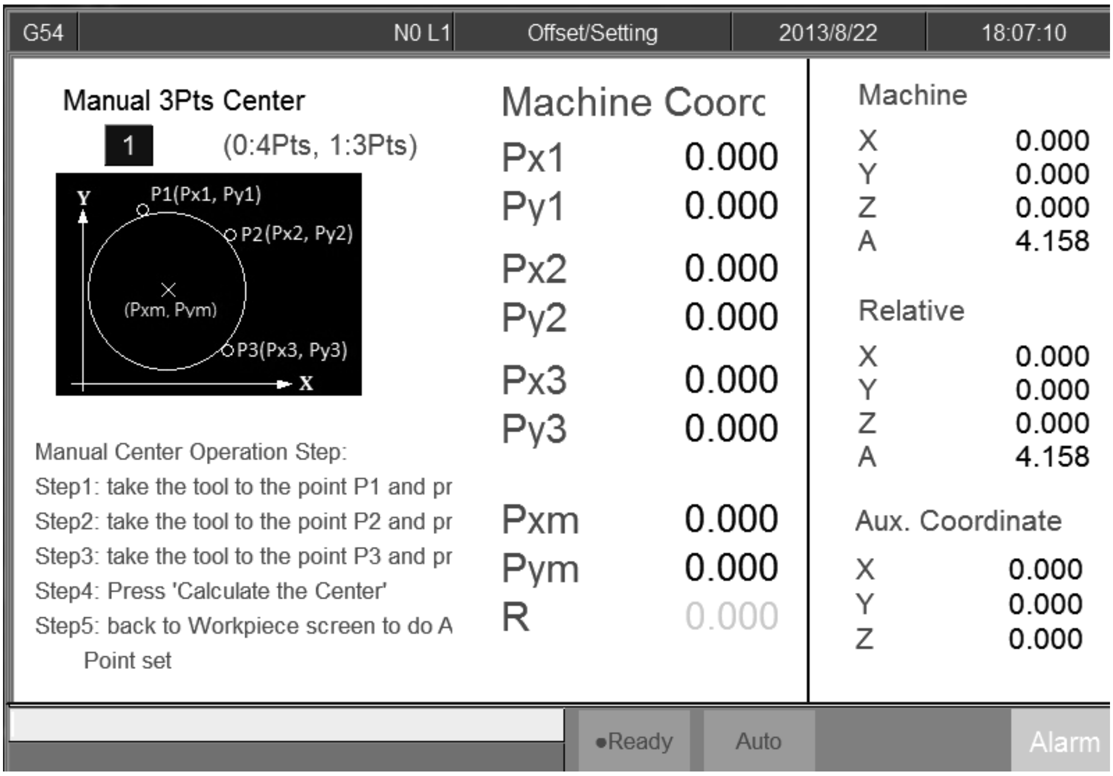
✧ Set the Middle Func. as 0

✧ User controls the machine by MPG, and then moves the tip of 3D machine to the X&Y end point of the workpiece. This system will calculate the center point of the workpiece automatically.

- Operation method
  - ✧ Move the maching by MPG, touching Px1 point in this figure and then press PX1 Set. The system will record the current machnical coord. to Px1. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm and Aux. X position.
  - ✧ Move the maching by MPG, touching Px2 point in this figure and then press PX2 Set. The system will record the current machnical coord. to Px2. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm and Aux. X position.
  - ✧ Move the maching by MPG, touching Py1 point in this figure and then press PY1 Set. The system will record the current machnical coord. to Py1. It will also compute the middle point of Py1 and Py2 and puts the result on Pym and Aux. Y position.
  - ✧ Move the maching by MPG, touching Py2 point in this figure and then press PY2 Set. The system will record the current machnical coord. to Py2. It will also compute the middle point of Py1 and Py2 and puts the result on Pym and Aux. Y position.
  - ✧ Now, the values of Pxm and Pym are the middle point of the workpiece.

- ✧ In the WorklPiece Cord. Screen, move the cursor to the coord you want to set, press F3 insert the Aux. Coord value then this system will set the value accordind to the Aux Coord.

3.3.2.2 Manual 3Pts Center



Manual 3Pts Center

1 (0:4Pts, 1:3Pts)

Manual Center Operation Step:

Step1: take the tool to the point P1 and pr

Step2: take the tool to the point P2 and pr

Step3: take the tool to the point P3 and pr

Step4: Press 'Calculate the Center'

Step5: back to Workpiece screen to do A Point set

Machine Coorc

Px1	0.000
Py1	0.000
Px2	0.000
Py2	0.000
Px3	0.000
Py3	0.000
Pxm	0.000
Pym	0.000
R	0.000

Machine

X	0.000
Y	0.000
Z	0.000
A	4.158

Relative

X	0.000
Y	0.000
Z	0.000
A	4.158

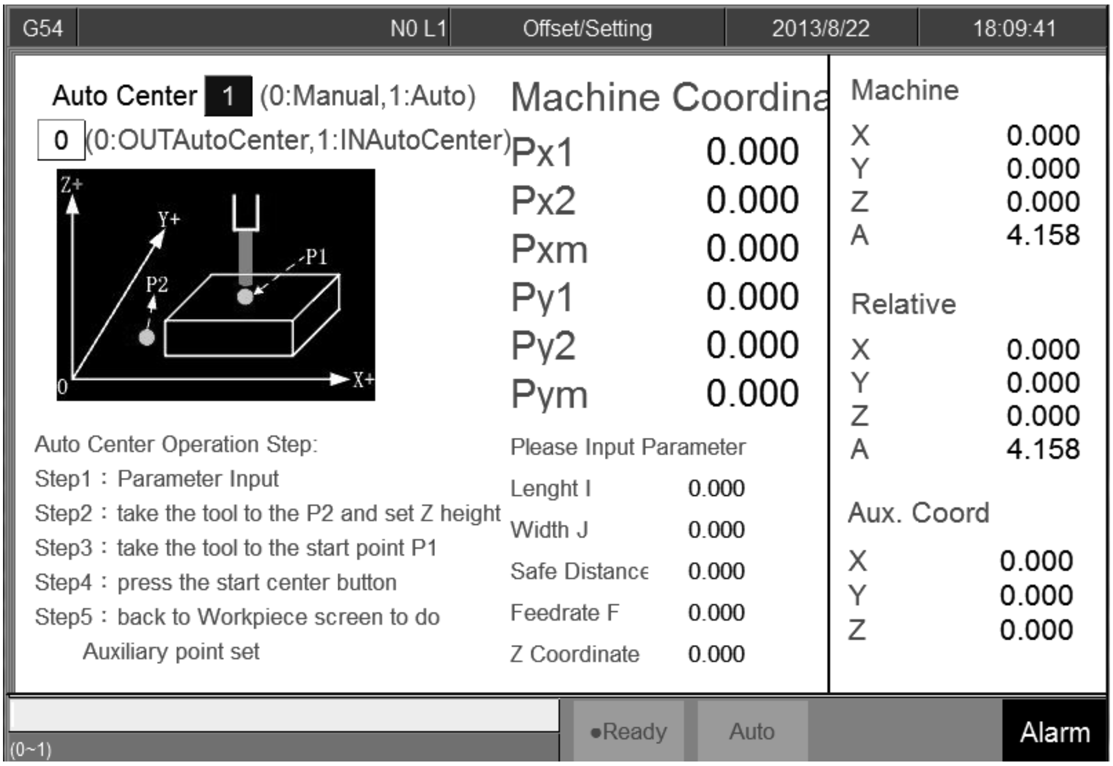
Aux. Coordinate

X	0.000
Y	0.000
Z	0.000

- Operation:
  - ✧ Set the Middle func. as 0, the left upper side will show Manual Center
  - ✧ Set the second line, the middle method, as 1. You can see the screen as this figure.
  - ✧ Move the tool or detector to the round workpiece edge.
  - ✧ Press P1 set, system will record the current X and Y coordinates to Px1 and Py1.
  - ✧ Move to another edge point, press P2 set. System will record the current X and Y coordinates to Px2 and Py2.

- ✧ Move to another edge point, press P3 set. System will record the current X and Y coordinates to Px3 and Py3.
- ✧ After setting these, press Calculate the Center. System will calculate the center point Pxm, Pym and R.
- ✧ The Aux. Coordinate will become Pxm and Pym.
- ✧ If you set the same point or this three point is inline, it will have a warning window pop out.
- ✧ Now you can press ◄ go back to previous page to reset the coordinate.
- ✧ In the WorklPiece Cord. Screen, move the cursor to the coord you want to set, press F3 insert the Aux. Coord value then this system will set the value accordind to the Aux Coord.

3.3.2.3 OUTAutoCenter



G54 | N0 L1 | Offset/Setting | 2013/8/22 | 18:09:41

Auto Center 1 (0:Manual,1:Auto)

0 (0:OUTAutoCenter,1:INAutoCenter)

Auto Center Operation Step:

Step1：Parameter Input

Step2：take the tool to the P2 and set Z height

Step3：take the tool to the start point P1

Step4：press the start center button

Step5：back to Workpiece screen to do Auxiliary point set

Machine Coordina

Px1	0.000
Px2	0.000
Pxm	0.000
Py1	0.000
Py2	0.000
Pym	0.000

Please Input Parameter

Lenght I	0.000
Width J	0.000
Safe Distance	0.000
Feedrate F	0.000
Z Coordinate	0.000

Machine

X	0.000
Y	0.000
Z	0.000
A	4.158

Relative

X	0.000
Y	0.000
Z	0.000
A	4.158

Aux. Coord

X	0.000
Y	0.000
Z	0.000

(0~1)

- Operation:
  - ✧ Set the Middle func. as 1, the left upper side will show Manual Center.
  - ✧ Set the second line, the middle method, as 0. You can see the screen as this figure.
  - ✧ Key in Length I and Width J for workpiece real length and width.
  - ✧ Set Z coordinate, this value is for the the automatical measurement that tool can touch the workpiece edge plane
  - ✧ Set Safe Distance H, this distance is that the tool will not hit with workpiece.
  - ✧ Set Feedrate F, this Feedrate is the measuring tool velocity during auto center process.

✧ Press Auto center start, system will move the tool according to the setting data to contact with workpiece and show the coordinate on the screen. It will also calculate the X Y center point of the workpiece

✧ The center point will be saved at Pxm and Pym.

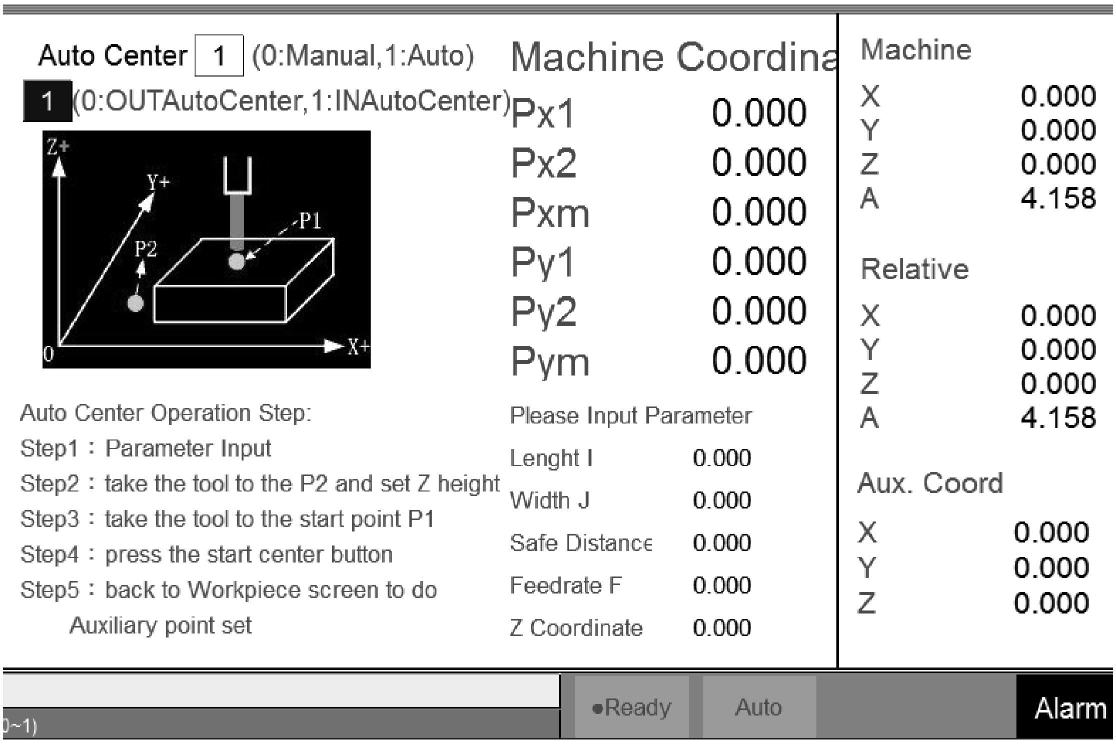
✧ After setting these, press Calculate the Center. System will calculate the center point Pxm, Pym and R.

✧ The Aux. Coordinate will become Pxm and Pym.

✧ Now you can press ◄ go back to previous page to set the coordinate.

✧ In the WorkPiece Cord. Screen, move the cursor to the coord you want to set, press F3 insert the Aux. Coord value then this system will set the value according to the Aux Coord.

3.3.2.4 INAutoCenter



Auto Center 1 (0:Manual,1:Auto)

1 (0:OUTAutoCenter,1:INAutoCenter)

Machine Coordina

Px1	0.000
Px2	0.000
Pxm	0.000
Py1	0.000
Py2	0.000
Pym	0.000

Auto Center Operation Step:

Step1：Parameter Input

Step2：take the tool to the P2 and set Z height

Step3：take the tool to the start point P1

Step4：press the start center button

Step5：back to Workpiece screen to do Auxiliary point set

Please Input Parameter

Lenght I	0.000
Width J	0.000
Safe Distance	0.000
Feedrate F	0.000
Z Coordinate	0.000

Machine

X	0.000
Y	0.000
Z	0.000
A	4.158

Relative

X	0.000
Y	0.000
Z	0.000
A	4.158

Aux. Coord

X	0.000
Y	0.000
Z	0.000

- Operation:
  - ✧ Set the Middle func. as 1, the left upper side will show Manual Center.
  - ✧ Set the second line, the middle method, as 1. You can see the screen as this figure.
  - ✧ Key in Length I and Width J for workpiece real length and width.
  - ✧ Set Z coordinate, this value is for the auto measurement that tool can touch the workpiece edge plane
  - ✧ Set Safe Distance H, this distance is that the tool will not hit with workpiece.
  - ✧ Set feed rate F, this feed rate is the measuring tool velocity during auto center process.
  - ✧ Press Auto center start

3.3.3 Tool Prepare

3.3.3.1 Tool Set

- Purpose
  - ✧ We can set the length and the diameter geometry and wear
  - ✧ Operation condition
  - ✧ Can be used at Manual or Auto mode
  - ✧ Wear value increment upper limit is 1.0, warning will appear if exceed 1.0.
- Operation method
  - ✧ Use direction key 【↑】【↓】【←】【→】move the cursor.
  - ✧ Use【PageUp】【PageDown】switch the page.
  - ✧ Key in method: Absolute, Increment, Measure method.
  - ✧ Absolute: for radius and length compensate.
  - ✧ Increment: for radius wear and length wear.
  - ✧ Radius compensate + radius wear = real G41/G42 compensate.
  - ✧ Length compensate +length wear = real G43/G44 compensate.

3.3.3.2 Manual Measurement of Tool Length

- Purpose
  - ✧ Describe how to use manual measurement to set tool length and work piece coordinate in order to set the machining tool.

- Function
    - ✧ We usually set the tool length and offset in the External Shift if we only use one tool in this machine.
    - ✧ If we have more than one tool, we need to set every tool length we can use relative length or absolute length.
- Operation condition
    - ✧ Manual mode.
- Manual Operation method

  -Relative method.

