SYNTEC

6 Series Mill Controller

Operation Manual

Date: 2014/03/02

Version: 1.2

Contents

![SYNTEC]

1 Function Key and System Configuration

1.1 Main Screen Sections
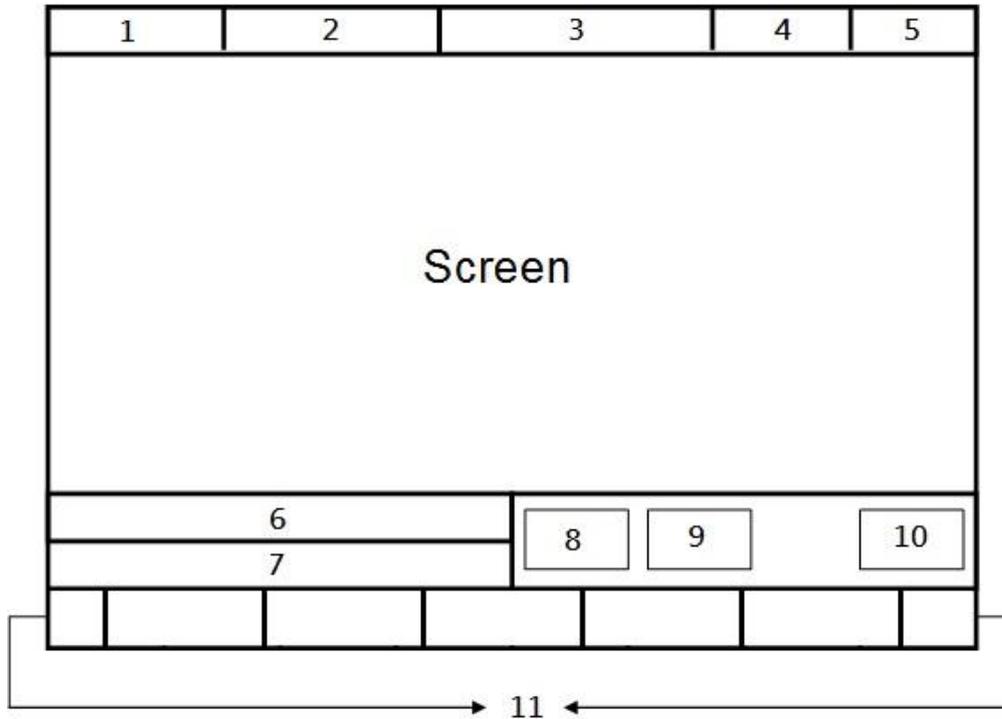
◆ Meanings for Sections on the Display:

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

SYNTEC

1.3 Coordinate

- 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
  - ✧ When 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.
  - ◇ Press 「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
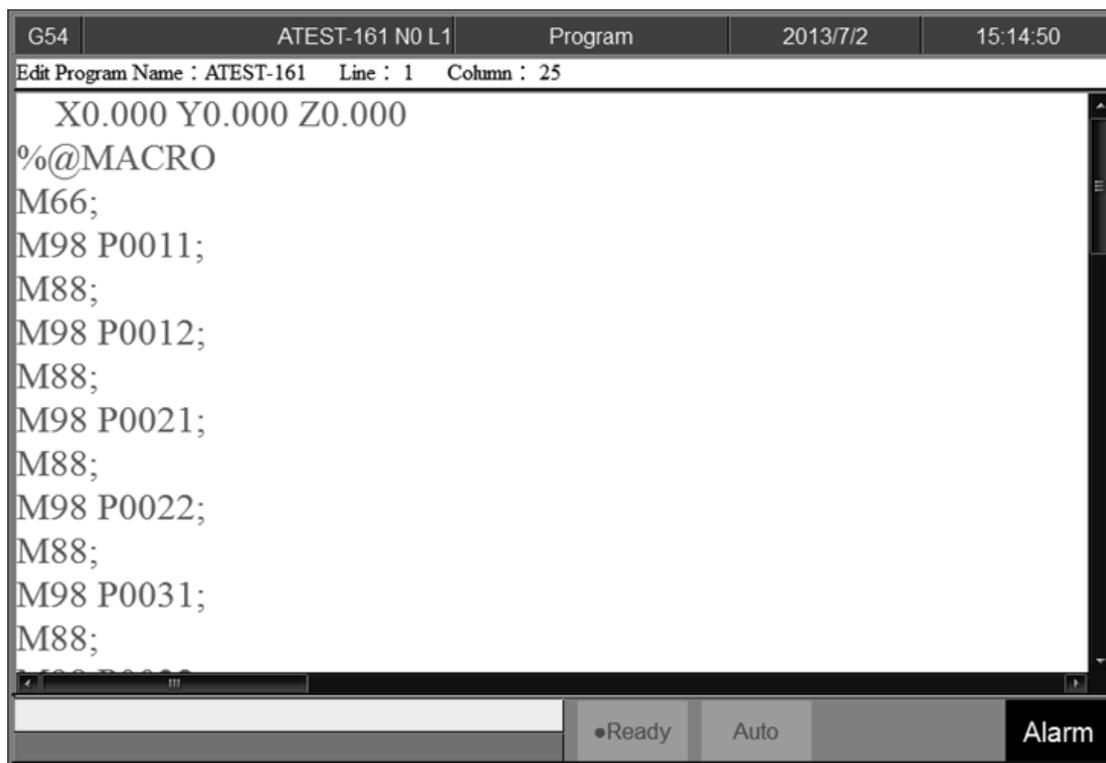
- Command
  - ◇ F2 Program
- Function
  - ◇ This function provides users program management and editing

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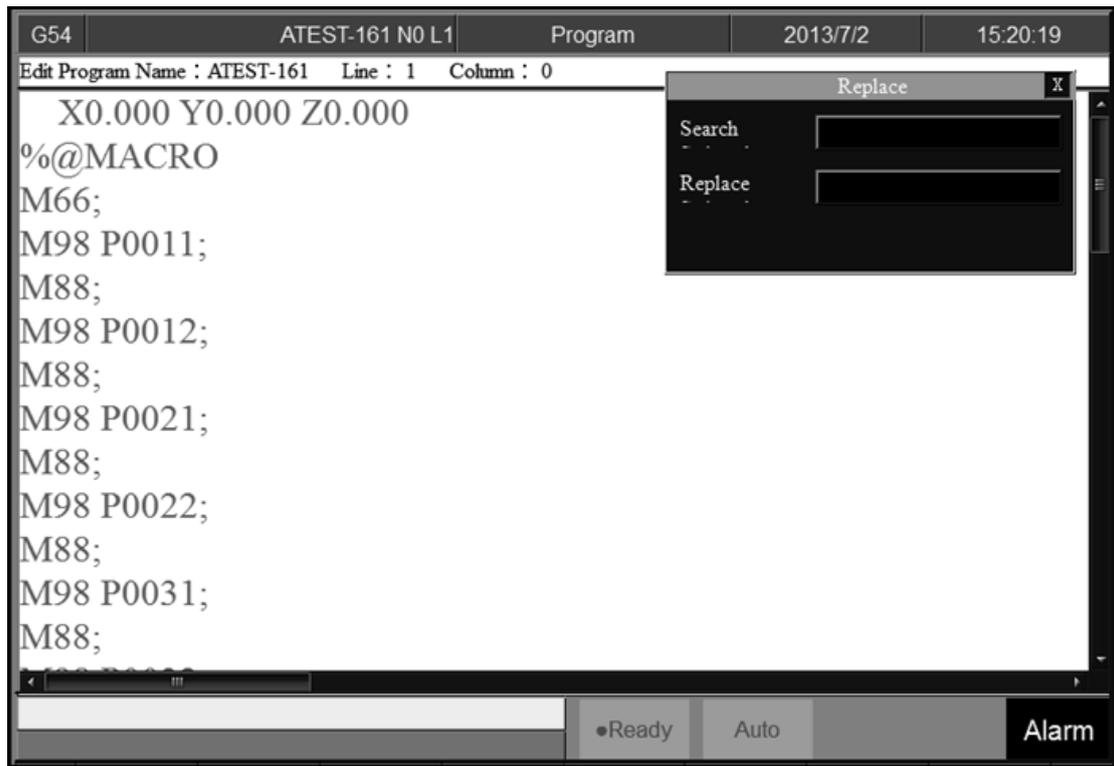
- Command
  - ◇ F2 Program→F3 Search/Replace
- Function
  - ◇ Quick search for every occurrence of a specific word or phrase and automatically replace text.
- Operation Method
  - ◇ Press 「Search/Replace」function key, then Replace box will appear, just enter the text that you want to search and replace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

● Operation Method

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

1.4.4 File Manager

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→F4 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.

1.4.4.1　　New File

● Command

　　◇ F2 Program→F4 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)

1.4.4.2　　Copy File

● Command

　　◇ F2 Program→F4 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)

1.4.4.3   Delete File

- Command

  ✧ F2 Program→F4 File Manager→F3 Delete File

- Function

  ✧ Delete file that remarked by cursor.

- Operation Method

  ✧ Press F3「Delete File」, checkbox 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 files.

  ✧ Cancel Select: Deselect all files.

  ✧ Delete File: Delete all of the selected files.

  ✧ Delete All: Delete all file within the data storage device.

- Note

  ✧ Current Programming and machining file cannot be deleted.

1.4.4.4　　　　File Transfer

● Command

　　◇ F2 Program→F4 File Manager→F4 File Transfer

● Function

　　◇ Transmit data between controller and outer device.

1.4.4.4.1 **File Import**

- Command
  - ✧ F2 Program→F4 File Manager→F4 File Transfer→F1 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 USBDisk.
  - ✧ 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 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.

1.4.4.4.2 File Export

- Command
  - ✧ F2 Program→F4 File Manager→F4 File Transfer→F2 File Export
- Function
  - ✧ Export files 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 left column will change and show the data structure of

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.

1.4.4.5   Execute

● Command

✧ F2 Program→F4 File Manager→F5 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.

1.4.5 Simulation
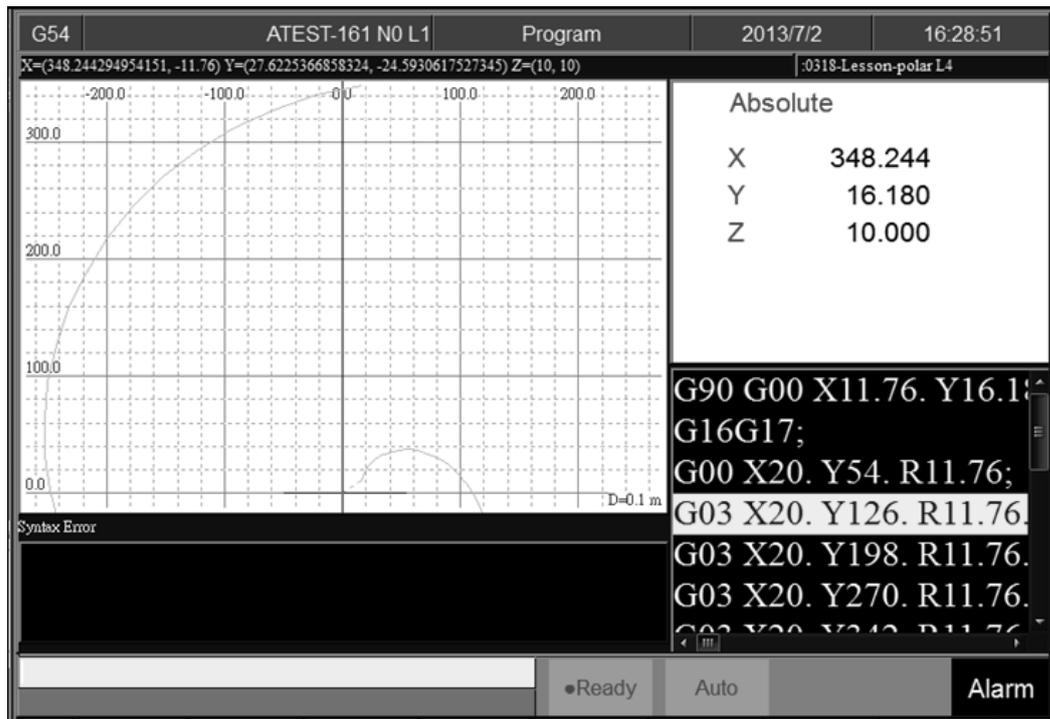
- Command

  ◇ F2 Program→F5 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」.