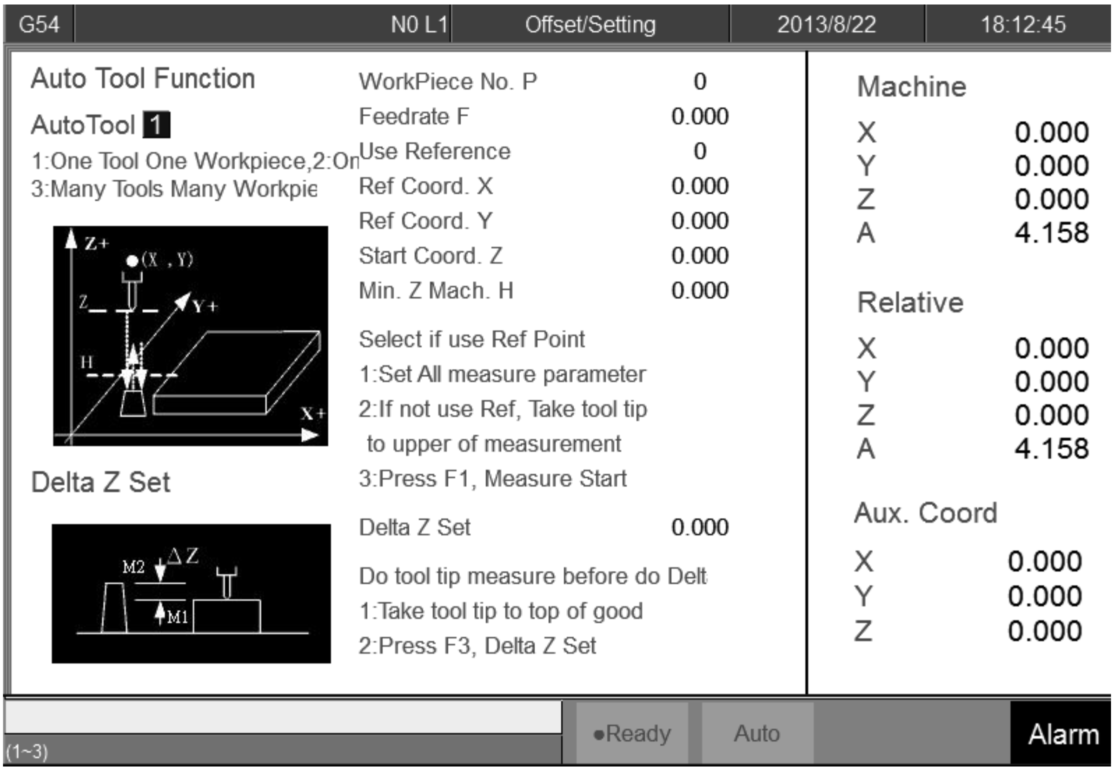
    - ✧ Use manual function moving the reference tool to the Z-coordinate code 0 position of the workpiece (Ex. surface), set the value at G54.
    - ✧ Clean all the relative coordinate value.
    - ✧ Install the measured tool, move the tool to the same position as before (reference surface).
    - ✧ The tool length is the different between measured tool and reference tool.
    - ✧ Key in this number to the corresponding tool No.
    - ✧ If tool break during machining and replace a new tool, we need to change to a new tool. If the Z-cord 0 position already disappear, we can use the standard tool to touch anywhere of the machine and then use new tool to touch the same position. The relative cord. is tool length.

  -Absolute method

- ✧ Use manual function moving the measuring tool to a reference point (Ex. Work piece table).
- ✧ Measure every tool repeatedly.
- ✧ Move the machining tool to the reference position, clean all the relative coordinate.
- ✧ Move the tool to the Z-coord code zero position, set it to G54.
- ✧ If tool break during machining so we need change a new tool, we must touch the reference position first, and then the Z coordinate value is tool length.

3.3.3.3 Tool Tip Measure

- Function
  - ✧ Measuring different tool No. tip position. Because the trigger position of tool alignment equipment is fixed, user can use different tool tip position key in the workpiece coordinate system for the reference of tool length offset. You must check this machine has tool alignment equipment.
- Operation Method
  - ✧ Offset/Setting→Tool Tip Measure



G54
N0 L1
Offset/Setting
2013/8/22
18:12:45
Z+
(X , Y)
Z
Y+
H
X+
M2
ΔZ
M1
●Ready
Auto
Alarm
(1~3)

Auto Tool Function

AutoTool 1

1:One Tool One Workpiece,2:On

3:Many Tools Many Workpie

Delta Z Set

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. H	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	0.000

Do tool tip measure before do Delt
1:Take tool tip to top of good
2:Press F3, Delta Z Set

Machine

X	0.000
Y	0.000
Z	0.000
A	4.158

Relative

X	0.000
Y	0.000
Z	0.000
A	4.158

Aux. Coord

X	0.000
Y	0.000
Z	0.000

3.3.3.4 Auto Tool Function

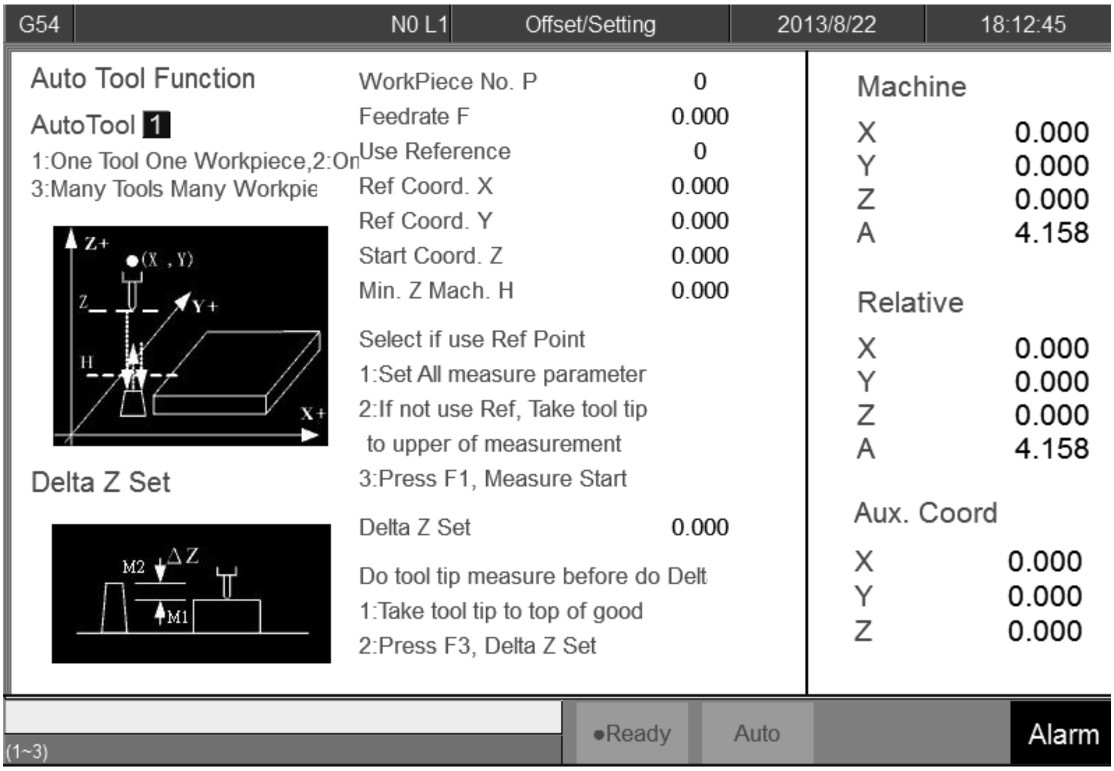
- Function

Z coord. Auto Tool Function is to measure different tool No. by tool alignment equipment. Because the trigger position of the tool alignment equipment is almost the same. User can key in different tool No. tip position into workpiece coordinate for the tool length offset setting, please this machine is equipped with tool alignment equipment.

- Operation description

Offset/Setting→Tool Tip Measure

- One Tool One Workpiece



Auto Tool Function
AutoTool 1
1:One Tool One Workpiece,2:On
3:Many Tools Many Workpie

Delta Z Set

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. H 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 0.000

Do tool tip measure before do Delt
1:Take tool tip to top of good
2:Press F3, Delta Z Set

Machine
X 0.000
Y 0.000
Z 0.000
A 4.158

Relative
X 0.000
Y 0.000
Z 0.000
A 4.158

Aux. Coord
X 0.000
Y 0.000
Z 0.000

✧ Set the Auto Tool number as 1(left-upper corner).

✧ Set the WorkPiece No. P as the workpiece coordinate.

- Table X- Coordinate table

Workpiece coord No. P	Workpiece coord	Workpiece coord No. P	Workpiece coord
0	Aux. Coord	1	G54
2	G55	3	G56

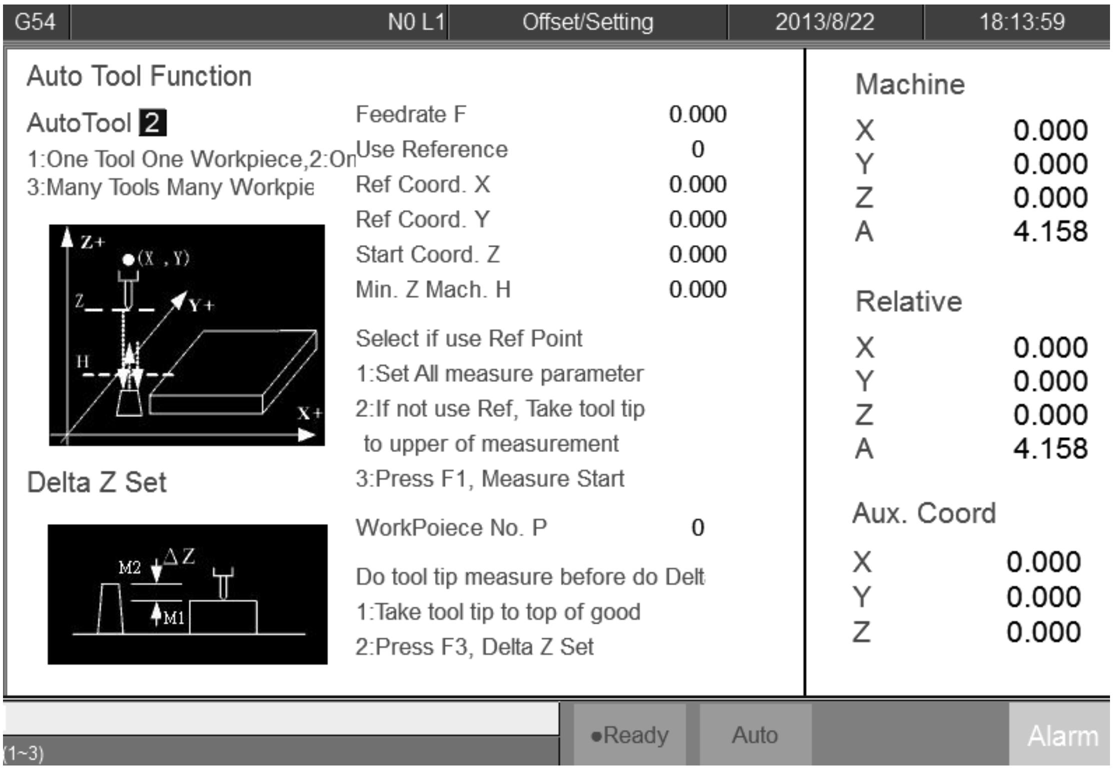
4	G57	5	G58

- Table X- Coordinate table
  - ✧ Set the Feedrate F for the first time alignment and pull back everytime.
  - ✧ Please skip to next step setting XY Ref. Coord. Teach if tool alignment equipment already has fixed mechanical coordinate. If current position is the same with tool alignment equipment, please set Use Reference as 0.
  - ✧ While setting the XY Ref. Coord. Teach, you will need code “520”, enter it and choose yes. These vaules will be entered in to Ref coord.X and Ref coord.Y.
  - ✧ Move the cursor to Start Coord. Z, setting the start point for the auto alignment. Press F10 and then press Z Mach. Coord. Teach, you can insert the current Z coord. value into Start Coord. Z.
  - ✧ Move the cursor to the Min. Z Mach. H, setting the minimum height for the alignment. System will pop out warning masseage and stop if the tool is lower than it.
  - ✧ Please switch to auto mode after setting these and then press Start.
  - ✧ After finishing alignment, this tool height value will be saved into Aux. Coord. and External Shift.
  - ✧ Move the tool tip (Manual) to the surface of workpiece, press Z Delta Set, the distance between alignment tool

and workpiece surface will be calculate and set into this workpiece Z coordinate.

✧ One Tool One Workpiece is finished

- One Tool Many Workpieces



Auto Tool Function

AutoTool 2

1:One Tool One Workpiece,2:On

3:Many Tools Many Workpie

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. H 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

WorkPoiece No. P 0

Do tool tip measure before do Delt

1:Take tool tip to top of good

2:Press F3, Delta Z Set

Machine	
X	0.000
Y	0.000
Z	0.000
A	4.158

Relative	
X	0.000
Y	0.000
Z	0.000
A	4.158

Aux. Coord	
X	0.000
Y	0.000
Z	0.000

(1~3)

- ✧ Set the Auto Tool number as 2.(left-upper corner)
- ✧ Set the feed rate F for the first time alignment and pull back every time.
- ✧ Please skip to next step setting XY Ref. Coord. Teach if tool alignment equipment already has fixed mechanical coordinate. If current position is the same with tool alignment equipment, please set Use Reference as 0.
- ✧ While setting the XY Ref. Coord. Teach, you will need code “520”, enter it and choose yes. These values will be entered in to Ref coord.X and Ref coord.Y.
- ✧ Move the cursor to Start Coord. Z, setting the start point for the auto alignment. Press F10 and then press Z

Mach. Coord. Teach, you can insert the current Z coord. value into Start Coord. Z.

✧ Move the cursor to the Min. Z Mach. H, setting the minimum height for the alignment. System will pop out warning message and stop if the tool is lower than it.

✧ Please switch to auto mode after setting these and then press Start.

✧ After finishing alignment, this tool height value will be saved into Aux. Coord. and External Shift.

✧ Move the tool tip (Manual) to the surface of workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.

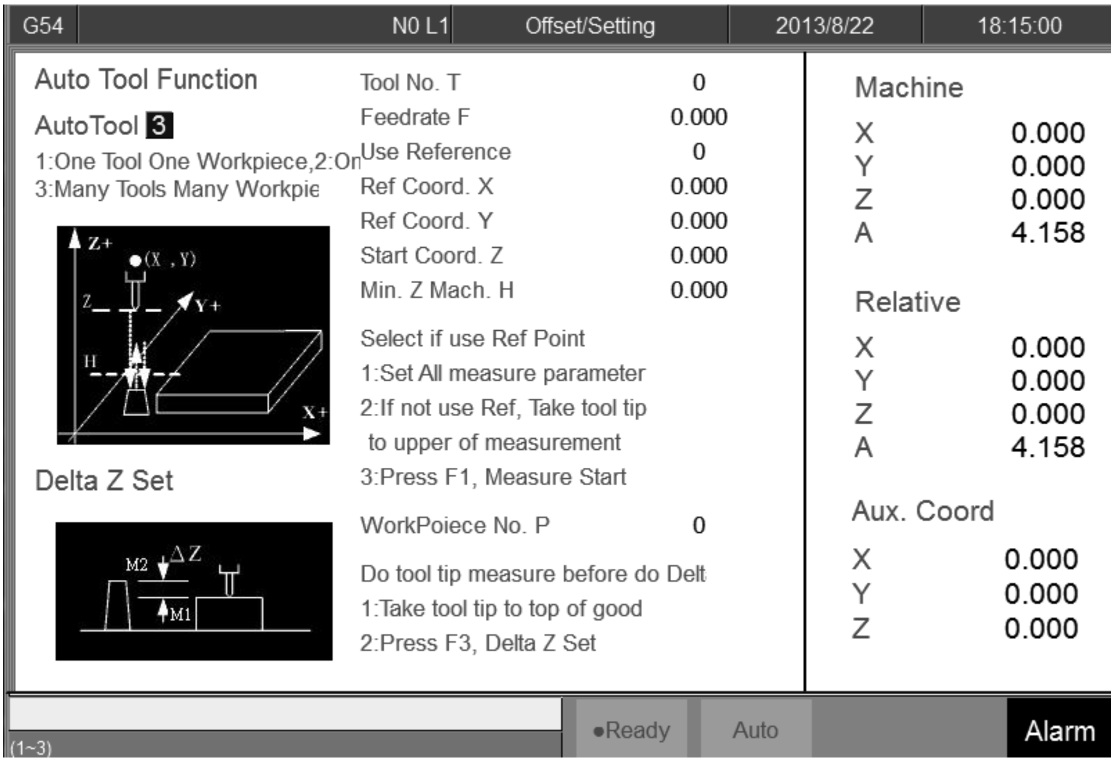
✧ Move the tool tip (Manual) to the surface of next workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.

Workpiece coord No. P	Workpiece coord	Workpiece coord No. P	Workpiece coord
1	G54	2	G55
3	G56	4	G57
5	G58		

Z Delta set table

✧ If you have another workpieces need to set do the previous step again.

- Many Tool Many Workpieces



Auto Tool Function

AutoTool 3

1:One Tool One Workpiece,2:On
3:Many Tools Many Workpie

Delta Z Set

Tool No. T 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. H 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

WorkPoiece No. P 0

Do tool tip measure before do Delt
1:Take tool tip to top of good
2:Press F3, Delta Z Set

Machine
X 0.000
Y 0.000
Z 0.000
A 4.158

Relative
X 0.000
Y 0.000
Z 0.000
A 4.158

Aux. Coord
X 0.000
Y 0.000
Z 0.000

- ✧ Set the Auto Tool number as 3.(left-upper corner)
- ✧ Set the Tool No. T for the tool you want to align.
- ✧ Set the feed rate F for the first time alignment and pull back every time.
- ✧ Please skip to next step setting XY Ref. Coord. Teach if tool alignment equipment already has fixed mechanical coordinate. If current position is the same with tool alignment equipment, please set Use Reference as 0.
- ✧ While setting the XY Ref. Coord. Teach, you will need code “520”, enter it and choose yes. These values will be entered in to Ref coord.X and Ref coord.Y.
- ✧ Move the cursor to Start Coord. Z, setting the start point for the auto alignment. Press F10 and then press Z

Mach. Coord. Teach, you can insert the current Z coord. values into Start Coord. Z.

- ✧ Move the cursor to the Min. Z Mach. H, setting the minimum height for the alignment. System will pop out warning message and stop if the tool is lower than it.
- ✧ Please switch to auto mode after setting these and then press Start.
- ✧ After finishing alignment, this tool height value will be saved into Aux. Coord. and External Shift.
- ✧ Move the tool tip (Manual) to the surface of workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.
- ✧ Move the tool tip (Manual) to the surface of next workpiece, press Z Delta Set, the distance between alignment tool and workpiece surface will be calculate and set into this workpiece Z coordinate.
- ✧ If you have another workpieces and tools need to set do the previous step again.
- ✧ Finish alignment.

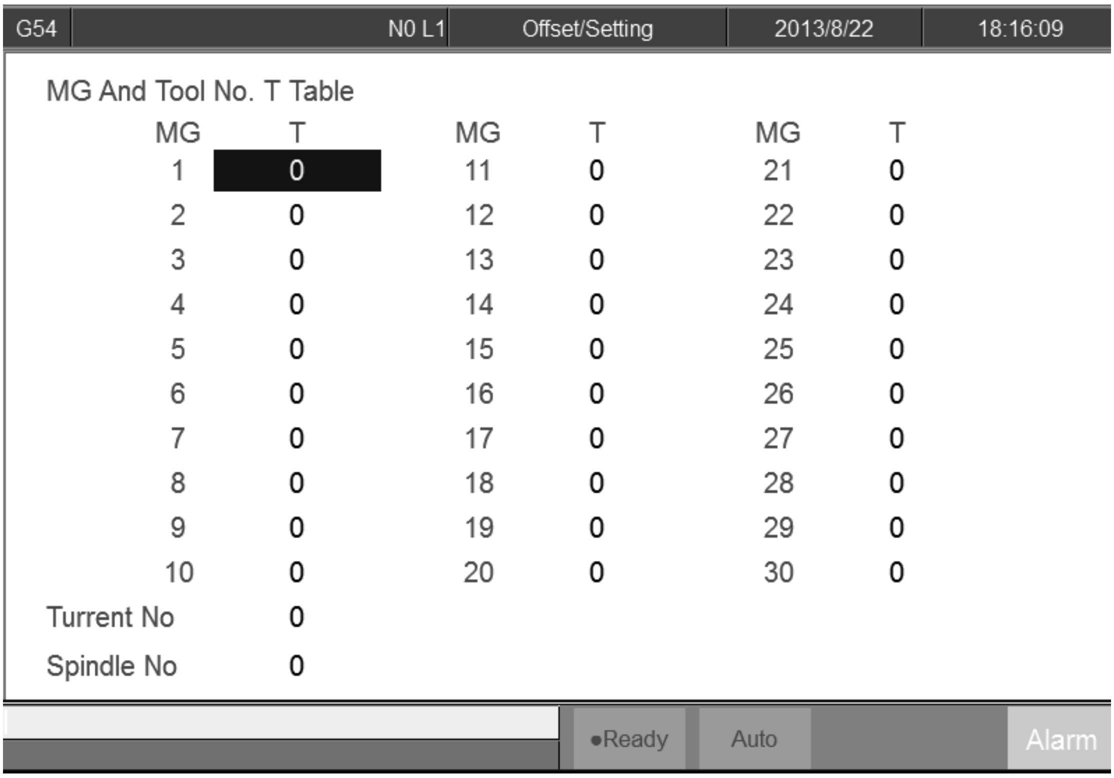
3.3.3.5 Tool No. Setting

This Chapter will describe how to set the tool No.

- ● Purpose:
  - ✧ We need to confirm the relation between Tool No. and real tool in order to change tool correctly.
- ● Condition:

- ✧ Both Auto mode or Manual mode are OK..

- ● Operation Method
  - ✧ Tool Set→Tool No.
  - ✧ You will see the table about Tool No. and Tool MG.
  - ✧ MG5—T = 7 means we install tool No.7 at Tool case No.5


G54
N0 L1
Offset/Setting
2013/8/22
18:16:09

MG And Tool No. T Table

MG	T	MG	T	MG	T
1	0	11	0	21	0
2	0	12	0	22	0
3	0	13	0	23	0
4	0	14	0	24	0
5	0	15	0	25	0
6	0	16	0	26	0
7	0	17	0	27	0
8	0	18	0	28	0
9	0	19	0	29	0
10	0	20	0	30	0

Turrent No 0

Spindle No 0

•Ready
Auto
Alarm

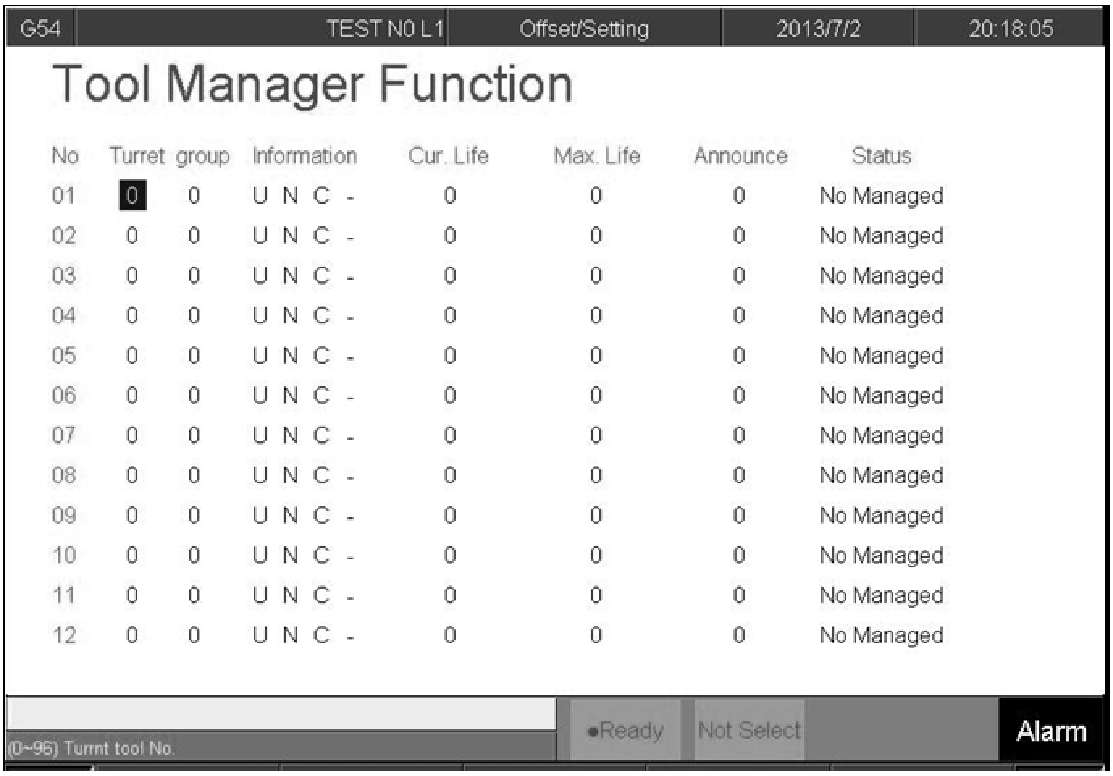
- ✧ This table will renew during change a new tool.

- ● Time to Modify
  - ✧ First initialized when it is made from factory.
  - ✧ Need to confirm the MG No. and T No. Manually when it is disordered.

3.3.3.6 Tool Life Manager Function

- Purpose:

Record the status of all cutting tool on machine, make users know whether cutting tool reached to Max. Life, avoid machining in case cutting tool is broken. This function needs a related PLC setting. In case, customer need to use this function, please contact to machine maker.



Tool Manager Function

No	Turret	group	Information	Cur. Life	Max. Life	Announce	Status
01	0	0	UNC-	0	0	0	No Managed
02	0	0	UNC-	0	0	0	No Managed
03	0	0	UNC-	0	0	0	No Managed
04	0	0	UNC-	0	0	0	No Managed
05	0	0	UNC-	0	0	0	No Managed
06	0	0	UNC-	0	0	0	No Managed
07	0	0	UNC-	0	0	0	No Managed
08	0	0	UNC-	0	0	0	No Managed
09	0	0	UNC-	0	0	0	No Managed
10	0	0	UNC-	0	0	0	No Managed
11	0	0	UNC-	0	0	0	No Managed
12	0	0	UNC-	0	0	0	No Managed

(0~96) Turrnt tool No.

- Condition
    - ✧ Both auto and manual can be used.
- Operation
    - ✧ Pr.3228 is the on/off control of 「Tool management」
- Description
    - ◆ Turret
    - ✧ Current tool case no. that tool located.
    - ◆ Group

- ✧ Same kind of tool within in one group, if the first tool of that group is on lock state or 「Tool life」 is end, whenever user use T code to change the tool, system will skip the first tool and use the second one, when the second one is lock or「Tool life」 is end, will use the third one, and so on.

◆ Tool information (Status)

- ✧ L—Lock / U—Unlock
- ✧ If the status of tool is lock, that tool cann't be use and when T code is use to change the tool, system will skip that tool.
- ✧ B—Large diameter Tool/ N—Normal Diameter Tool
- ✧ Adjacent side of large diameter tool set is empty(for display)
- ✧ T—working time T / C—Number of working times
- ✧ Decide the current life time, the maximum life time, life time prediction, unit of timing and number of time.
- ✧ R—effective value / - —non effective value.
- ✧ Current tool are using tool management or not.

◆ Current Life time

- ✧ Current Tool Using Condition

◆ Maximum Life Time

- ✧ Maximum lifetime of tool.

◆ Lifetime prediction

- ✧ When lifetime of tool is greater than lifetime prediction, alarm will be show up.

- Current Status of Tool

(0)Without management: Set values are not effective.

(1) Without use: Lifetime of tool is zero.

(2) Usable:0< Tool Life Time <lifetime prediction

(3) End prediction: Lifetime prediction < Tool Lifetime < Maximum Lifetime

(4) End of Life: Maximum Lifetime < Tool Lifetime

(5)ware of tool

3.4 Program Preparation and Execute Machining

3.4.1 Specifying Machining Program

- Condition
  - Except single block mode
- Operation
  - Specify current edit program as machining program
  - Switch to edit page
  - Select F1-Excute, and the program will be designated as the machining program
  - Specify machining program in file manager.
  - Switch to the "File Management" page
  - Move the cursor to the expected program and press Enter
  - Select F1-Excute, and the program will be designated as the machining program
- Confirmation:

- ✧ There are two ways to confirm whether machining program is specified successfully.
- ✧ The screen displays the correct machining program name
- ✧ The content of machining program is displayed when pressing F4-Monitor

3.4.2 Simu. Setting

SYNTEC's controller provides simulation program, after editing machining program, users can easily simulate the path machining process, this feature also contains checking features that help users to quickly verify the syntax error in machining program or unreasonable actions, we suggest users should use this function to check machining program.

- Condition
    - ✧ Except single block mode
- Operation
    - ✧ In the "File Management" page, select the program you want to edit after completing edit program, press F7-Simulation
    - ✧ Screen will switch to the "graphic simulation" page and scan the contents of the program
- Detail description

  -Simulation screen

    - ✧ The solid line represents the cutting path
    - ✧ The dashed line represents the moving path
    - ✧ In the scanning process, if there is any syntax or content error, they will be displayed on the screen with corresponding error line number.
    - ✧ F1-step: To simulate tool path corresponding to single block in NC files. It is used for coordinate checking purpose.
    - ✧ F2-Continue: System scans the whole program first before executing simulation.

- ✧ F3-zoom: To zoom in/out the workpiece graph. Users can use the arrow key “←”, “↑”, “→”, “↓” to move the frame to the determined area, use “PageUp” “PageDn” to zoom in/out this area. After selecting zoom scales, press “enter” to finish.
- ✧ F5- simu. Setting: To set simulation parameter

3.4.3 Machining Test

3.4.3.1 MPG Simulation

- ● Condition
  - ✧ Only for single block and auto mode
- ● Operation
  - ✧ Select Auto mode
  - ✧ Press MPG simulation button on operation panel
  - ✧ Turn MPG to execute machining
  - ✧ If MPG is turned in CW direction, Program will be run from current NC line down to below NC line
  - ✧ If MPG is turned in CCW direction, Program will be run from current NC line up to above NC line
- ● Confirm
  - ✧ We can confirm MPG simulation successful or not by these two method.
  - ✧ Not machining, execute MPG function, and then in the monitor page try to rotate the MPG. If you can see G01 is zero before you rotate and has value after you rotate MPG.

- ✧ While machining, execute MPG function, and then machine will stop to 0 immediately, until you rotate MPG or cancel MPG function.

3.4.3.2 Single Block

- ● Condition
  - ✧ Only for single block and auto mode
- ● Operation
  - ✧ Select Auto mode
  - ✧ Press single block button on operation panel
  - ✧ After programming and decelerating to 0, system status changes to B-stop
  - ✧ Press CYCLE START again
  - ✧ After completing next single block in NC file, system will be on B-stop status again

3.4.3.3 Start MPG Coordinate

This section will introduce how to execute the function of part count and work record

- ● Condition
  - ✧ MDI or Auto mode
- ● Operation Method
  - ✧ F4 Monitor→F6 Start MPG Coordinate
  - ✧ You will see the MPG coordinate system window.
  - ✧ Switch to the MPG mode.
  - ✧ Rotate the MPG, you can see the value at this window.
- ● Confirmation

  - ✧ Switch to workpiece coordinate page, confirm the value in MPG Shift is correct.
- ● Relative parameter
  - ✧ .Pr 3201 Set the Lathe Rule, you must set 0 if you want to use Start MPG Coordinate function

- ● Note
  - ✧ R606 must be 1
  - ✧ Only accept MPG command, JOG&INJOG are invalid.
  - ✧ It is still work while Machine Lock
- ● Time to disable

    -Keep the MPG coordinate while starting the machine.

    -Don’t clear the value while G54/G55 is changed.

    -Don’t clear the value while change the machining code.

    -Don’t clear the value after go back to Home.

    -Don’t clear the value after go back to referent point by G28/G29/G30.
- ● Limit
  - ✧ Use the Start MPG Coordinate function under MPG simulation mode. Command will send to MPG coordinate while rotate the MPG until you finish the MPG coordinate function, the command will go to MPG simulation.

3.4.3.4 Break Point

This section will introduce how to execute the function of part count and work record

- Condition
  - ✧ Under Auto mode
- Operation Method
  - ✧ Move cursor to the Start Block No.
  - ✧ Enter the line number you want return
  - ✧ Line number can be refer to break point line number.
  - ✧ System will pop out confirm window.
  - ✧ Press enter, wait the cursor move to the line number you choose.
  - ✧ Execute.

3.4.4 Machining Monitor

This section will introduce how to use Break Point Return function.

3.4.4.1 Part Count Manager

- Description
  1. Total accum part
  - ✧ The total accumulative part machined by CNC
  2. Required part count
  - ✧ Once machining program specifies the needed workpiece number, and CNC continues machining when it meets M99, if the demand workpiece number is reached, machining process will be paused and notification message will be displayed.

3. Part count
   - ✧ Once CNC continues machining when it meets M99, this number will be accumulated until reset
   - ✧ Part count reset (clear to 0) condition
   - ✧ Required part count is reached
   - ✧ Change machining files
   - ✧ Modify the required part count, and the required part count is smaller than part count.

3.4.4.2 Work Record Function

- Description
  - ✧ Once CNC continues machining when it meets M99, work record function will automatically record status
- Work record condition
  - ✧ Required part count is reached
  - ✧ Change machining files
  - ✧ Modify the required part count, and the required part count is smaller than part count.

3.4.5 Alarm Processing

In order to avoid wrong operation effects on safety of human and machine, the system and PLC have many kinds of protection. When these protection conditions are triggered, the system will issue warning or alarm to users. This section will describe how to view and troubleshooting alarm.