1.4.5.1 Step

● Command

✧ F2 Program→F5 Simulation→F1 Step

● Function

✧ Simulate the program block by block.

✧ Monitor the variation of the coordinate for single block.

1.4.5.2 Continue

● Command

✧ F2 Program→F5 Simulation→F2 Continue

● Function

✧ System will scan all of the programs and then do the simulation.

1.4.5.3 Zoom

● Command

✧ F2 Program→F5 Simulation→F3 Zoom

● Function

✧ Zoom in/out the simulation window.

● Operation Method

✧ Press F3「Zoom」,there will be a block show up. Use【↑】【↓】【←】
【→】to move the window up, down, left and right.

✧ Use【Page Up】【Page Down】to change the zooming area.

✧ Press【ENTER】to check the result.

1.4.5.4　　Graph Reset

● Command

 ✧ F2 Program→F5 Simulation→F4 Graph Reset

● Function

 ✧ Reset the simulation result to default.

1.4.5.5　　Simu. Setting
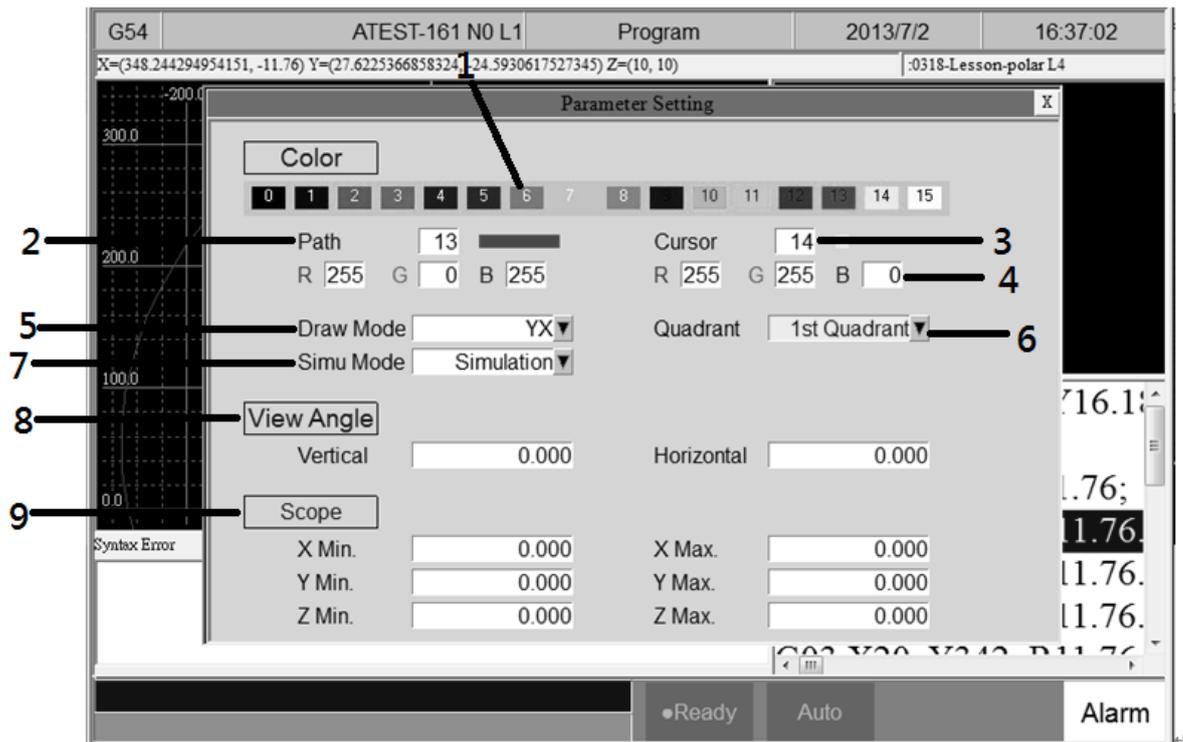
● Command

 ✧ F2 Program→F5 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 themself.

    5. Draw Mode

◇ User can define the profile simulate plane.

- Plane can define are as below.
  - ◆ XYZ
  - ◆ XY
  - ◆ YZ
  - ◆ ZX
  - ◆ YX
  - ◆ ZY
  - ◆ XZ
  - ◆

    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.

- 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

1.4.6 Can Cycle
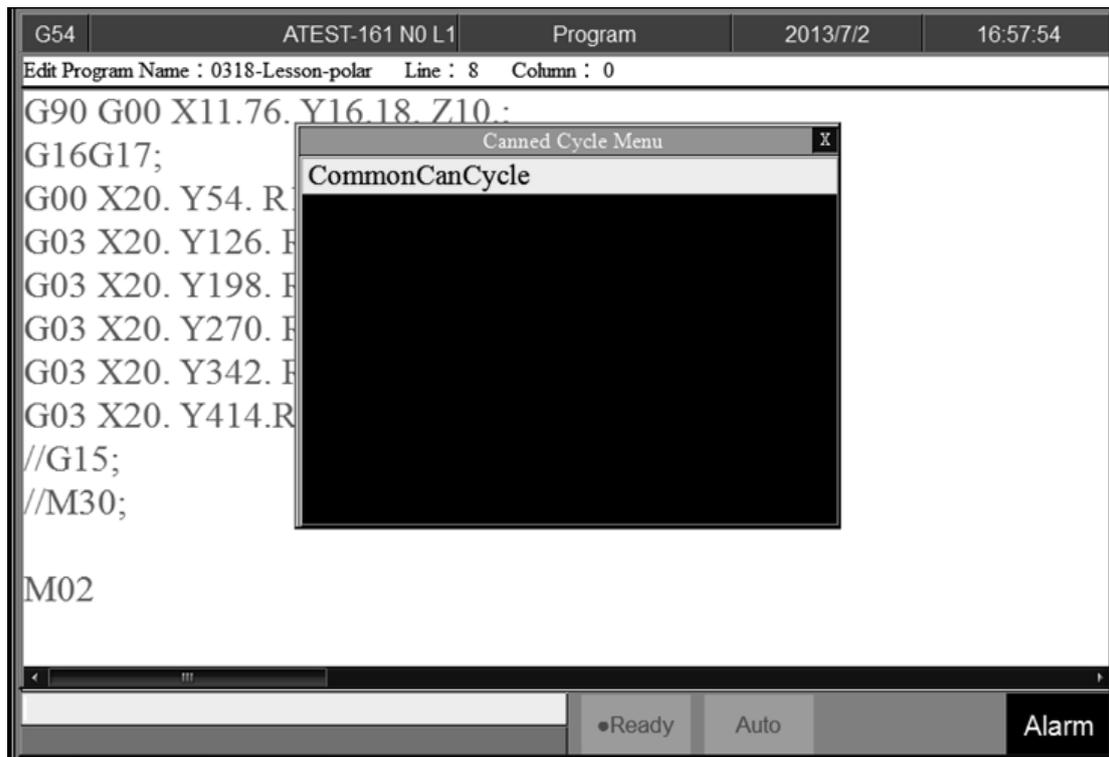
- Command
  - ✧ F2 Program→Next→F1 Can Cycle
- Function
  - ✧ Because Syntec system provides many G codes and different G code has each function. When editing the program, this function can help user to edit G code easily.

1.4.6.1　　Insert Cycle

● Command

　　✧ F2 Program→Next→F1 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.

1.4.6.2　　Edit Cycle

● Command

　　✧ F2 Program→Next→F1 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.

1.4.7 Block Copy
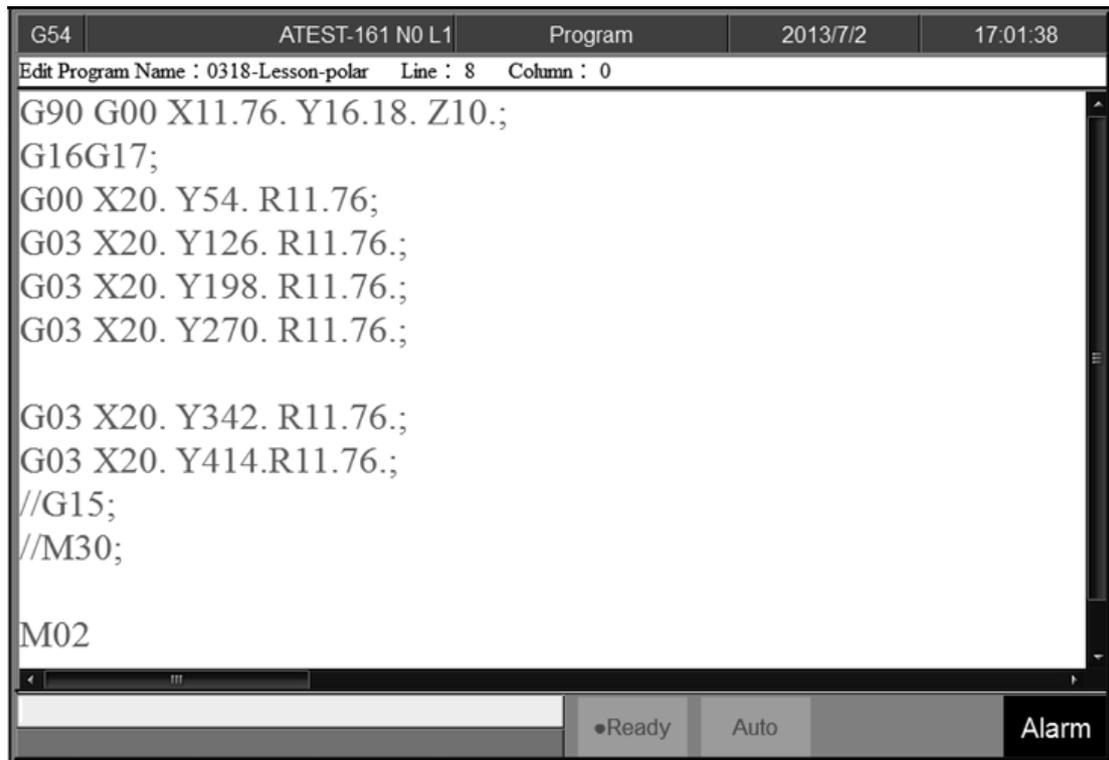
G54	ATEST-161 N0 L1	Program	2013/7/2	17:01:38

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
```

●Ready   Auto   Alarm

- Command

    ✧ F2 Program→Next→F2 Block Copy

- Function

    ✧ Select, cut, copy and paste more than one line of programs.

1.4.7.1　　　Start Line

● Command

　　✧ F2 Program→Next→F2 Block Copy→F1 Start Line

● Function

　　✧ Define the start line of block.

1.4.7.2　　　End Line

● Command

　　✧ F2 Program→Next→F2 Block Copy→F2 End Line

● Function

　　✧ Define the end line of block.

1.4.7.3　　　Block Cut

● Command

　　✧ F2 Program→Next→F2 Block Copy→F3 Block Cut

● Function

　　✧ Cut the block that had been selected.

1.4.7.4　　　Block Copy

● Command

　　✧ F2 Program→Next→F2 Block Copy→F4 Block Copy

● Function

　　✧ Copy the block that had been selected.

1.4.7.5　　　Block Paste

● Command

　　✧ F2 Program→Next→F2 Block Copy→F5 Block Paste

● Function

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

1.4.7.6　　　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 copied 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.

1.4.8 Teach
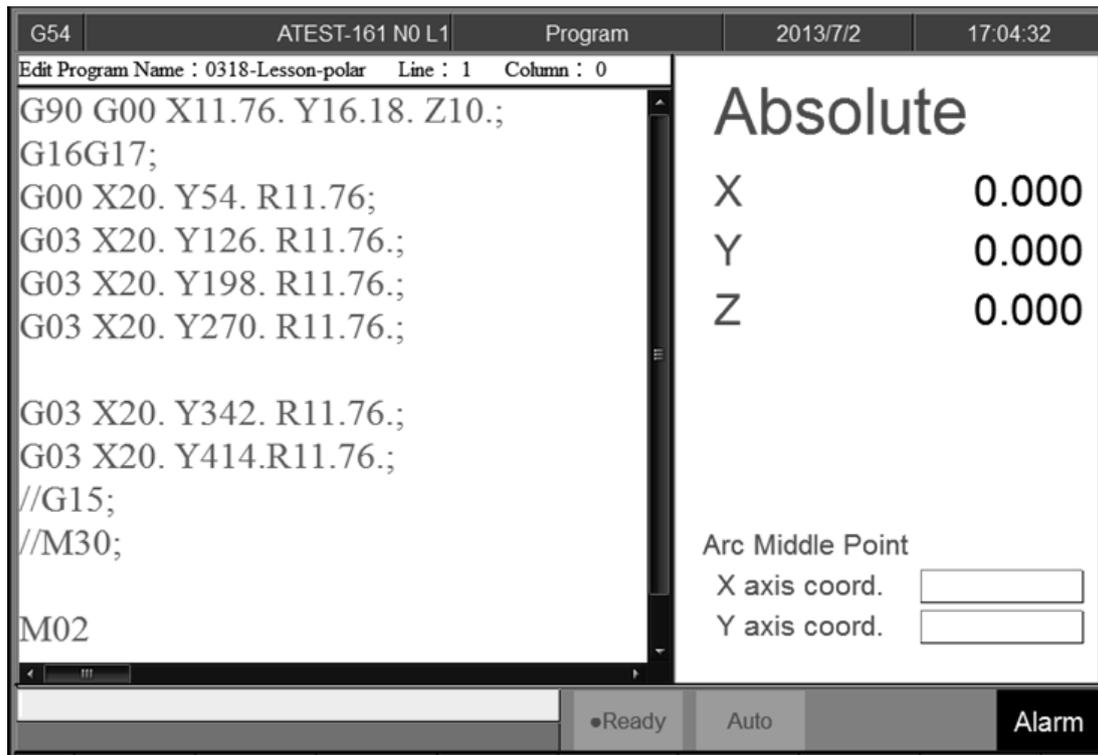
- Command

  ◇ 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.

1.4.8.1 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.

1.4.8.2 Line Cut Teach

- Command

  ◇ F2 Program→F6 Teach→F2 Line Cut Teach

● Function

   ✧ Add the current absolute coordinate as the input value of 「G01 Linear

   Cutting」 function in current program.

1.4.8.3   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.

1.4.8.4   Cancel Arc Middle

● Command

   ✧ F2 Program→Next→F3 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.

1.4.8.5 Point Teach

● Command

✧ F2 Program→Next→F3 Teach→F5 Point Teach

● Function

✧ Move the worktable to the arc center and press 「Point Teach」, current

absolute coordinate will be input into the cursor location.

1.5 Offset/Setting

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

External Shift		G54P1(G54)		G54P2(G55)		Machine	
X	0.000	X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000	Z	0.000
A	0.000	A	0.000	A	0.000	A	4.158

Relative

X	0.000
Y	0.000
Z	0.000
A	4.158

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

	●Ready	Auto		Alarm

- 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.

G54			N0 L1	Offset/Setting	2013/8/22	17:12:48

External Shift		G54P1(G54)		G54P2(G55)		Machine	
X	0.000	X	0.000	X	0.000	X	0.000
Y	0.000	Y	0.000	Y	0.000	Y	0.000
Z	0.000	Z	0.000	Z	0.000	Z	0.000
A	0.000	A	0.000	A	0.000	A	4.158

Relative
X 0.000
Y 0.000
Z 0.000
A 4.158

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

●Ready | Auto | Alarm

- 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 related 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 value of X axis of G54 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 desired 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

SYNTEC

1.5.1.5 Middle Func.

- Command
  - ♦ F3 Offset/Setting→F1 Work Piece Cord.→F5 Middle Func.
- Function
  - ♦ Calibrate the middle point of the work piece before machining.
- Operation method
  - ♦ Input the Middle Func calculated result into G54~G59 work piece coordinate.
  - ♦ There are two kind of operation method for the Middle Func, one is manual and another is automatic.

1.5.1.5.1 Manual

- Function
  - ◇ Mode of Middle Func. is 0.
  - ◇ Move the tool with MPG to the edge of the workpiece XY axis. Record the machine coordinate and system will calculate the center point of the workpiece automatically.

- Operation method
  - ◇ Move the measurement tool with MPG to Px1 point according to the figure and then press PX1 set. The system will input the current machine Coord. into PX1.
  - ◇ Measure PX2 with the same method. Combine with PX2, system will compute the center point of PX1 and PX2 and set the result into Pxm and Aux. X Coord.
  - ◇ Move the measurement tool with MPG to PY1 point according to the figure and then press PY1 set. The system will input the current machine Coord. into PY1.
  - ◇ Measure PY2 with the same method. Combine with PX2, system will compute the middle point of PY1 and PY2 and set the result into Pym and Aux. Y Coord.
  - ◇ The values of Pxm and Pym are the center point of the workpiece.
  - ◇ In the WorkPiece Cord setting page, move the cursor to the relative coord, press F3 「Aux. Coord. Teach」 then system will set the Aux Coord value into relate work piece coordinate.

1.5.1.5.2 Auto

● Function

  ✧ Mode of Middle Func. is 1.

  ✧ Auto middle func. is different from Manual func. User only needs to enter the dimension of the workpiece and enter the boundary coordinate. Move the tool to the start point, system will find out the center point of work piece automatically.

● Parameter Description

  ✧ Length I: length of Workpiece in X dimension

  ✧ Width J: width of Workpiece in Y dimension

  ✧ Safe Distance H: The distance between start point P2 and workpiece, in X or Y direction.

  ✧ Feedrate F: Auto detection speed.

  ✧ Safe altitude of Z Coordinate: safety distance for tool when moving as the P1 on the figure.

● Operation Method

  ✧ Move the tool to the safe altitude, as P1 point on the figure, press 「Z Coordinate Set」, then system will record the present Z coordinate value as the safe distance.

  ✧ Move the tool down to P2 point, that point will be the starting point of 「Auto Center」.

  ✧ Press 「Auto Center Start」, system will move the tool to touch the work piece according to the setting data and show the coordinate on the screen. It will also calculate the X Y center point of workpiece.

  ✧ Go back to WorkPiece Cord. setting page, move the cursor to the workpiece coordinate, press F3 「Aux. Coord. Teach」then the system will set the coordinate with relate Aux.Coord value.

1.5.1.6 Mach. Coord Inc. Teach

- Command

  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→Next→F1 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

						Machine	
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

●Ready　Auto　Alarm

- 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 coord

            relative coord value.

- 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→F4 Tool No.

● Function

✧ Refer the chapter 3.3.3.5

1.5.2.3　　　Tool Life Manag.

● Command

✧ F3 Offset/Setting→F2 Tool Set→F5 Tool Life Manag.

● Function

✧ Refer the chapter 3.3.3.6.

1.5.3　Tool Tip Measurement

● Command

✧ F3 Offset/Setting→F3 Tool Tip Measurement

● Function

✧ User can key in different tool No. tip position into workpiece

coordinate for the tool length offset setting, with tool alignment

equipment.

✧ Detail Description please 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

- Command:
  - ✦ F4 Monitor
- Function
  - ✦ This page monitors key machining information during machining

    process.

1.6.1 Monitor Area of Machining Information
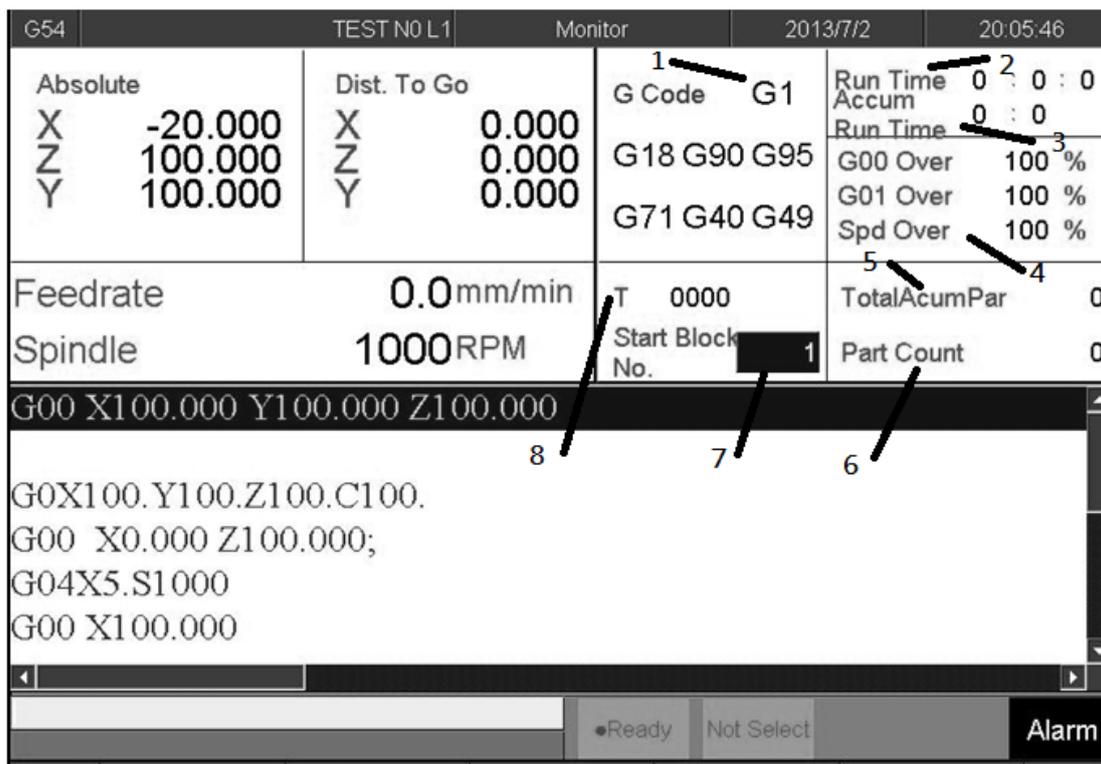
1.6.1.1 Machine Status Monitor

- 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
   - Time accumulated for current single workpiece
3. Run Time
   - Total accumulated machining 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.

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

- Feedrate
  - ◇ 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.

$\diamond$ 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