3.4.5.1 Emergency Stop

Machine failure or unexpected movements may cause un-safety for human and machine. Pressing emergency stop button, you can immediately stop the machine.

3.4.5.2 Alarm Display

Alarm is basically divided into the pending alarm and history alarm.

3.4.5.3 Pending Alarm

- ✧ The current status of system alarm
- ✧ Once an alarm occurs, the controller will issue alarm and display the current alarm content on screen
- ✧ Press ESC to jump that window
- ✧ If the alarm is still not remove, press reset button, alarm window will be not displayed.
- ✧ Press F8-Maintain to display pending alarm contents.

3.4.5.4 History Alarm

Accessing into this page enables user to see all system alarms which have occurred, so users may find out the alarm reason.

- Command:
  - ✧ F8- maintain→ F1-Alarm→F2 History alarm
- Display history alarm
  - ✧ The smaller No. alarm is, the sooner alarm occurres

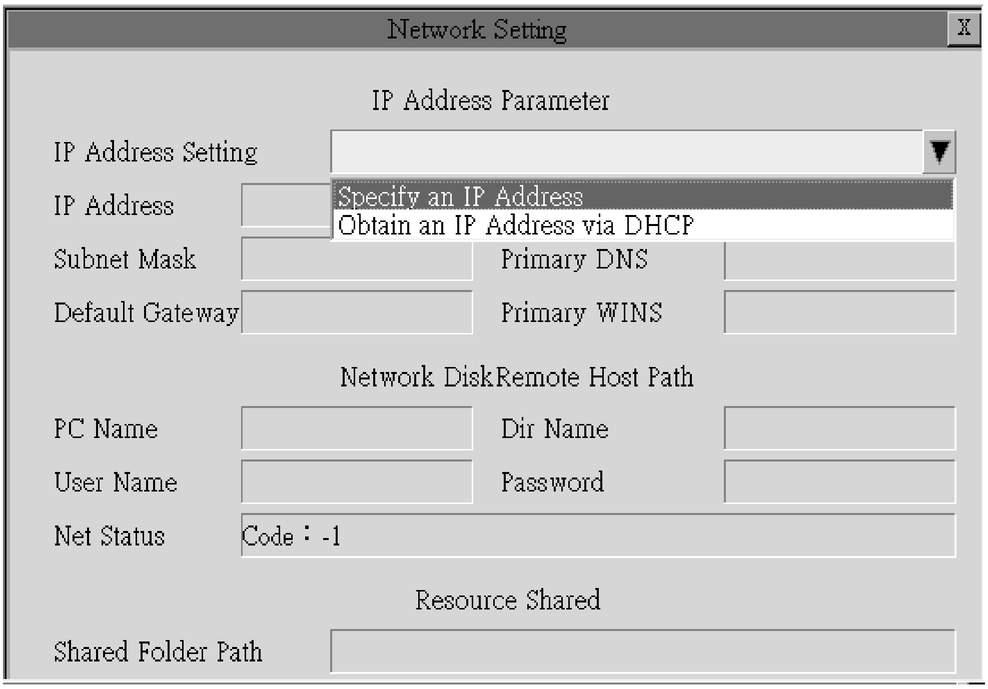
3.4.5.5 Save Alarm

In case users need support from machinery manufactory to repair once alarm appears, users can export the alarm contents to an external storage device, and send it to machinery manufactory. By that way, they could clarify and find out the possible reasons.

- Operation
  - ✧ Insert the external storage device into controller, or set the corresponding network folder
  - ✧ Switch to “Alarm” page(F8-maintain→F1-alarm)
  - ✧ To export the pending alarm, press F1-pending alarm→ F3-save alarm
  - ✧ To export the history alarm, press F2-history alarm→ F3-save alarm
  - ✧ External storage device will be displayed on screen, select the destination folder to save
  - ✧ Select OK to complete export alarm content
- File Name
  - ✧ Actual alarm：Actalm.txt
  - ✧ History alarm：Histalm.txt

3.4.6 Network Setting

1. On the interface screen, press down**“F8 Maintain” => “F2 Network Setting”** to access IP address setting.
2. **IP Address Setting**: select “Specify an IP Address” when the PC connects with controller directly. And select “Obtain an IP Address via DHCP” if using network connection via Dynamic Host Configuration Protocol
3. **IP Address**: if you select “Specify an IP Address”, enter the free IP address


Network Setting
X
IP Address Parameter
IP Address Setting
Specify an IP Address
Obtain an IP Address via DHCP
IP Address
Subnet Mask
Primary DNS
Default Gateway
Primary WINS
Network DiskRemote Host Path
PC Name
Dir Name
User Name
Password
Resource Shared
Shared Folder Path

Net Status Code：-1

4. **Subnet Mask**: Enter the IP address for subnet mask (the same with PC subnet mask).
5. **PC Name**: Enter the full computer name of your PC.

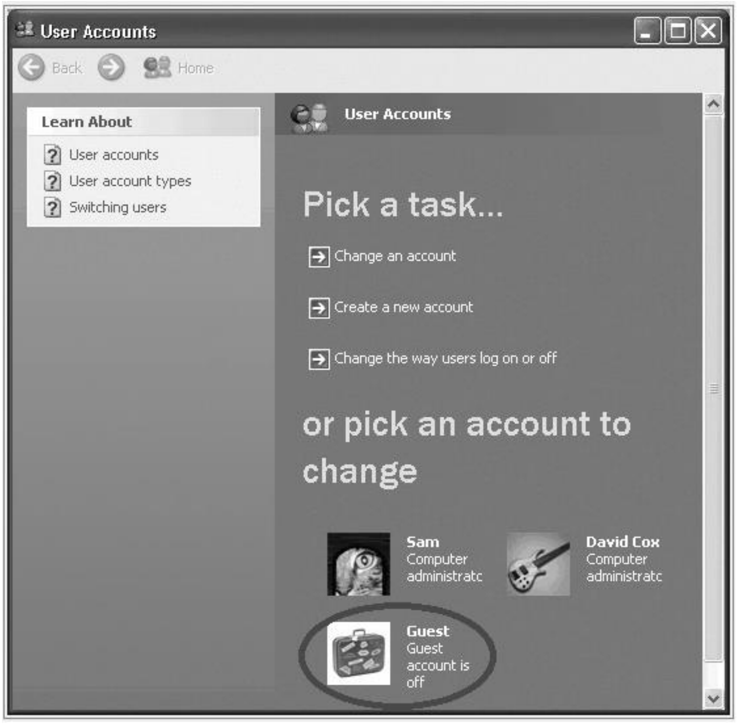
6. **Dir Name**: Enter the sharing folder name (the same name with PC sharing folder)
7. **User Name**: Enter GUEST
8. Press「**F1 OK**」, and then reboot controller to finish installation.

3.4.7 PC Setting

3.4.7.1 XP OS

1. Guest account setting

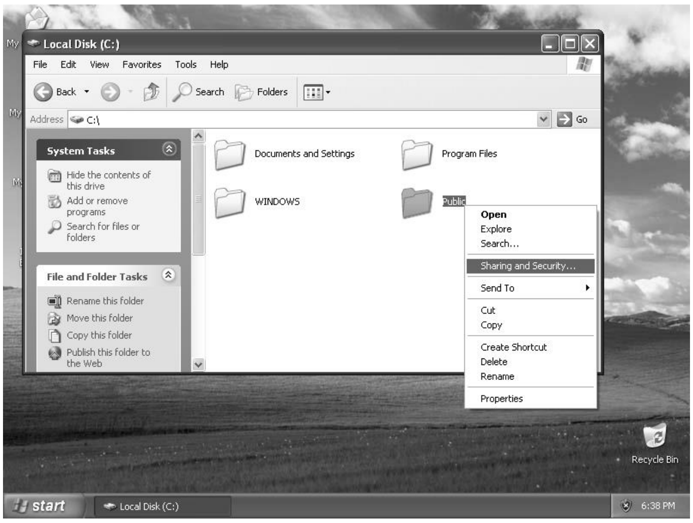
Log in as an Administrator and select "start"→ "control panel"→"user account"→ Guest

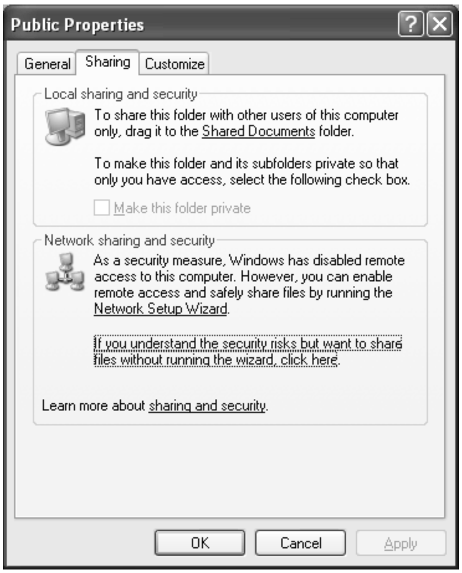

User Accounts
Back
Home
Learn About
User accounts
User account types
Switching users
User Accounts
Pick a task...
Change an account
Create a new account
Change the way users log on or off
or pick an account to change
Sam
Computer administratc
David Cox
Computer administratc
Guest
Guest account is off

2. Sharing resource setting

- Right click the folder you want to share and select "Sharing and security"

- Click on “If you understand security risks but want to share files without running the wizard, click here”


Local Disk (C:)
File Edit View Favorites Tools Help
Back
Search
Folders
Address C:\
Go
System Tasks
Hide the contents of this drive
Add or remove programs
Search for files or folders
File and Folder Tasks
Rename this folder
Move this folder
Copy this folder
Publish this folder to the Web
Documents and Settings
Program Files
WINDOWS
Public
Open
Explore
Search...
Sharing and Security...
Send To
Cut
Copy
Create Shortcut
Delete
Rename
Properties
Recycle Bin
start
Local Disk (C:)
6:38 PM


Public Properties
General
Sharing
Customize
OK
Cancel
Apply

Local sharing and security

To share this folder with other users of this computer only, drag it to the Shared Documents folder.

To make this folder and its subfolders private so that only you have access, select the following check box.

☐ Make this folder private

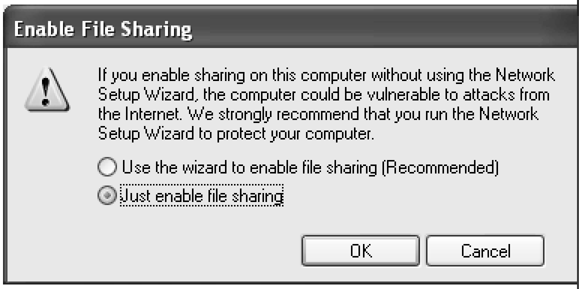
Network sharing and security

As a security measure, Windows has disabled remote access to this computer. However, you can enable remote access and safely share files by running the Network Setup Wizard.

If you understand the security risks but want to share files without running the wizard, click here.

Learn more about sharing and security.

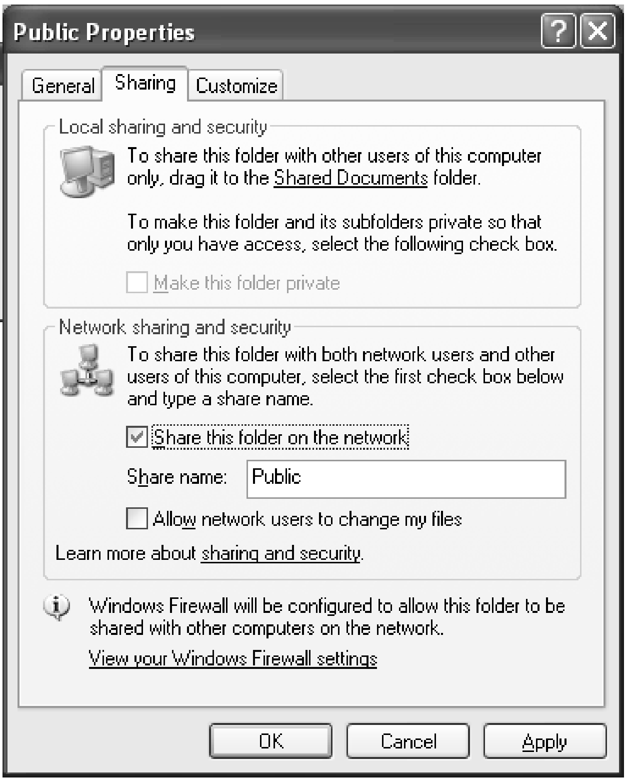
3. Click “OK” to confirm sharing setting; Select “Share this folder on the network”, and “Allow network users to change my files”.


Enable File Sharing
OK
Cancel

If you enable sharing on this computer without using the Network Setup Wizard, the computer could be vulnerable to attacks from the Internet. We strongly recommend that you run the Network Setup Wizard to protect your computer.

○ Use the wizard to enable file sharing (Recommended)

● Just enable file sharing


Public Properties
General
Sharing
Customize
OK
Cancel
Apply

Local sharing and security

To share this folder with other users of this computer only, drag it to the Shared Documents folder.

To make this folder and its subfolders private so that only you have access, select the following check box.

☐ Make this folder private

Network sharing and security

To share this folder with both network users and other users of this computer, select the first check box below and type a share name.

☑ Share this folder on the network

Share name: Public

☐ Allow network users to change my files

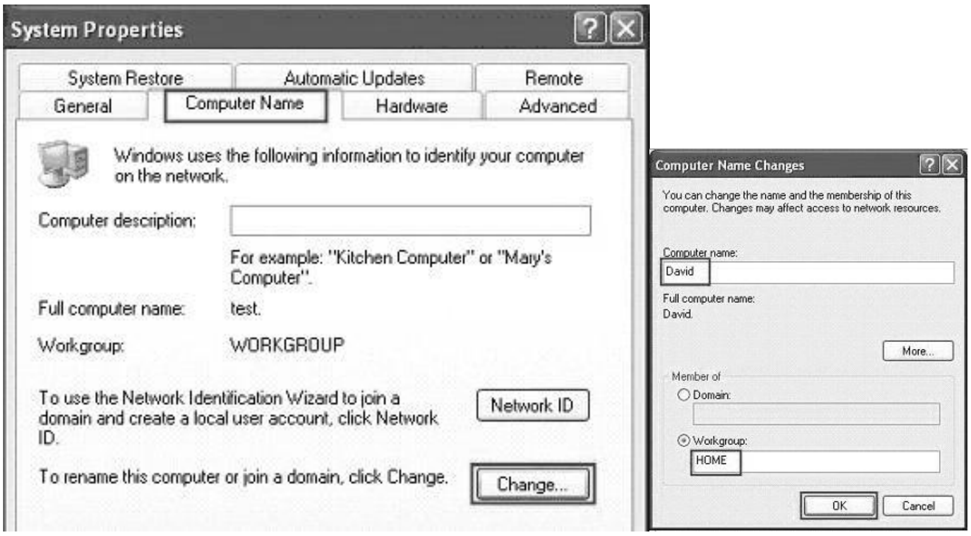
Learn more about sharing and security.

Windows Firewall will be configured to allow this folder to be shared with other computers on the network.

View your Windows Firewall settings

4. Setting PC name and workgroup

"Start"→ "control panel"→"System"→ "change" to set "Computer Name" and "Workgroup", and remember these setting contents to use later on when setting controller.


System Properties
System Restore
Automatic Updates
Remote
General
Computer Name
Hardware
Advanced
Network ID
Change...
Computer Name Changes
More...
OK
Cancel

Windows uses the following information to identify your computer on the network.

Computer description:

For example: "Kitchen Computer" or "Mary's Computer".

Full computer name: test.

Workgroup: WORKGROUP

To use the Network Identification Wizard to join a domain and create a local user account, click Network ID.

To rename this computer or join a domain, click Change.

You can change the name and the membership of this computer. Changes may affect access to network resources.

Computer name: David

Full computer name: David.

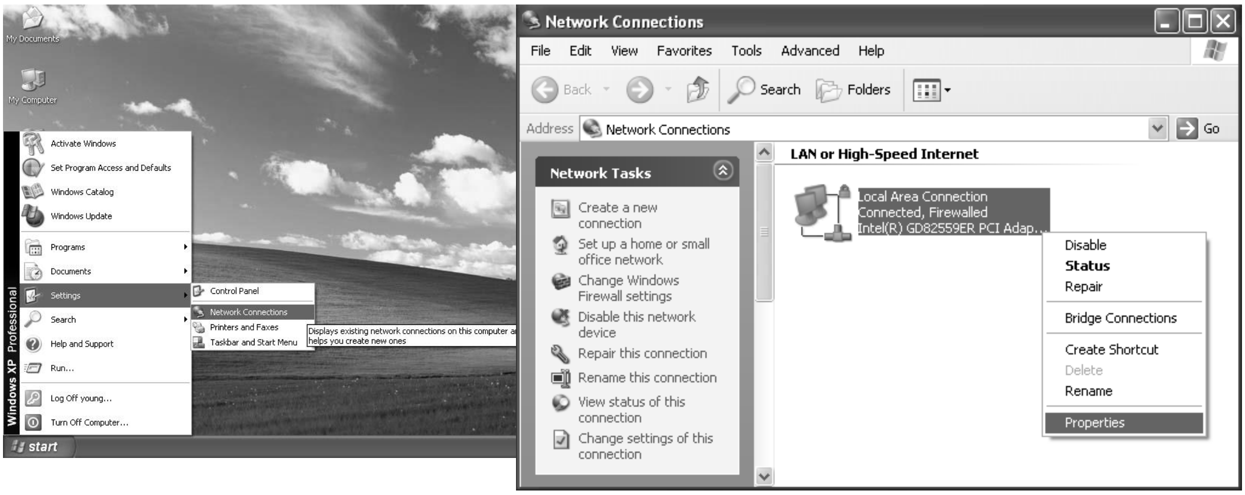
Member of

○ Domain:

● Workgroup: HOME

5. TCP/IP setting

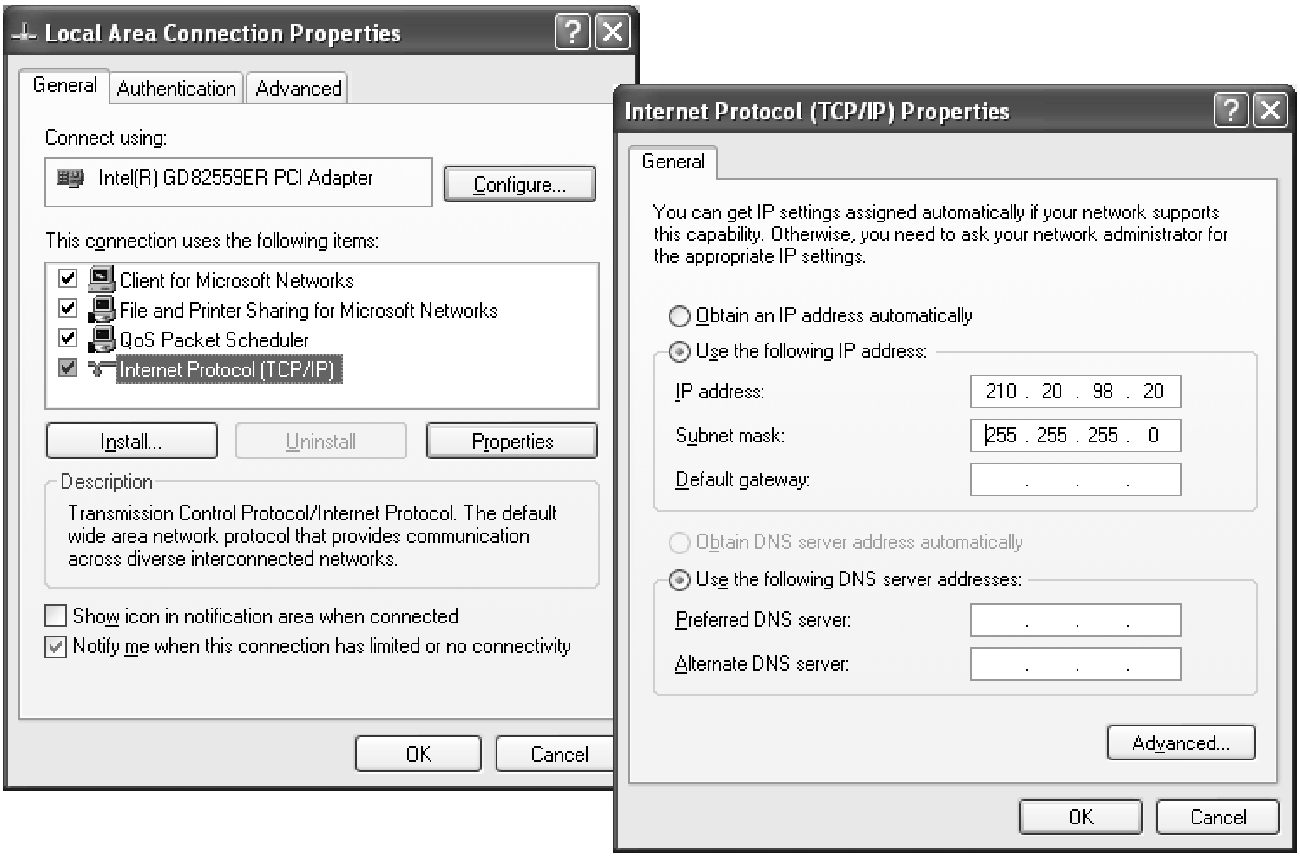
“Start” => “Setting” => “Network connections” and right click on


My Documents
My Computer
Activate Windows
Set Program Access and Defaults
Windows Catalog
Windows Update
Programs
Documents
Settings
Search
Help and Support
Run...
Log Off young...
Turn Off Computer...
Windows XP Professional
start
Control Panel
Network Connections
Printers and Faxes
Taskbar and Start Menu
Displays existing network connections on this computer a helps you create new ones
Network Connections
File Edit View Favorites Tools Advanced Help
Back
Search
Folders
Address
Network Connections
Go
Network Tasks
Create a new connection
Set up a home or small office network
Change Windows Firewall settings
Disable this network device
Repair this connection
Rename this connection
View status of this connection
Change settings of this connection
LAN or High-Speed Internet
Local Area Connection
Connected, Firewalled
Intel(R) GD82559ER PCI Adap...
Disable
Status
Repair
Bridge Connections
Create Shortcut
Delete
Rename
Properties

“Properties”, and select “Internet Protocol [TCP/IP]”

✧ Jumper cable (without HUB), select “use the following IP address” and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)

✧ Network cable (with HUB), select “Obtain an IP address automatically”


Local Area Connection Properties
General
Authentication
Advanced
Configure...
Install...
Uninstall
Properties
OK
Cancel
Internet Protocol (TCP/IP) Properties
General
Advanced...
OK
Cancel

Connect using:

Intel(R) GD82559ER PCI Adapter

This connection uses the following items:

- ☑ Client for Microsoft Networks
- ☑ File and Printer Sharing for Microsoft Networks
- ☑ QoS Packet Scheduler
- ☑ Internet Protocol (TCP/IP)

Description

Transmission Control Protocol/Internet Protocol. The default wide area network protocol that provides communication across diverse interconnected networks.

☐ Show icon in notification area when connected

☑ Notify me when this connection has limited or no connectivity

You can get IP settings assigned automatically if your network supports this capability. Otherwise, you need to ask your network administrator for the appropriate IP settings.

○ Obtain an IP address automatically

● Use the following IP address:

IP address: 210 . 20 . 98 . 20

Subnet mask: 255 . 255 . 255 . 0

Default gateway: . . .

○ Obtain DNS server address automatically

● Use the following DNS server addresses:

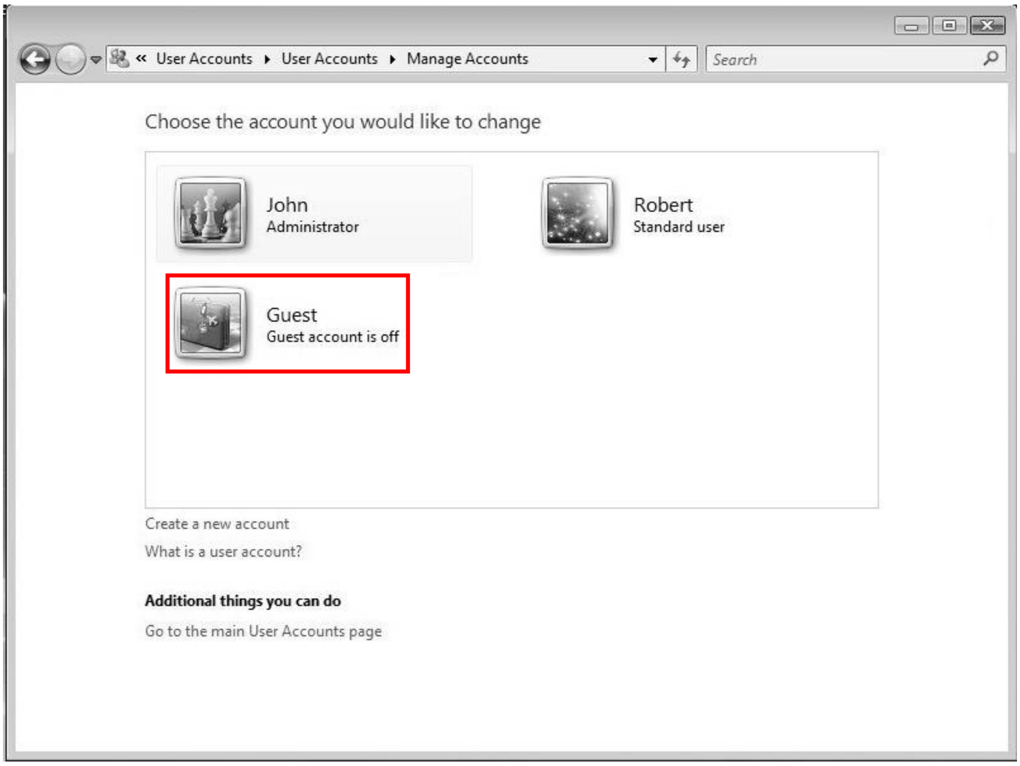
Preferred DNS server: . . .

Alternate DNS server: . . .

3.4.7.2 VISTA OS

1. Guest account setting

Log in as Administrator and select “Start”→ “Control Panel”→ “User Account”→ Guest


« User Accounts ▸ User Accounts ▸ Manage Accounts
Search

Choose the account you would like to change

John
Administrator

Robert
Standard user

Guest
Guest account is off

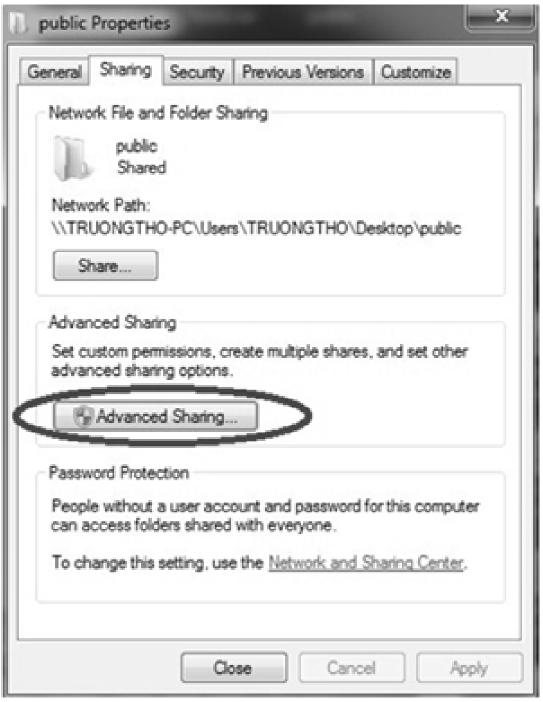
Create a new account
What is a user account?
Additional things you can do
Go to the main User Accounts page

2. Sharing Resource Setting

Create a sharing folder, and change this folder’s setting to offer controller to use, as the below figure.

A. Click on “advanced sharing”

B. Click on “share this folder”

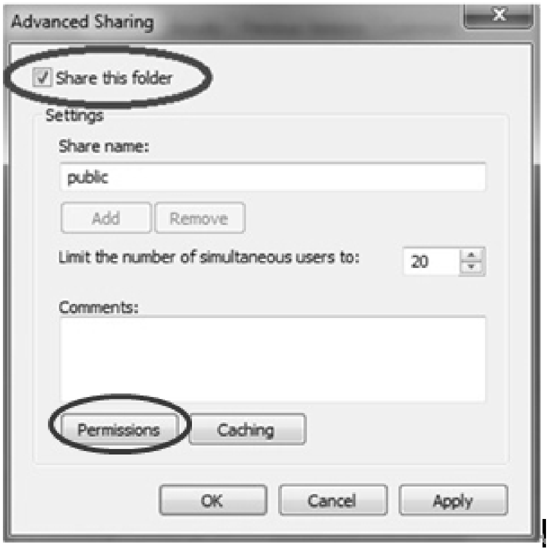

public Properties
General
Sharing
Security
Previous Versions
Customize
Network File and Folder Sharing
public
Shared
Share...
Advanced Sharing
Advanced Sharing...
Password Protection
Close
Cancel
Apply

Network Path:
\\TRUONGTHO-PC\Users\TRUONGTHO\Desktop\public

Set custom permissions, create multiple shares, and set other advanced sharing options.

People without a user account and password for this computer can access folders shared with everyone.

To change this setting, use the Network and Sharing Center.


Advanced Sharing
Settings
Add
Remove
Comments:
Permissions
Caching
OK
Cancel
Apply

☑ Share this folder

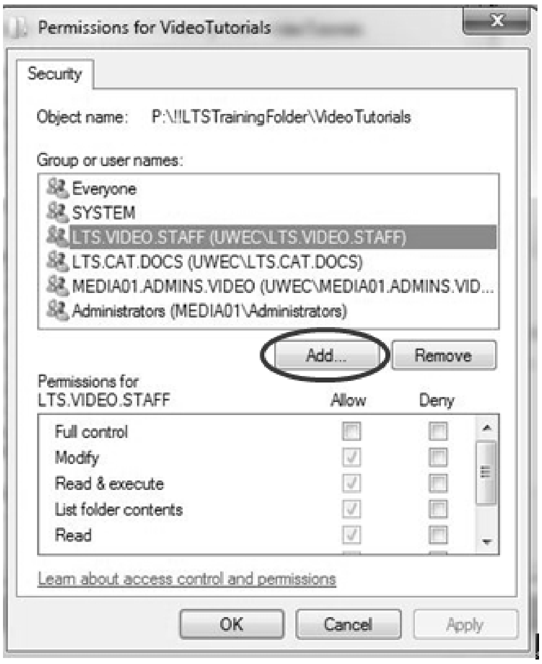
Share name: public

Limit the number of simultaneous users to: 20

C. Click on “permission”

D. Click on “add”

E. Enter “GUEST” as the new group name, click “OK” to complete setting


Permissions for VideoTutorials
Security
Add...
Remove
Allow
Deny
OK
Cancel
Apply
Learn about access control and permissions

Object name: P:\!!LTSTrainingFolder\VideoTutorials

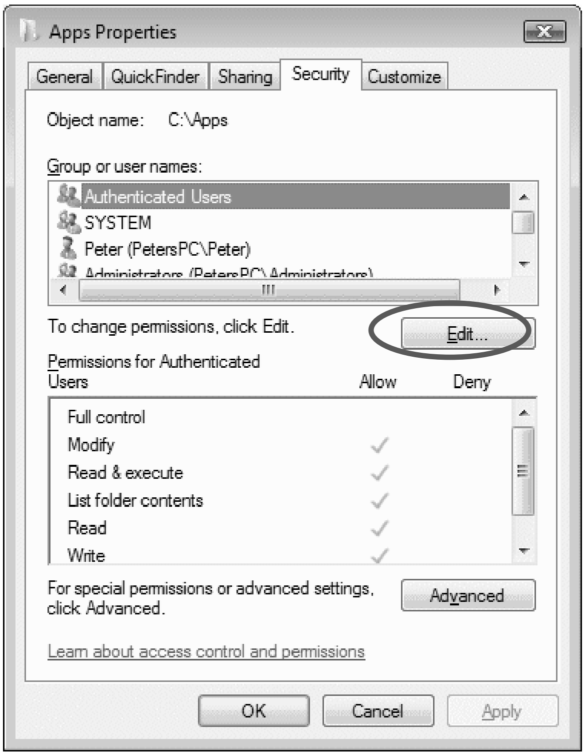
Group or user names:
- Everyone
- SYSTEM
- LTS.VIDEO.STAFF (UWEC\LTS.VIDEO.STAFF)
- LTS.CAT.DOCS (UWEC\LTS.CAT.DOCS)
- MEDIA01.ADMINS.VIDEO (UWEC\MEDIA01.ADMINS.VID...
- Administrators (MEDIA01\Administrators)

Permissions for LTS.VIDEO.STAFF

	Allow	Deny
Full control	☐	☐
Modify	☑	☐
Read & execute	☑	☐
List folder contents	☑	☐
Read	☑	☐

3. Security setting

Right click on folder to share→ properties→ security→ Edit→ add “Guest” as a new group, then open group permissions to maximum.


Apps Properties
General
QuickFinder
Sharing
Security
Customize
Edit...
Advanced
OK
Cancel
Apply
Learn about access control and permissions

Object name: C:\Apps

Group or user names:
- Authenticated Users
- SYSTEM
- Peter (PetersPC\Peter)
- Administrators (PetersPC\Administrators)

To change permissions, click Edit.