$\diamond$ Set the upper limit of part count number.

$\diamond$ 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

$\diamond$ Display the tool trajectory of current program.

$\diamond$ Related setting, please see F2-program→ F5-simulaiton→ F5- Simu.

Setting.

$\diamond$ Use F2「Simulation Switch」 to change the display content

1.6.2　Open File to Edit

● Command:

$\diamond$ F4 Monitor→F1 Open File to Edit

● Description

$\diamond$ Load and edit the current machining program, also switch to

"F2-program" interface.

● Note:

$\diamond$ Once system is on running state, edit function will be disabled.

1.6.3　Simulation Switch

● Command:

$\diamond$ F4 Monitor→F2 Simulation Switch

● Description

$\diamond$ Display or hide simulation display

$\diamond$ 「Graph Adjust」will enable under 「Simulation Switch」 conditions.

SYNTEC

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

● Description

✧ 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

- Command:
  - ◇ F4 Monitor→Next→ F1 Start MPG Coordinate
- Description
  - ◇ Detail descriptions please refer to 3.4.3.3.

1.6.8 Work Record

- Command:

  ◇ F4 Monitor→ Next→F2 Work Record

- Description

  ◇ Check current machining record and export to external storage device.

1.6.9 Clear Acum Cycle Time

- Command:

  ◇ F4 Monitor→ Next→F3 Clear Acum Cycle Time

- Description

  ◇ Clear the accumulative time

1.6.10 Graph Adjust

- Command:

  ◇ F4 Monitor→ Next→F4 Graph Adjust

- Description

  ◇ Zoom in/out simulation graph, this function will enable under

  「Simulation Switch」is open.

- Operation

  ◇ Please refer to 1.4.5 「simulation」.

1.7 Maintain

- Command:

  ✧ F5 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:
  - ✧ F5 Maintain→F1 Alarm
- Description
  - ✧ Display alarm messages on the screen.

1.7.1.1　　Pending alarm

● Command:

♦ F5 Maintain→F1 Alarm→ F1 Pending Alarm

● Description

♦ Display current system alarm.

1.7.1.2　　History alarm

● Command:

♦ F5 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:

♦ F5 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

- Command:
  - ✧ F5 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.

SYNTEC

1.7.2.1 Set Kernel Server

- Command:

  ✧ F5 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:

  ✧ F5 Maintain→F2 Network setting→ F5 Set Kernel Server→ F1 Start

    Server

- Description

  ✧ Start server immediately

1.7.3 Fast Diagnostic

| G54 | | TEST N0 L1 | Fast Diagnostic | | 2013/7/2 | 20:13:35 |

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

●Ready   Not Select   Alarm

- Command:
  - F5 Maintain→F3 Fast diagnostic
- Description
  - Display simple diagnostic information of system and axes

1.7.3.1 System Data

G54		TEST N0 L1	Fast Diagnostic		2013/7/2	20:13:35

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

●Ready   Not Select   Alarm

- Command:
  - ✧ F5 Maintain→F3 Fast Diagnostic→ F1 System Data
- Description
  - ✧ Display simple diagnostic information of system

1.7.3.2　　　Axes Data

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:
  - ✧　F5 Maintain→F3 Fast Diagnostic→ F2 Axes Data
- Description
  - ✧ 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:
    - ◇ F5 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

- Command:
    - ✧ F5 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 |

System Data Backup

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

●Ready   Not Select   **Alarm**
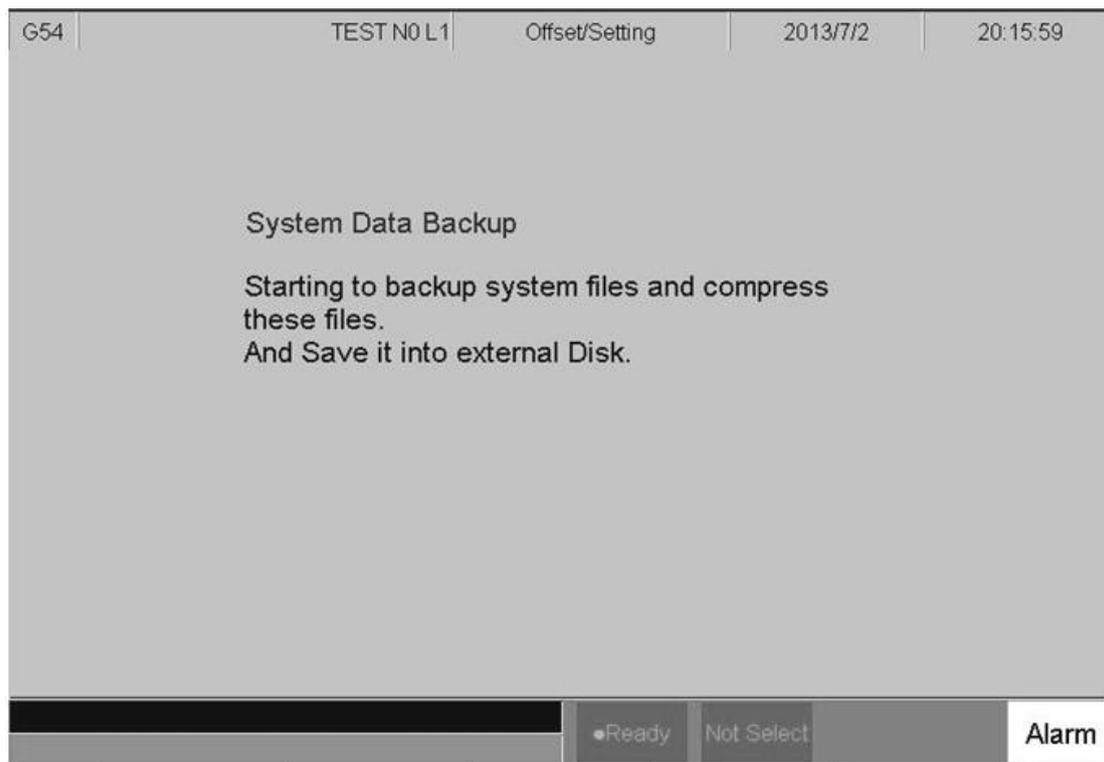
- ● Command:
    - ✧ F5 Maintain → Next→ F1 Backup System
- ● Description
    - ✧ Compress backup system data and save it into external device.

1.7.7 About

- ● Command:
    - ✧ F5 Maintain → Next→ F5 About
- ● Description
    - ✧ 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**
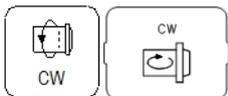
	Spindle CW rotation
	Spindle stop
	Spindle CCW rotation
	Spindle rate deceleration: Spindle speed will decelerate 10%.
	Spindle rate 100%: Spindle will rotate with 100%.
	Spindle rate acceleration: Spindle speed will accelerate 10%.
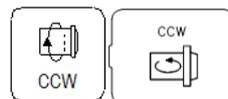
■ **Working LED**
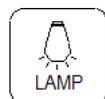
● Turn ON/OFF working LED

■ **G01 Rank**
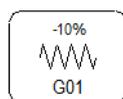
Decrease G01 rate 10%   Increase G01rate10%   G01 rate100%
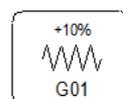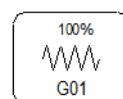
■ **G00 Rank**
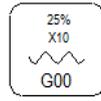
G00 rate x1   G00 rate x10   G00 rate x100   G00 rate x1000
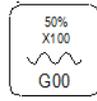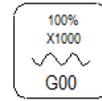
- Those function key can be regard as G00, MPG or INJOG rank.

  - **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**
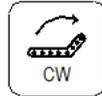
  - **Chip conveyor machine move in counter clockwise direction**
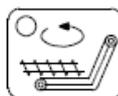
  - **Augment key(User define)**
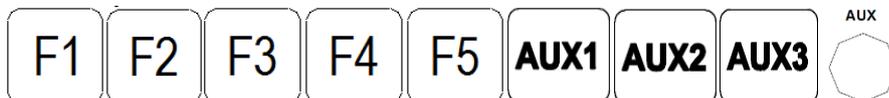
- Machine builder augment key, combine with PLC ladder.

■ **Direction of linear Axes &Rotation Axes & fast travel key**
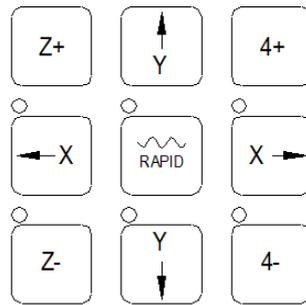
● Move the axes with fast travel speed key when press the axes and fast travel key at the same time.

2.2 Text Key Description

| A | ~ | Z | : | Place key : total 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 "CNC status, only use this key if necessary |

ENTER	:	To input current data to input box
HELP	:	show Help function/message on screen
/	:	For optional skip of program
;	:	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
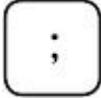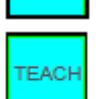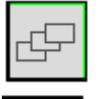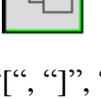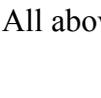
"(", ")", "[", "]", "|", "!", "&", "$", "#", "<", ">", "=", "%", "@", "*", ":", ",", "+", "-"

All above symbols are used for "Program Edit" mode.

↑
↓

Page Up / Page Down key

↑
← →
↓

Directional of cursor key

3 How to Operate 6 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)        Machine
                                                      X         0.000
   X    0.000      X    0.000      X    0.000          Y         0.000
   Y    0.000      Y    0.000      Y    0.000          Z         0.000
   Z    0.000      Z    0.000      Z    0.000          A         4.158
   A    0.000      A    0.000      A    0.000
                                                      Relative
                                                      X         0.000
                                                      Y         0.000
   MPG Shift        G54P3(G56)      G54P4(G57)         Z         0.000
                                                      A         4.158
   X    0.000      X    0.000      X    0.000
   Y    0.000      Y    0.000      Y    0.000         Aux. Coord
   Z    0.000      Z    0.000      Z    0.000          X         0.000
   A    0.000      A    0.000      A    0.000          Y         0.000
                                                       Z         0.000

                                   ●Ready    Auto                  Alarm
```

✧ 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 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 5 buttons system, user can use this function by press F3 Offset/Setting →F1 Workpeice Cord. →F5 Middle Func.