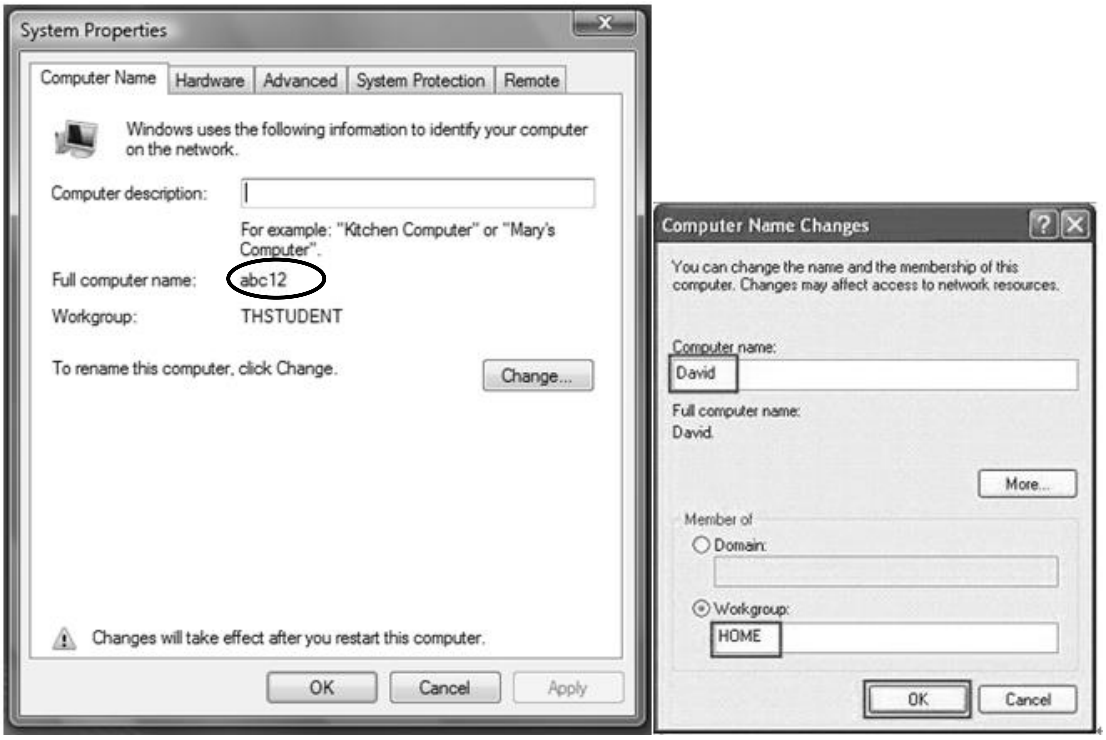
Permissions for Authenticated Users

	Allow	Deny
Full control		
Modify	✓	
Read & execute	✓	
List folder contents	✓	
Read	✓	
Write	✓	

For special permissions or advanced settings, click Advanced.

4. Setting PC name and workgroup

“Start”→ “control panel”→”System”→ “change” to set “Computer Name” and “Workgroup”, and remember these setting contents to use later on when setting controller.



System Properties
Computer Name
Hardware
Advanced
System Protection
Remote
OK
Cancel
Apply
Change...

Windows uses the following information to identify your computer on the network.

Computer description:

For example: "Kitchen Computer" or "Mary's Computer".

Full computer name: abc12

Workgroup: THSTUDENT

To rename this computer, click Change.

Changes will take effect after you restart this computer.

Computer Name Changes
More...
OK
Cancel

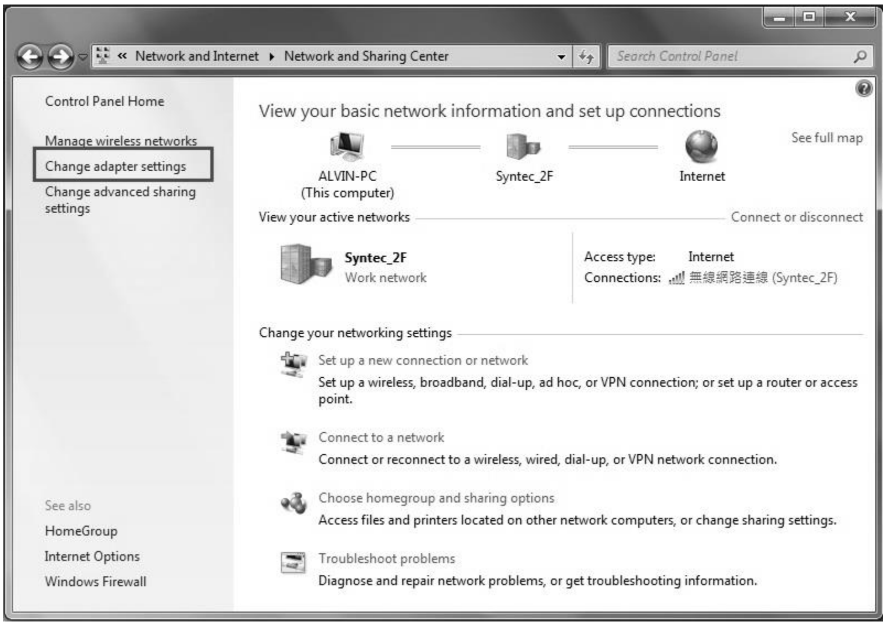
You can change the name and the membership of this computer. Changes may affect access to network resources.

Computer name: David

Full computer name:
David

Member of
○ Domain:
● Workgroup: HOME

5. TCP/IP Setting
   a. “Start” → “control panel” → “Network and Internet”→“Network and Sharing Center” → “Properties”

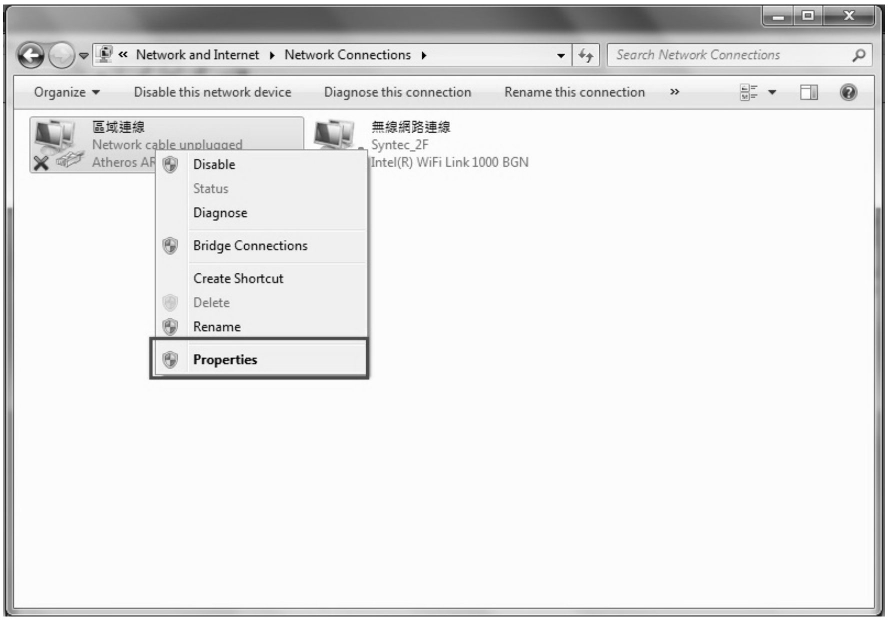

« Network and Internet ▸ Network and Sharing Center
Search Control Panel
Control Panel Home
Manage wireless networks
Change adapter settings
Change advanced sharing settings
See also
HomeGroup
Internet Options
Windows Firewall
View your basic network information and set up connections
See full map
ALVIN-PC
(This computer)
Syntec_2F
Internet
View your active networks
Connect or disconnect
Syntec_2F
Work network
Access type: Internet
Connections: 無線網路連線 (Syntec_2F)
Change your networking settings
Set up a new connection or network
Connect to a network
Choose homegroup and sharing options
Troubleshoot problems

Set up a wireless, broadband, dial-up, ad hoc, or VPN connection; or set up a router or access point.

Connect or reconnect to a wireless, wired, dial-up, or VPN network connection.

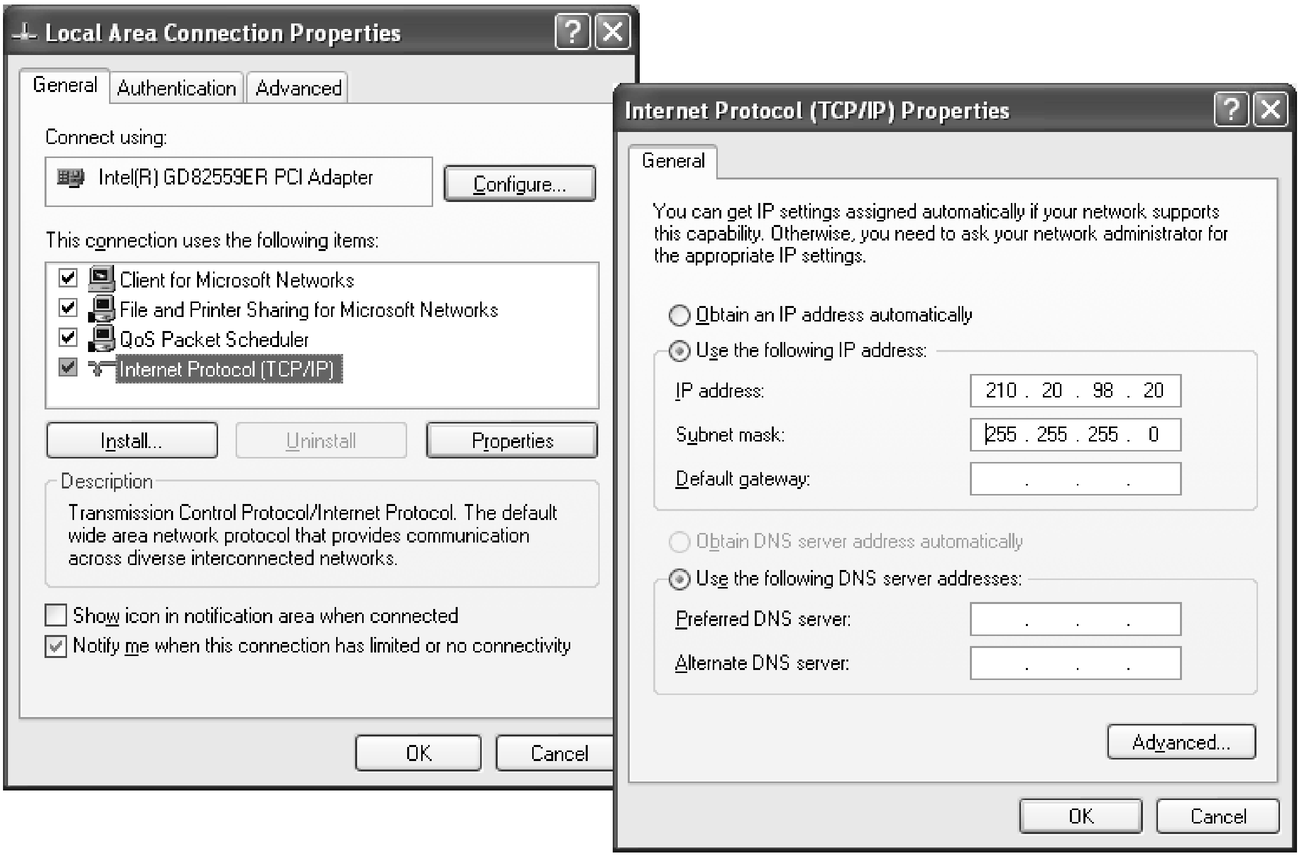
Access files and printers located on other network computers, or change sharing settings.

Diagnose and repair network problems, or get troubleshooting information.


« Network and Internet ▸ Network Connections ▸
Search Network Connections
Organize
Disable this network device
Diagnose this connection
Rename this connection
»
區域連線
Network cable unplugged
Atheros AR
無線網路連線
Syntec_2F
Intel(R) WiFi Link 1000 BGN
Disable
Status
Diagnose
Bridge Connections
Create Shortcut
Delete
Rename
Properties

b. Select “internet protocol(TCP/IP)” as shown below:

- ✧ Jumper cable (without HUB), select “use the following IP address” and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)
- ✧ Network cable (with HUB), select “Obtain an IP address automatically”


Local Area Connection Properties
General
Authentication
Advanced
Configure...
Install...
Uninstall
Properties
Description
OK
Cancel
Internet Protocol (TCP/IP) Properties
General
Advanced...
OK
Cancel

Connect using: Intel(R) GD82559ER PCI Adapter

This connection uses the following items:

☑ Client for Microsoft Networks
☑ File and Printer Sharing for Microsoft Networks
☑ QoS Packet Scheduler
☑ Internet Protocol (TCP/IP)

Transmission Control Protocol/Internet Protocol. The default wide area network protocol that provides communication across diverse interconnected networks.

☐ Show icon in notification area when connected
☑ Notify me when this connection has limited or no connectivity

You can get IP settings assigned automatically if your network supports this capability. Otherwise, you need to ask your network administrator for the appropriate IP settings.

○ Obtain an IP address automatically
● Use the following IP address:

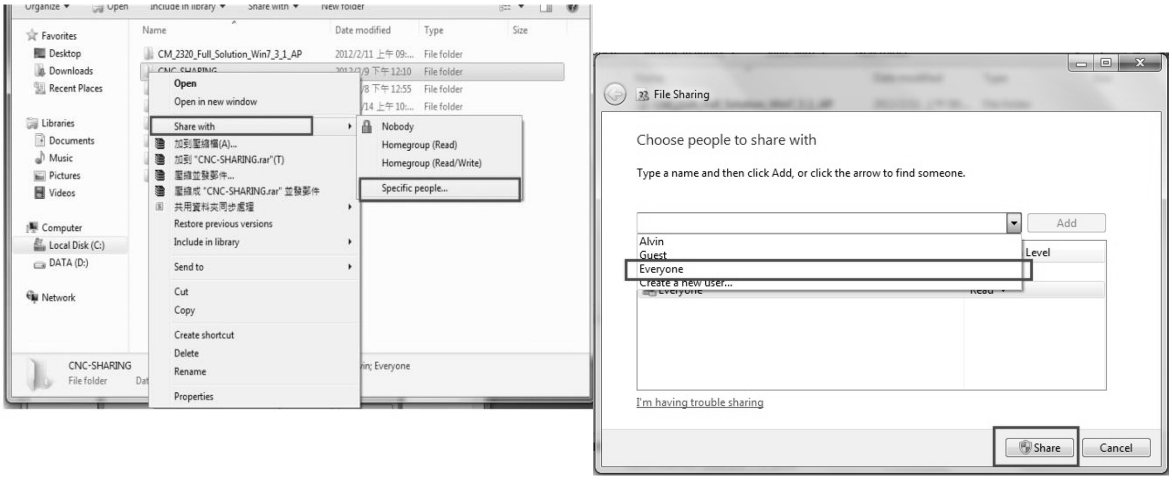
IP address: 210 . 20 . 98 . 20
Subnet mask: 255 . 255 . 255 . 0
Default gateway: . . .

○ Obtain DNS server address automatically
● Use the following DNS server addresses:

Preferred DNS server: . . .
Alternate DNS server: . . .