.

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 [Aux. Coord. Teach] insert the Aux. Coord value then this system will set the value accordind to the Aux Coord.

3.3.2.2 **Manual 3Pts Center**

◇ 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 [Aux. Coord. Teach] insert the Aux. Coord value then this system will set the value accordind to the Aux Coord.

3.3.2.3 **OUTAutoCenter**

◇ 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 [Aux. Coord. Teach] insert the Aux. Coord value then this system will set the value according to the Aux Coord.

### 3.3.2.4 **INAutoCenter**

$\diamondsuit$ Operation:

$\diamondsuit$ Set the Middle func. as 1, the left upper side will show Manual Center.

$\diamondsuit$ Set the second line, the middle method, as 1. You can see the screen as this figure.

$\diamondsuit$ Key in Length I and Width J for workpiece real length and width.

$\diamondsuit$ Set Z coordinate, this value is for the auto measurement that tool can touch the workpiece edge plane

$\diamondsuit$ Set Safe Distance H, this distance is that the tool will not hit with workpiece.

$\diamondsuit$ Set feed rate F, this feed rate is the measuring tool velocity during auto center process.

$\diamondsuit$ 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 <u>reference tool</u> 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

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

✧ 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~

Delta Z Set

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

●Ready   Auto   Alarm

(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

| G54 | | N0 L1 | Offset/Setting | 2013/8/22 | 18:15:00 |

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

●Ready Auto Alarm

(1~3)

✧ 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.

G54			TEST N0 L1	Offset/Setting		2013/7/2		20:18:05

Tool Manager Function

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

(0~96) Turrnt tool No.   ●Ready   Not Select   Alarm

✧ 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**"F5 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


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.

SYNTEC

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
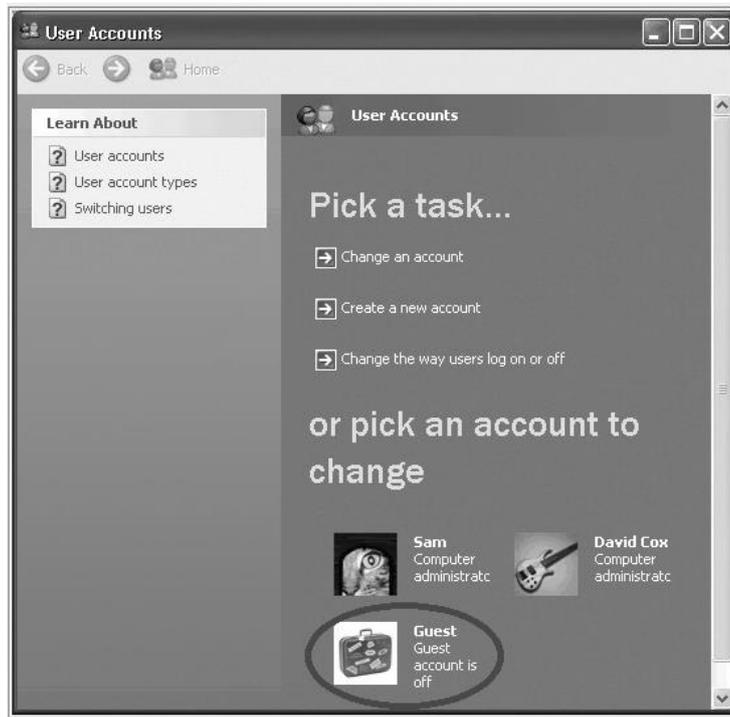
2. Sharing resource setting

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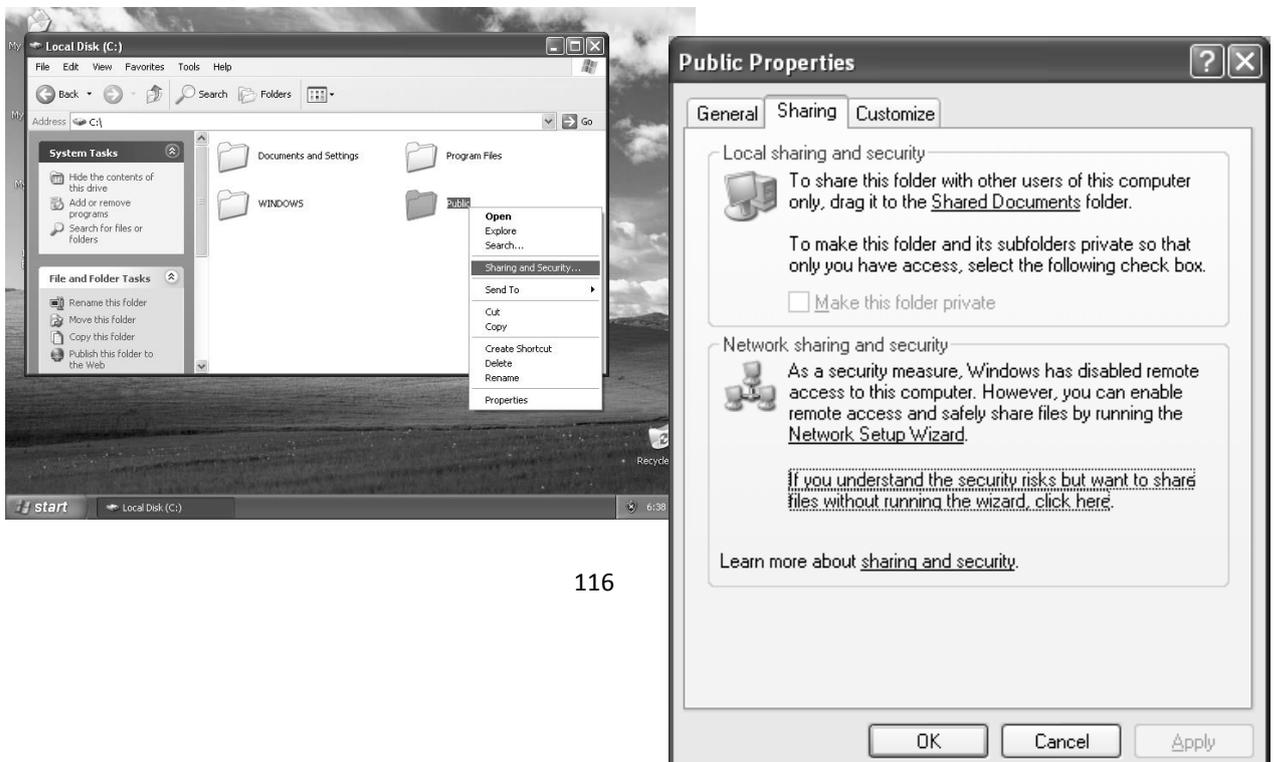
3. Click "OK" to confirm sharing setting; Select "Share this folder on the network", and "Allow network users to change my files".
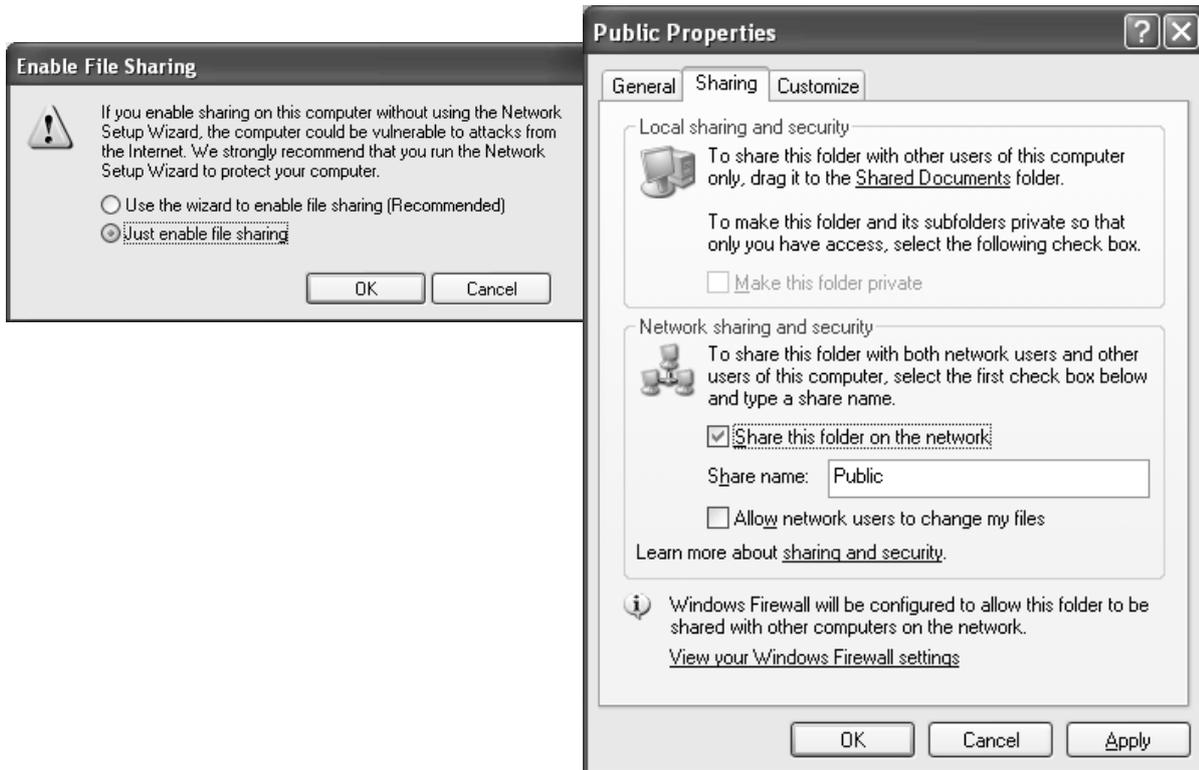
SYNTEC

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.
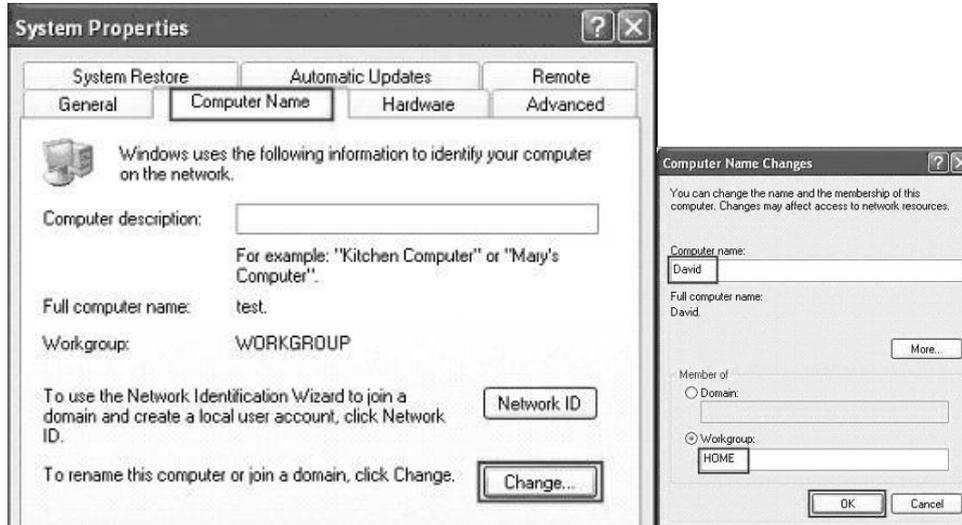
5. TCP/IP setting

"Start" => "Setting" => "Network connections" and right click on
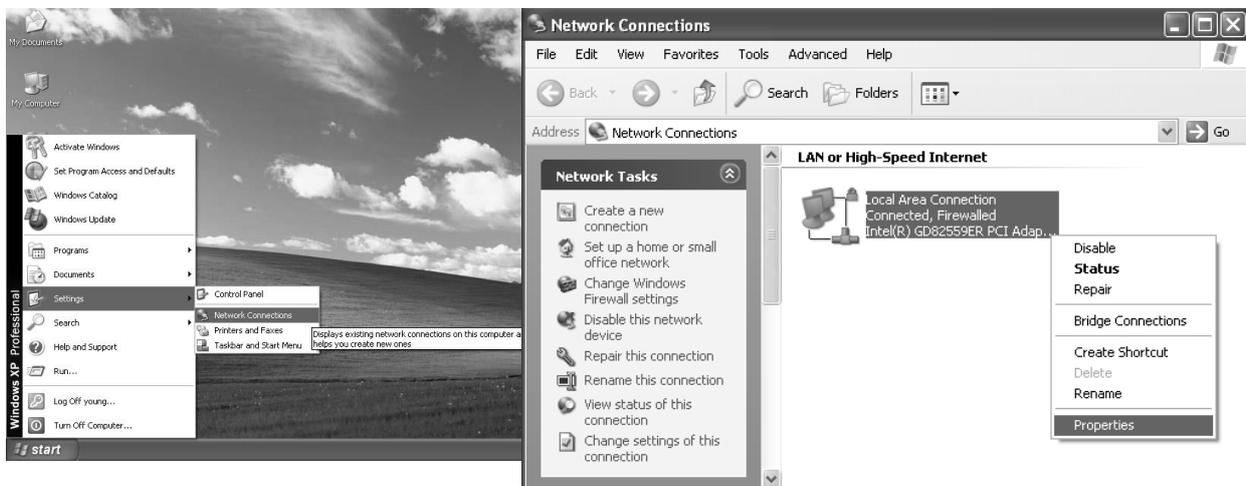

"Properties", and select "Internet Protocol [TCP/IP]"

✧ 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

Connect using:

Intel(R) GD82559ER PCI Adapter   Configure...

This connection uses the following items:

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

Install...   Uninstall   Properties

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

OK   Cancel

Internet Protocol (TCP/IP) 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: 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: .   .

Advanced...

OK   Cancel

3.4.7.2 **VISTA OS**

1. Guest account setting

Log in as Administrator and select "Start"→ "Control Panel"→ "User
Account"→ Guest
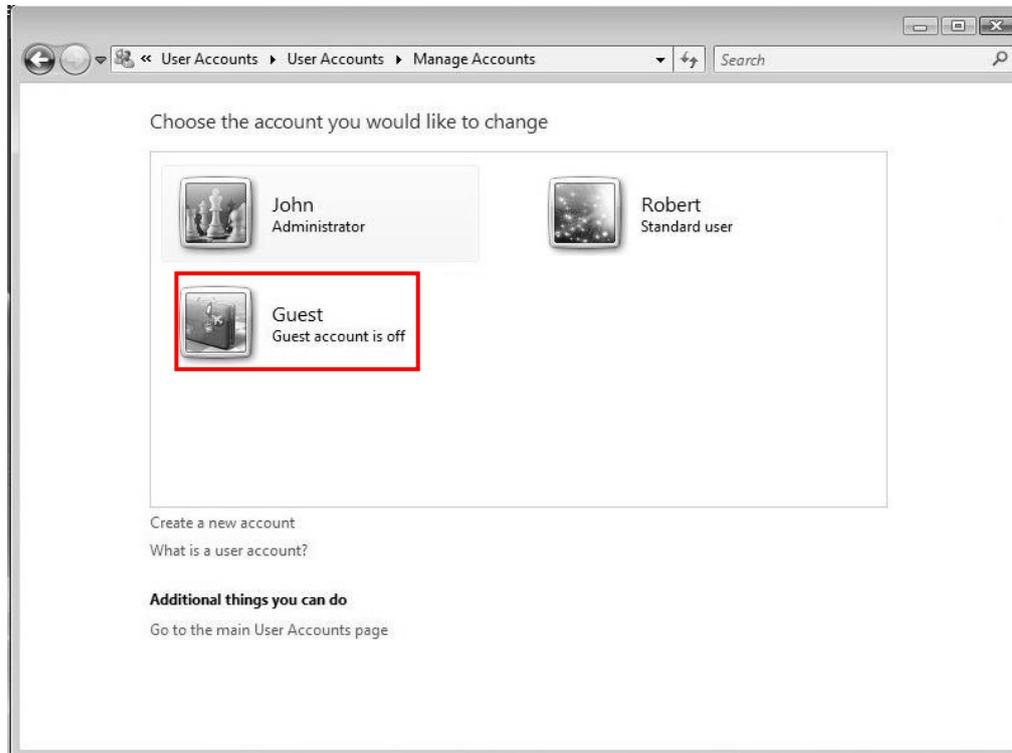
2. Sharing Resource Setting

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"
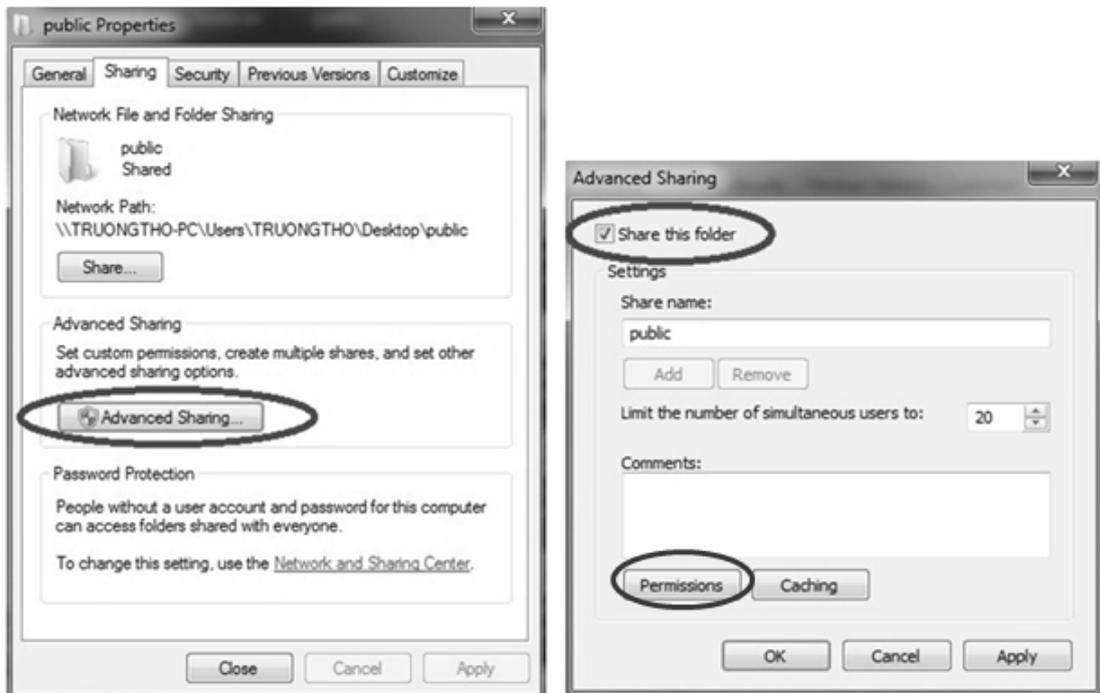
    C.   Click on "permission"

    D.   Click on "add"

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

3. Security setting

Right click on folder to share→ properties→ security→ Edit→ add "Guest" as a new group, then open group permissions to maximum.
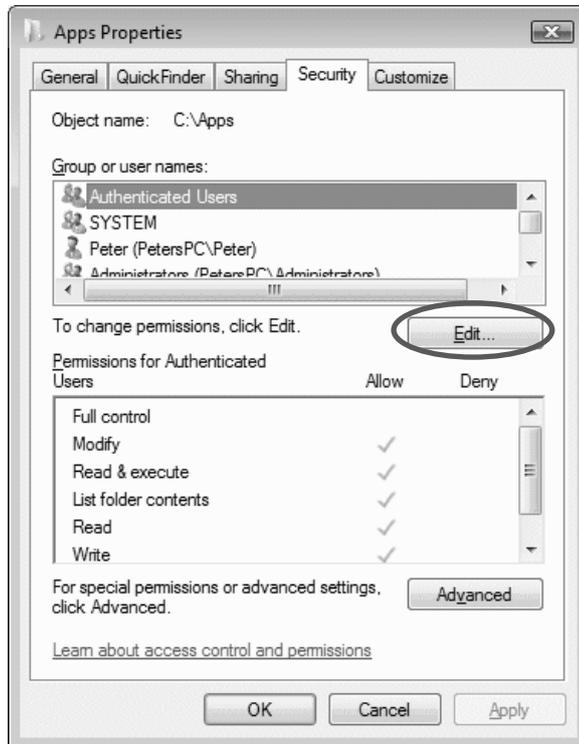
4. Setting PC name and workgroup

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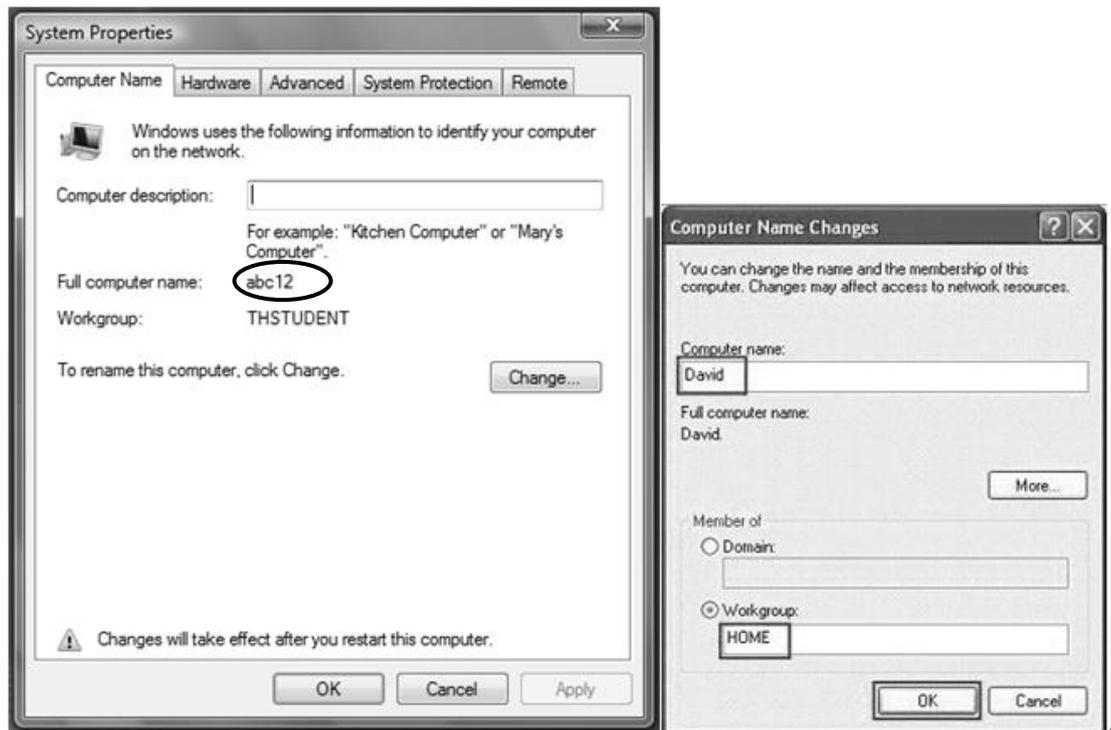
5. TCP/IP Setting