3.4.7.3 Win 7 OS

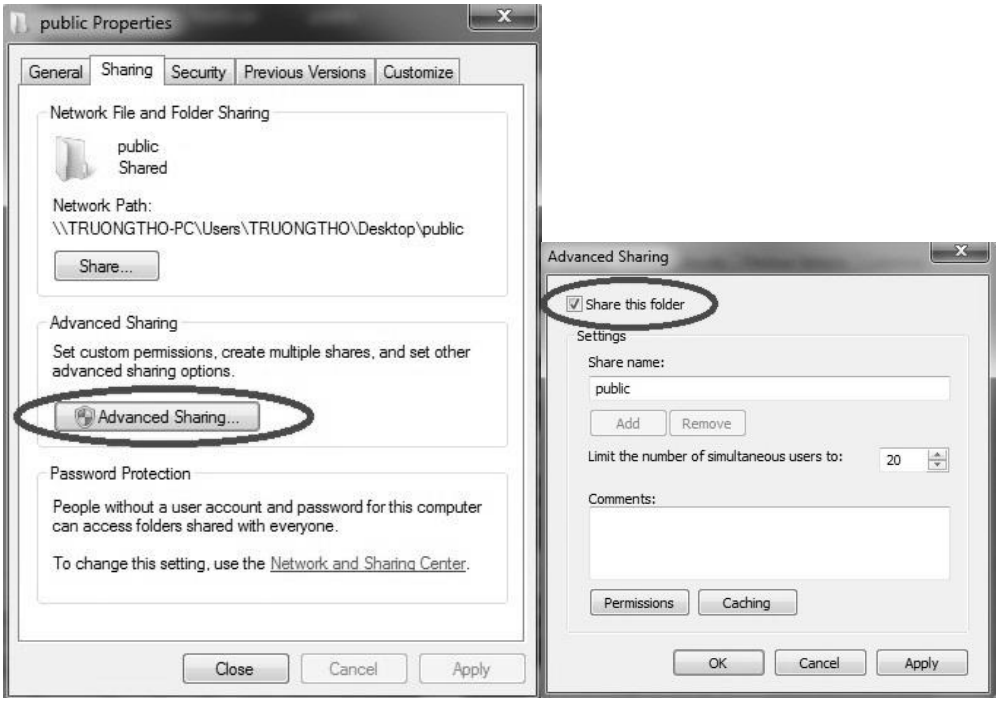
1. Sharing resource setting
   - ✧ Right-click on folder wants to share, select “share with” and “specific people”
   - ✧ Share this folder to everyone, and then click “Share” as follows.
   - ✧ Set permission as write/read


Organize
Open
Include in library
Share with
New folder
Favorites
Desktop
Downloads
Recent Places
Libraries
Documents
Music
Pictures
Videos
Computer
Local Disk (C:)
DATA (D:)
Network
Name
Date modified
Type
Size
CM_2320_Full_Solution_Win7_3_1_AP
2012/2/11 上午 09:...
File folder
CNC-SHARING
File folder
Open
Open in new window
Share with
加到壓縮檔(A)...
加到 "CNC-SHARING.rar"(T)
壓縮並發郵件...
壓縮成 "CNC-SHARING.rar" 並發郵件
共用資料夾同步處理
Restore previous versions
Include in library
Send to
Cut
Copy
Create shortcut
Delete
Rename
Properties
Nobody
Homegroup (Read)
Homegroup (Read/Write)
Specific people...
CNC-SHARING
File folder
File Sharing
Add
Alvin
Guest
Everyone
Create a new user...
Level
Read
I'm having trouble sharing
Share
Cancel

Choose people to share with

Type a name and then click Add, or click the arrow to find someone.

   - ✧ Left-click on “advanced sharing” and select “share this folder” to everyone.



public Properties
General
Sharing
Security
Previous Versions
Customize
Network File and Folder Sharing
Share...
Advanced Sharing
Advanced Sharing...
Password Protection
Close
Cancel
Apply
Advanced Sharing
Settings
Add
Remove
Comments:
Permissions
Caching
OK
Cancel
Apply

public
Shared

Network Path:
\\TRUONGTHO-PC\Users\TRUONGTHO\Desktop\public

Set custom permissions, create multiple shares, and set other advanced sharing options.

People without a user account and password for this computer can access folders shared with everyone.

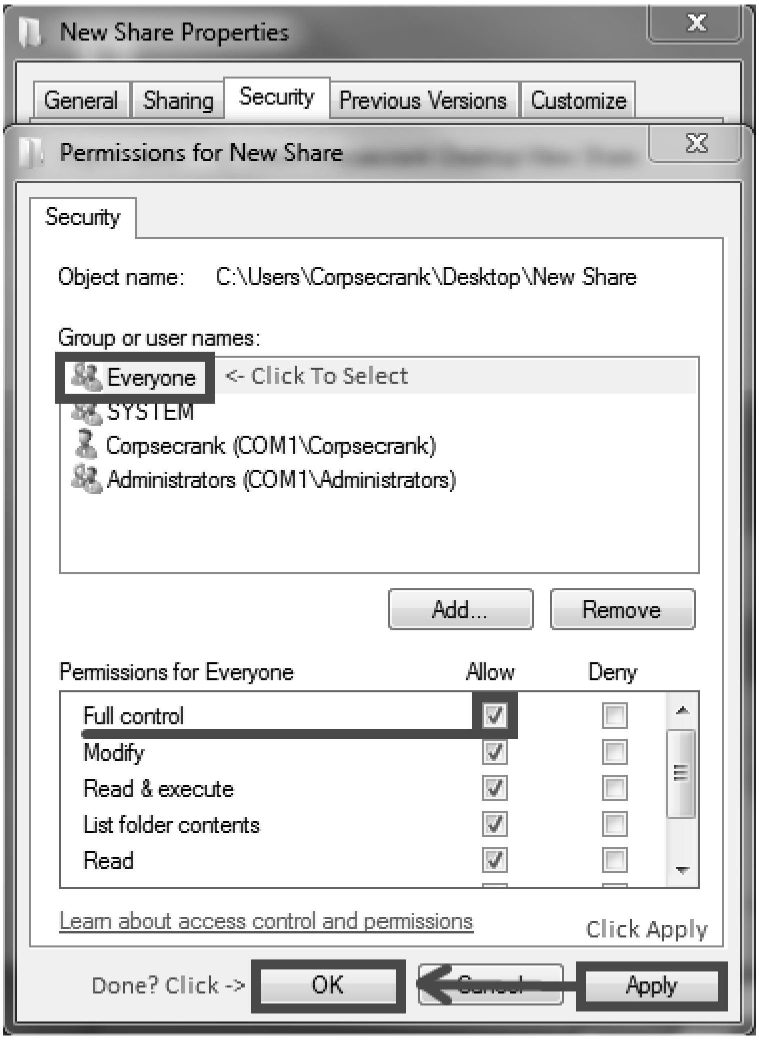
To change this setting, use the Network and Sharing Center.

☑ Share this folder

Share name: public

Limit the number of simultaneous users to: 20

✧ Select "permission" and select "full control" "only read" and "change"



New Share Properties
General
Sharing
Security
Previous Versions
Customize
Permissions for New Share
Security

Object name: C:\Users\Corpsecrank\Desktop\New Share

Group or user names:

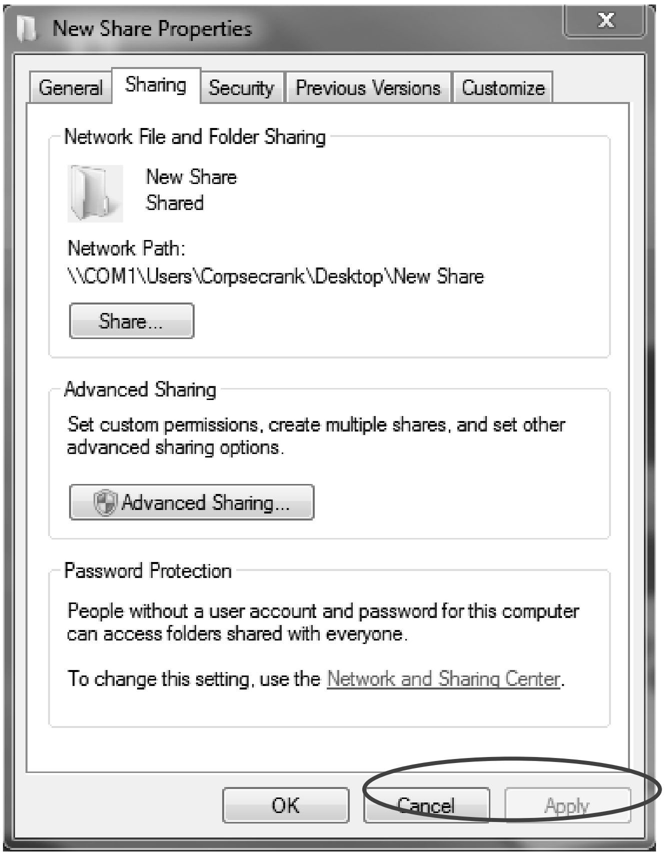
- Everyone <- Click To Select
- SYSTEM
- Corpsecrank (COM1\Corpsecrank)
- Administrators (COM1\Administrators)

Add...
Remove

Permissions for Everyone	Allow	Deny
Full control	☑	☐
Modify	☑	☐
Read & execute	☑	☐
List folder contents	☑	☐
Read	☑	☐

Learn about access control and permissions
Click Apply
Done? Click ->
OK
Cancel
Apply

- ✧ Open “Network and sharing center”, select “turn off password protected sharing” and “Open sharing…..”


New Share Properties
General
Sharing
Security
Previous Versions
Customize
Share...
Advanced Sharing...
OK
Cancel
Apply

Network File and Folder Sharing

New Share
Shared

Network Path:
\\COM1\Users\Corpsecrank\Desktop\New Share

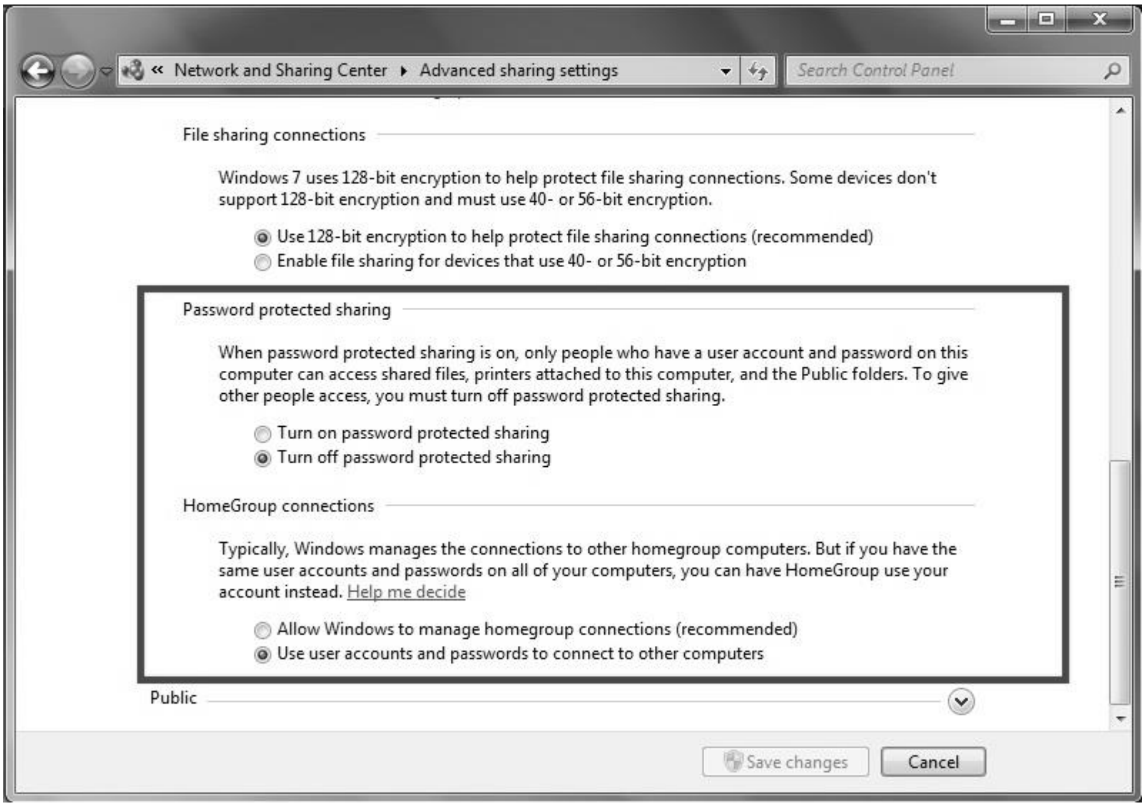
Advanced Sharing

Set custom permissions, create multiple shares, and set other advanced sharing options.

Password Protection

People without a user account and password for this computer can access folders shared with everyone.

To change this setting, use the Network and Sharing Center.


« Network and Sharing Center ▸ Advanced sharing settings
Search Control Panel
Public
Save changes
Cancel

File sharing connections

Windows 7 uses 128-bit encryption to help protect file sharing connections. Some devices don't support 128-bit encryption and must use 40- or 56-bit encryption.

● Use 128-bit encryption to help protect file sharing connections (recommended)
○ Enable file sharing for devices that use 40- or 56-bit encryption

Password protected sharing

When password protected sharing is on, only people who have a user account and password on this computer can access shared files, printers attached to this computer, and the Public folders. To give other people access, you must turn off password protected sharing.

○ Turn on password protected sharing
● Turn off password protected sharing

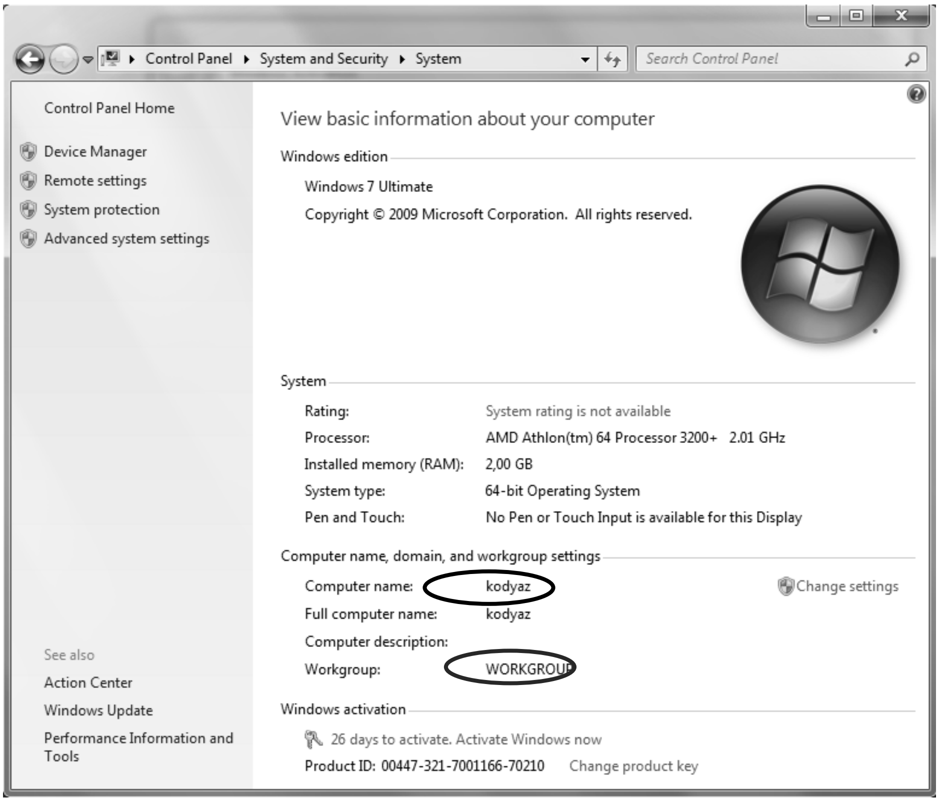
HomeGroup connections

Typically, Windows manages the connections to other homegroup computers. But if you have the same user accounts and passwords on all of your computers, you can have HomeGroup use your account instead. Help me decide

○ Allow Windows to manage homegroup connections (recommended)
● Use user accounts and passwords to connect to other computers

2. Setting PC name and workgroup

“Start”→ “control panel”→”system and security”→ ”System”→ "change " to set "Computer Name" and "Workgroup", remember these setting contents to use later on when setting controller.



Control Panel ▸ System and Security ▸ System
Search Control Panel
Control Panel Home
Device Manager
Remote settings
System protection
Advanced system settings
See also
Action Center
Windows Update
Performance Information and Tools
Change settings
Change product key

View basic information about your computer

Windows edition

Windows 7 Ultimate

Copyright © 2009 Microsoft Corporation. All rights reserved.