a. "Start" → "control panel" → "Network and Internet"→"Network and Sharing Center" → "Properties"
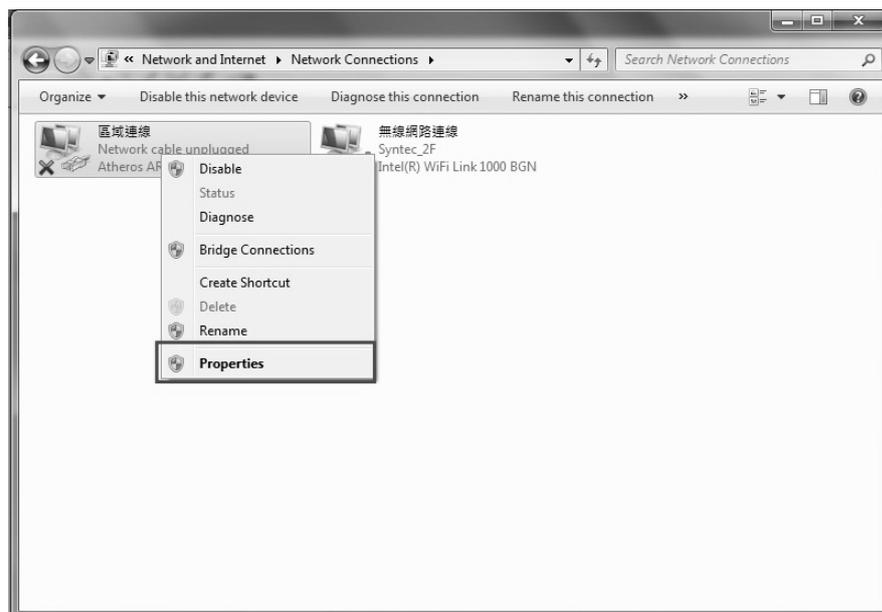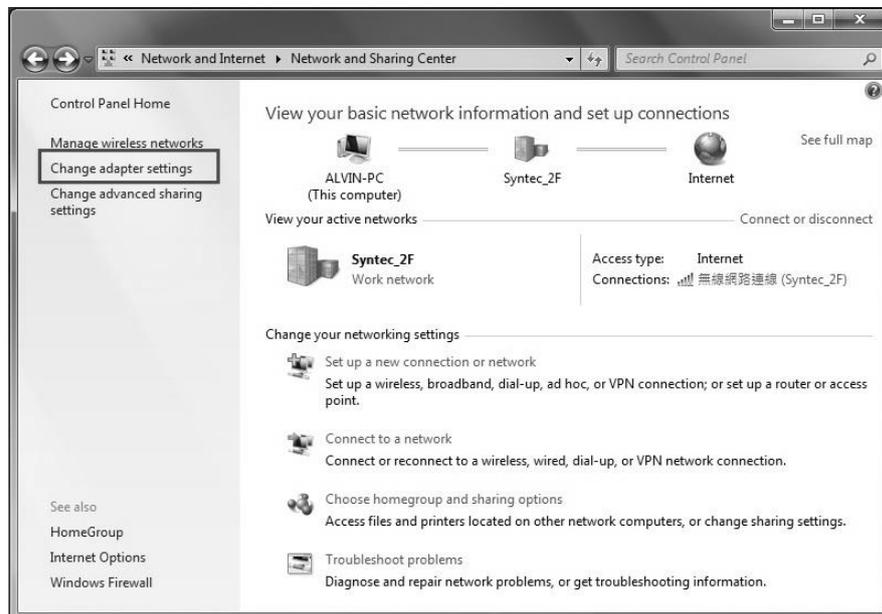
b. Select "internet protocol(TCP/IP)" as shown below:

❖ 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"
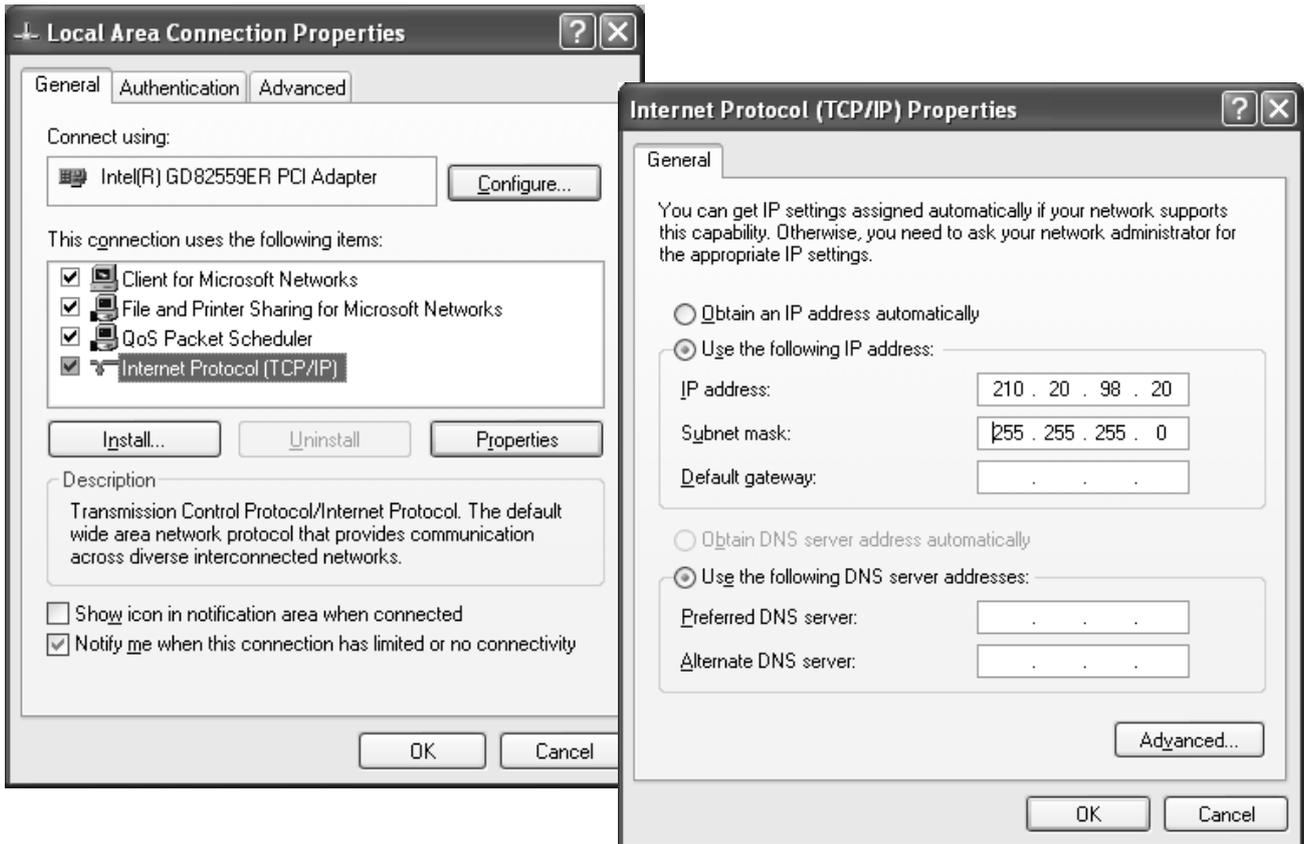
3.4.7.3　**Win 7 OS**