System

Rating: System rating is not available

Processor: AMD Athlon(tm) 64 Processor 3200+ 2.01 GHz

Installed memory (RAM): 2,00 GB

System type: 64-bit Operating System

Pen and Touch: No Pen or Touch Input is available for this Display

Computer name, domain, and workgroup settings

Computer name: kodyaz

Full computer name: kodyaz

Computer description:

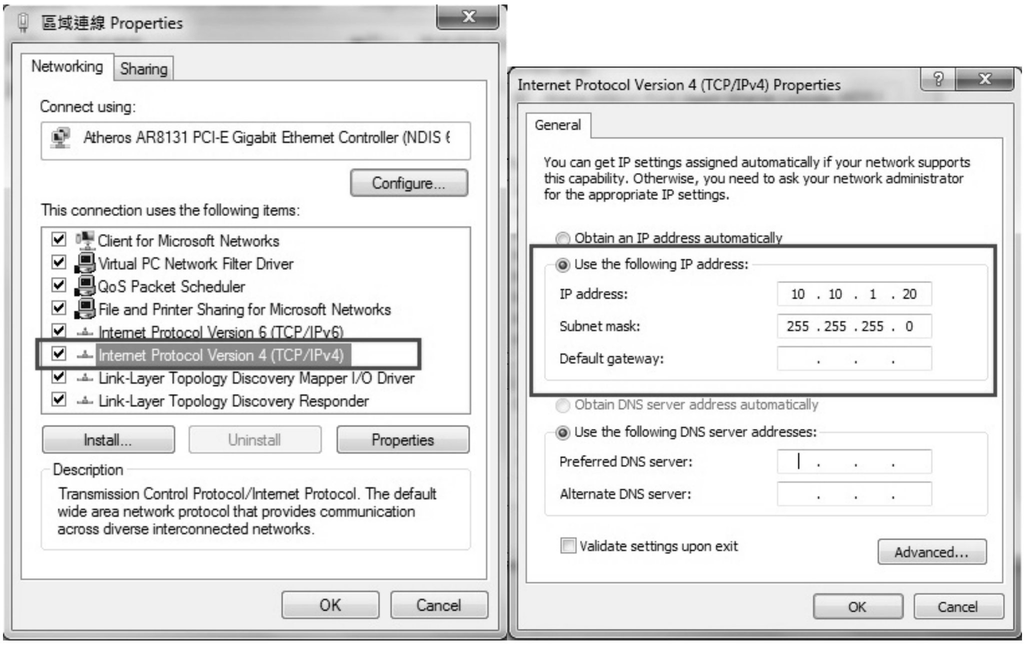
Workgroup: WORKGROUP

Windows activation

26 days to activate. Activate Windows now

Product ID: 00447-321-7001166-70210

3. TCP/IP Setting
   - Double click “Internet Protocol Version 4 (TCP/IPv4)”
   - Jumper cable(without HUB), select “use the following IP address” and enter IP address(the forth number is different from controller setting) and Subnet mask(same with controller setting)
   - Network cable(with HUB), select ”Obtain an IP address automatically”



區域連線 Properties

Networking | Sharing

Connect using:

Atheros AR8131 PCI-E Gigabit Ethernet Controller (NDIS 6

This connection uses the following items:

☑ Client for Microsoft Networks
☑ Virtual PC Network Filter Driver
☑ QoS Packet Scheduler
☑ File and Printer Sharing for Microsoft Networks
☑ Internet Protocol Version 6 (TCP/IPv6)
☑ Internet Protocol Version 4 (TCP/IPv4)
☑ Link-Layer Topology Discovery Mapper I/O Driver
☑ Link-Layer Topology Discovery Responder

Description

Transmission Control Protocol/Internet Protocol. The default wide area network protocol that provides communication across diverse interconnected networks.

Internet Protocol Version 4 (TCP/IPv4) Properties

General

You can get IP settings assigned automatically if your network supports this capability. Otherwise, you need to ask your network administrator for the appropriate IP settings.

○ Obtain an IP address automatically
● Use the following IP address:

IP address: 10 . 10 . 1 . 20
Subnet mask: 255 . 255 . 255 . 0
Default gateway: . . .

○ Obtain DNS server address automatically
● Use the following DNS server addresses:

Preferred DNS server: . . .
Alternate DNS server: . . .

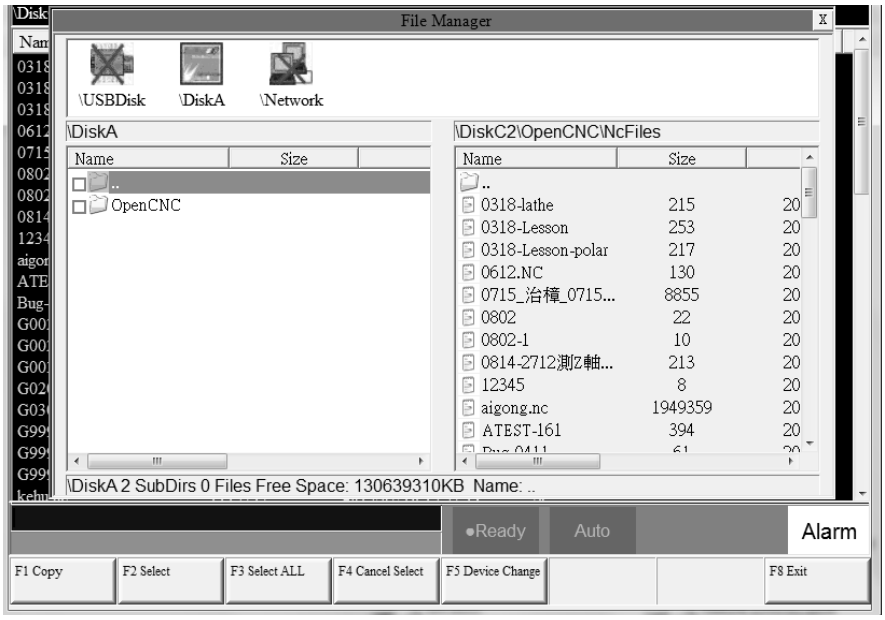
☐ Validate settings upon exit

3.5 File Transfer

This section will show how to transfer files, files transfer is divided into import and export files, allowing controller share files to external devices, such as USB, CF card or users on the network.

3.5.1 File Import

- Operation
  - ✧ Path: F2-program→ F8-file manager → F4 File import
  - ✧ Other interface will appear on screen, press F5-“device change” to move cursor to desired external device on the status bar
  - ✧ Press [Enter] key to access to inside device, if the device icon has red cross, which means that there is no connection to this device,
  - ✧ Select the file wants to import and press [Copy] to complete import file
  - ✧ Press F4 cancel select to cancel the selected file
  - ✧ After complete file transfer, press [Left] or [ESC] to leave this screen


File Manager
X
\USBDisk
\DiskA
\Network
F1 Copy
F2 Select
F3 Select ALL
F4 Cancel Select
F5 Device Change
F8 Exit
•Ready
Auto
Alarm

\DiskA

Name	Size
☐ ..	
☐ OpenCNC	

\DiskC2\OpenCNC\NcFiles

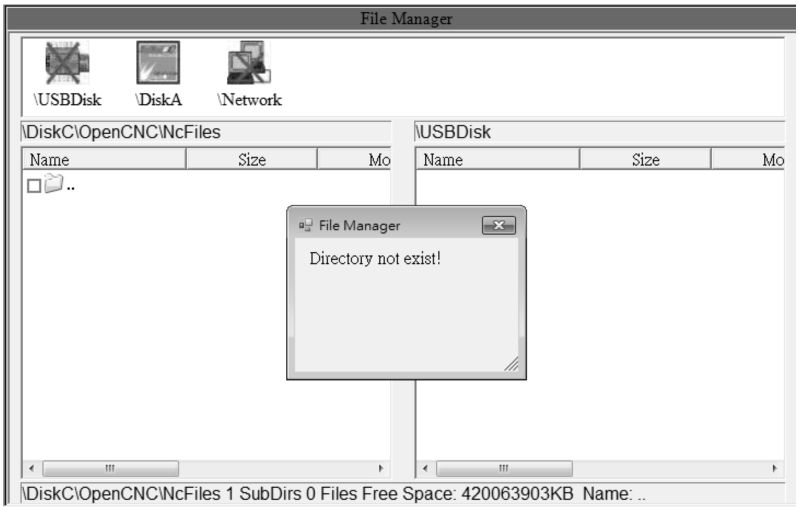
Name	Size	
..		
0318-lathe	215	20
0318-Lesson	253	20
0318-Lesson-polar	217	20
0612.NC	130	20
0715_治樟_0715...	8855	20
0802	22	20
0802-1	10	20
0814-2712測Z軸...	213	20
12345	8	20
aigong.nc	1949359	20
ATEST-161	394	20

\DiskA 2 SubDirs 0 Files Free Space: 130639310KB Name: ..

3.5.2 File Export

- Operation
  - ✧ Path: F2-program→ F8-file manager → F5 File export
  - ✧ Other interface will appear on screen, press F5-device change to move cursor to desired external device on the status bar
  - ✧ Press [Enter] key to access to inside device, if the device icon has red cross, which means that there is no connection to this device,

    Select the file wants to export and press [Copy] to complete export file
  - ✧ Press F4 cancel select to cancel the selected file
  - ✧ After complete file transfer, press [left] or [ESC] to leave this screen
- Note:
  - ✧ if destination of export file does not exist, below alarm will appear


File Manager
\USBDisk
\DiskA
\Network
\DiskC\OpenCNC\NcFiles
Name
Size
Mo
..
\USBDisk
Name
Size
Mo
File Manager

Directory not exist!

\DiskC\OpenCNC\NcFiles 1 SubDirs 0 Files Free Space: 420063903KB Name: ..

4 Appendix

4.1 Release Note

Doc. Ver.	Content	Release Date	Author	Reviwer	CNC Ver.
1.0	1st Version	2013/08/15	Tsai Chiju	Sandy Duan	1.0.6
1.1	Modify the content	2013/10/30	Sandy Duan	Yuliu.Duma	1.0.6
1.2	Add description of Set Tool Mach./ Rel. Coord. for tool length	2014/01/14	HungAn Chang	HungAn Chang	2.2.0
1.3	Modify [WorkPiece Cord] key	2014/03/02	Andy Ngo	HungAn Chang	2.2.2
1.4	Add Chinese topic, and increase front size	2015/11/13	Linda Chen	Yu-An Chiang	

4.2 Contact Window

Hsinchu Headquarters Tel：+886-3-6663553 Fax：+886-3-6663505 E-Mail：syntec@syntecclub.com.tw Address： No.21, Industry E.Rd. 4, Hsinchu Science Park 30077,Taiwan,R.O.C	Jiangsu-Suzhou SYNTEC Equipment CO.,Ltd. TEL：+86-512-69008860 FAX：0512-69560818 Address： D 1-8,Teng Fei New Industrial Zone, 5# Xing Han Street, Suzhou Industrial Park, China
Taichung Taichung Service Agency TEL：+886-4-25337731 FAX：+886-4-25349224 Address： No.31, Alley 9, Lane 271, Shepi Rd., Fengyuan City, Taichung County 420, Taiwan, R.O.C	Taichung Taichung Service Agency TEL：+886-4-23102626 FAX：+886-4-23102636 Address： No.42, Jingming St., West Dist., Taichung City 403, Taiwan

Tainan Tainan Service Agency TEL：+886-6-2796707 FAX：+886-6-2796705 Address： No.218, Denan Rd., Rende Dist., Tainan City 71756, Taiwan	Jiangsu-SuZhou Suzhou Huaxin Numerical Control Technology Development CO.,LTD TEL：0512-69560828 FAX：0512-69560818 Address： Xing Han Street, Suzhou Industrial Park, Jiangsu Province, to take off the new Su Industrial Square D, 2nd Floor 01-08 unit
Zhejiang-Hangzhou SuZhou SYNTEC Equipment CO.,Ltd. Hangzhou Branch TEL：+86-571-82751187 FAX：+-86-571-82751186 Address： Room1202 Unit2 Buiding1, Edifice DiKai , Jincheng Road, Beigan Street,Xiaoshan District , Hangzhou City,Zhejiang Province, China	Zhejiang-Ningbo SuZhou SYNTEC Equipment CO.,Ltd. Ningbo Branch Ningbo City. TEL：+86-574-87750305 FAX：+86-574-87750306 Address： Room 12-07,No.262,416 Alley,Zhaohui Road,Jiangdong District,
Zhejiang-Wenling The Suzhou new generation of CNC Equipment Co., Ltd. Wenling Branch TEL：0576- 86138372 FAX：0576-86119106 Address： Room 1206, Department A, Zhenxing Plaza, Taiping Subdistrict, Wenling City	Guangdong-Guangzhou SuZhou SYNTEC Equipment CO.,Ltd. Guangzhou Branch TEL：+86-20-34583040 FAX：+86-20-34583220 Address： Room 403 Stairs1 Youyi Building!ALuojia Village 8,Fuyi Road,ShijiTown,Panyu Distrct,Guangzhou City,Guangdong Province, China
Guangdong-Dongguan SuZhou SYNTEC Equipment CO.,Ltd. Dongguan Branch TEL：+86-769-81660318 FAX：+86-769-81660328 Address： Room 705 JinGuo Business affairs centre Xin an District	Shaanxi-Xi´an The Suzhou new generation of CNC Equipment Co., Ltd. Xi'an Office TEL：029-88287423 FAX：029-88287423 Address： Room 2503,Unit 1,Rancho Santa

Chang an	Fe,No.36, Dianzi Third Road, Yanta District, Xi ˊ an, Shaanxi
Shaanxi-Baoji The Suzhou new generation CNC Equipment Co., Ltd. Baoji offices TEL：18700712118 Address： Room 2401, Building 1, Xinjian Road, Weibin District, Baoji	Shandong-Jinan SuZhou SYNTEC Equipment CO.,Ltd. Jinan Branch TEL：+86-53185907208 FAX：+86-53185905708 Address： Room 1112, Unit A, Jiahui Global Plaza, No.548, Beiyuan Street, Tianqiao District, Jinan, China
Fujian-Xiamen SuZhou SYNTEC Equipment CO.,Ltd. Xiamen Branch TEL：+86-592-7191901 FAX：+86-592-7220536 Address： No. 100,B Chuang Chang Fang 3F East,Jin Fu Road,Tong An District,Xiamen City, Fujian Province, China	Tianjin The Suzhou new generation of CNC Equipment Co., Ltd. Tianjin Branch TEL：+86-22-87134111 FAX：022-87134111 Address： Tianjin Huayuan Industrial Zone, Rong Yuan Road No. 4 day science and Technology Park Building No. 1, 2 door 501
Henan-Luoyang The Suzhou new generation of CNC Equipment Co., Ltd. Luoyang Company TEL：0379-65110352 FAX：0379-65110352 Address： 601B, Runsheng Building, Cross of Sanshan Road & Heluo Road, Hi-and-New Tech Park of Luoyang, Henan	Chongqing The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch TEL：023-67913296 FAX：023-67634382 Address： Room 41-4, Building A, Hongding International, Guanyinqiao, Jiangbei District, Chongqing
Jiangsu-Nanjing The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch TEL：0512-69008860-300 Address： Room 505, Unit 3, Building 12,Wuyi Luzhou Guan Chu	Hubei-Wuhan Suzhou new generation of CNC Equipment Co., Ltd. Wuhan Office TEL：027-87638876 FAX：027-87204137 Address： Room2003-2006,A#Guangguguoji, 456#Luoyu

Yuan, No.99, Tianyuan Middle Road, Jiangning District, Nanjing	Road,Donghukaifaqu,Wuhan City, China
Shenyang The Suzhou new generation CNC Equipment Co., Ltd. Shenyang Office TEL：024-25821398 FAX：024-25821398	Anhui-Hefei The new generation of CNC Equipment Co., Ltd. in Suzhou, Hefei Office TEL：15951989576 Address： Room 704, Building 13,Wanzhen Xiaoyao Garden Ⅳ, Taihu East Road, Hefei
Shenzhen Shenzhen branch District, Sham Chun City, Guandon Province TEL：0755-84584085 Address： Room 706, Hsin Toun Building, Toun Chung Road, Ai Lian Hsin Toun Village, Loggan	Thailand SIAM RADERMEN Co., Ltd. TEL：+66-27553536 FAX：+66-27575476 E-Mail： lee_cheahow@yahoo.com Address： 52/4,MOO5,THEPARAK RD.(KM11.5)BANGPLEEYAI,BANGPLEE, SAMUTPRAKRAN 10540
Malaysia SURE FIRST ENTERPRISE (M) SDN BHD. TEL：+603-61577632 FAX：+603-61574632 Address： 47000 NO 19.JALAN TSB 6,TAMAN INDUSTRI SUNGAI BULOH.47000 SELANGOR DARUL.EHSAN	Lebanon / Damascus Kesmeyan Group Co. Lebanon Beirut, Jdeideh - Ammaieh - Kessmeyan Group Build. Damascus TEL: +963 11 6731602 Mob.: +963 988 211407 E-Mail : info@rockwell.tw TEL：+961-76-883778 FAX：+961-1-901807 E-Mail： kesmeyan@rockwell.tw
France INTEGRATION CNC TEL：+33 2 35 06 07 83 FAX：+33 2 35 06 07 83	Turkey KASIKCIOGLU ELEKTROMEKANIK OTOMASYON

http://www.integrationcnc.fr/ E-Mail： vaque.j@online.fr Address： 6, Rue l'arché de Copigny, 76630 BAILLY, FRANCE	TEL：+90-224-4434684 FAX：+90-224-4434685 E-Mail： murat@kasikcioglu.com Address： Besevler K.S.S. 17 BLOK NO:72 Nilufer/BURSA/TURKYIE
ISTANBUL/TURKEY AKSIS MAKINA TEL：+90 212 613 87 83 FAX：+90 212 613 87 85 http://www.aksismakina.com E-Mail： info@aksismakina.com Address： Yeni doğan mahallesi karakaş Sk. Emintaş Erciyes Sanayi Sitesi no:12/206	