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
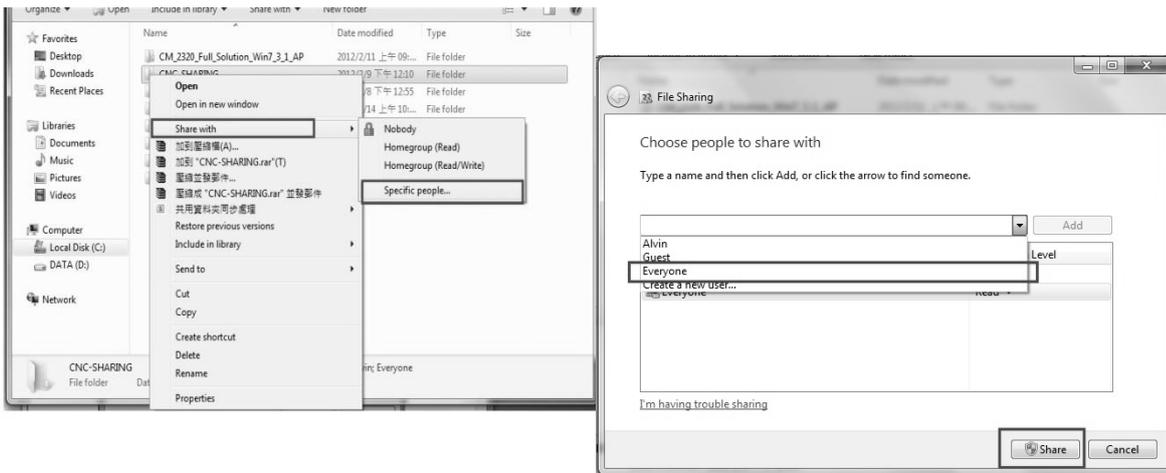
   ✧ Left-click on "advanced sharing" and select "share this folder" to everyone.
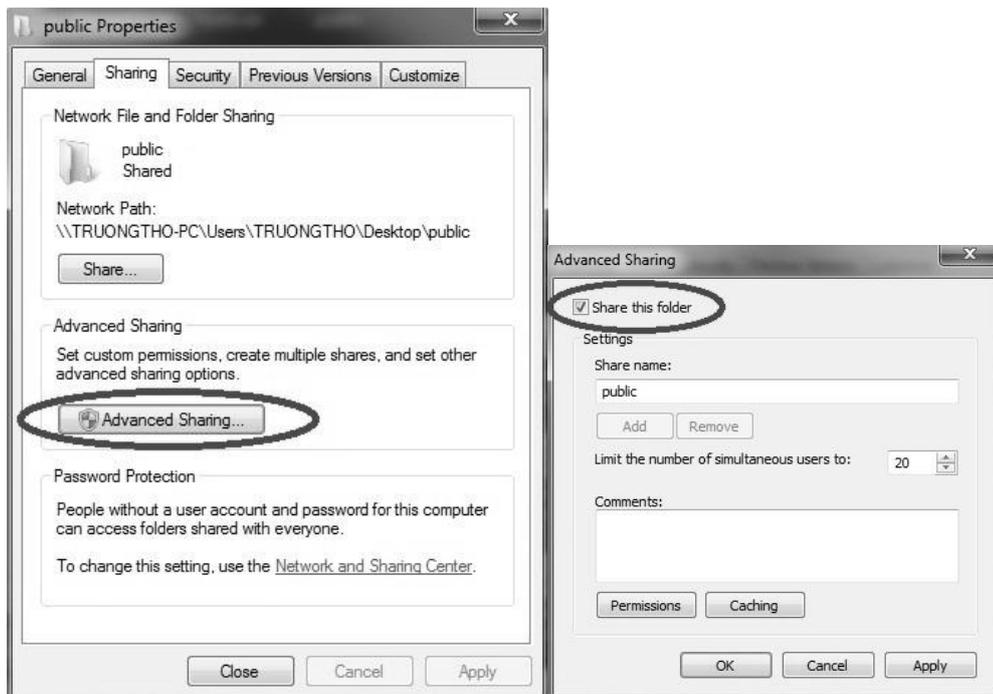
✧ Select "permission" and select "full control" "only read" and "change"
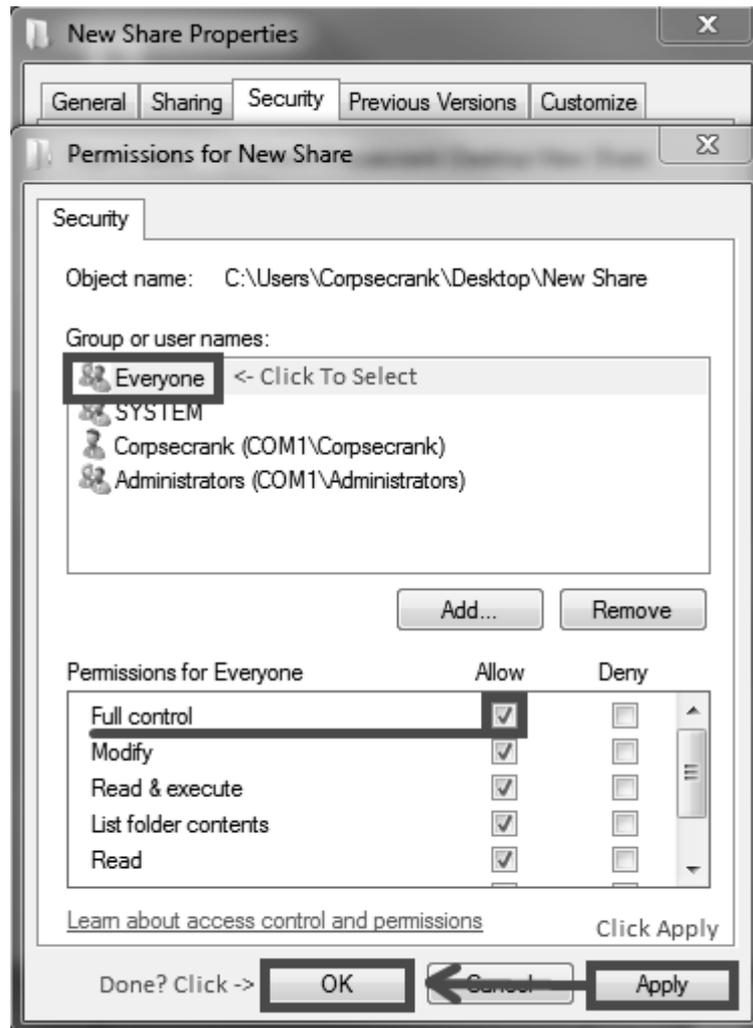
✧ Open "Network and sharing center", select "turn off password protected sharing" and "Open sharing….."
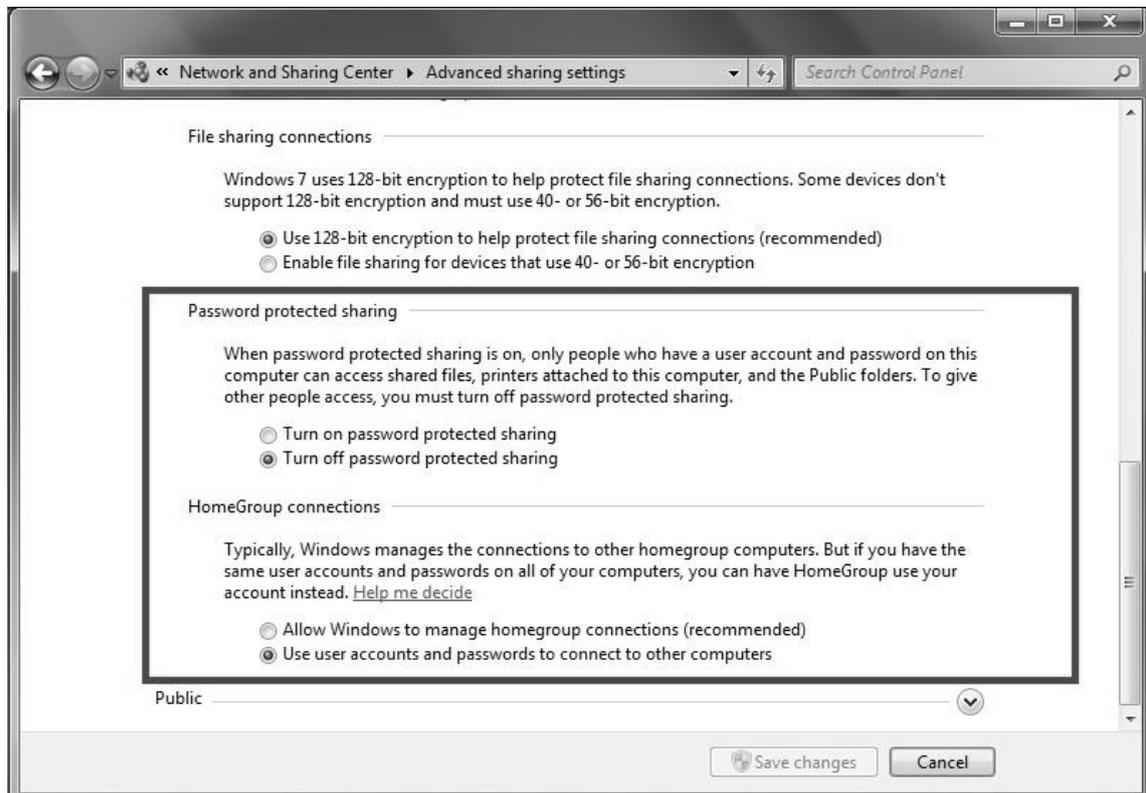
2. Setting PC name and workgroup

"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.

SYNTEC
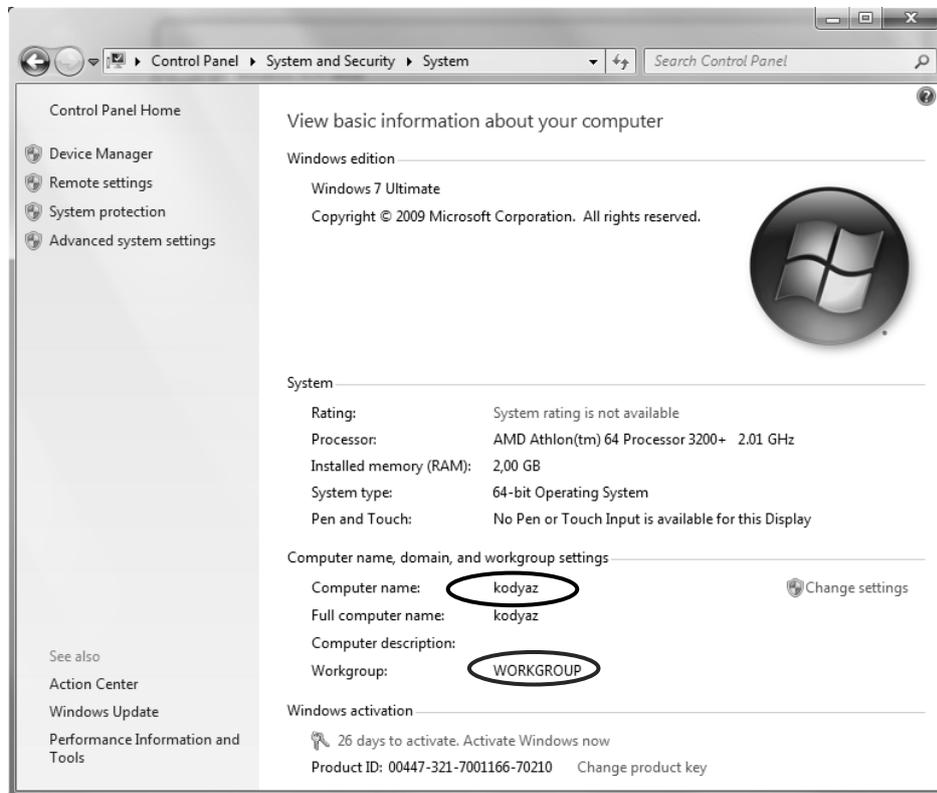
3. TCP/IP Setting

- 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"
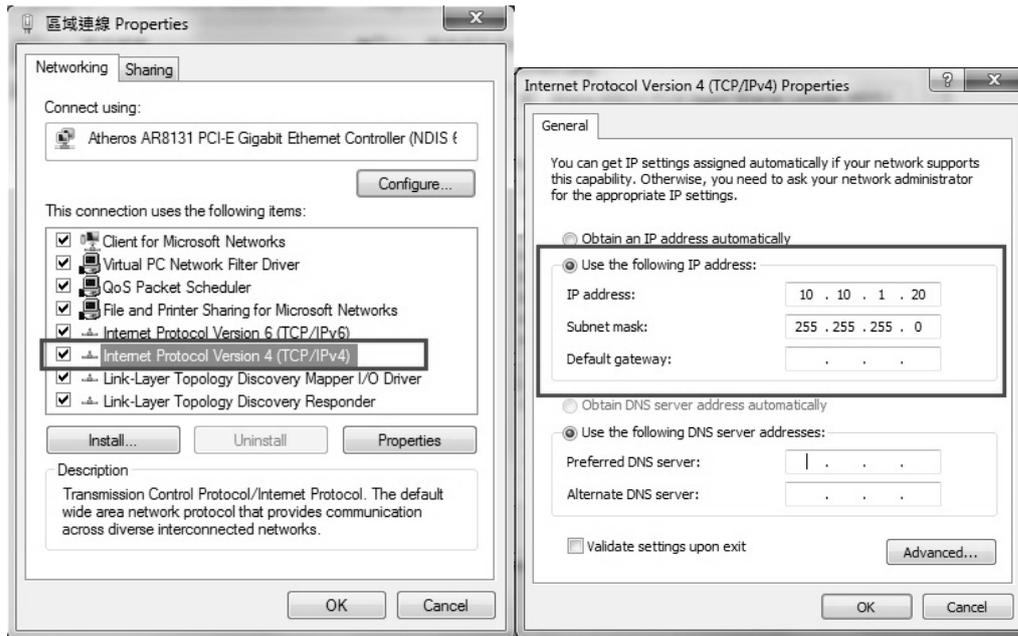
3.5 File Transfer

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
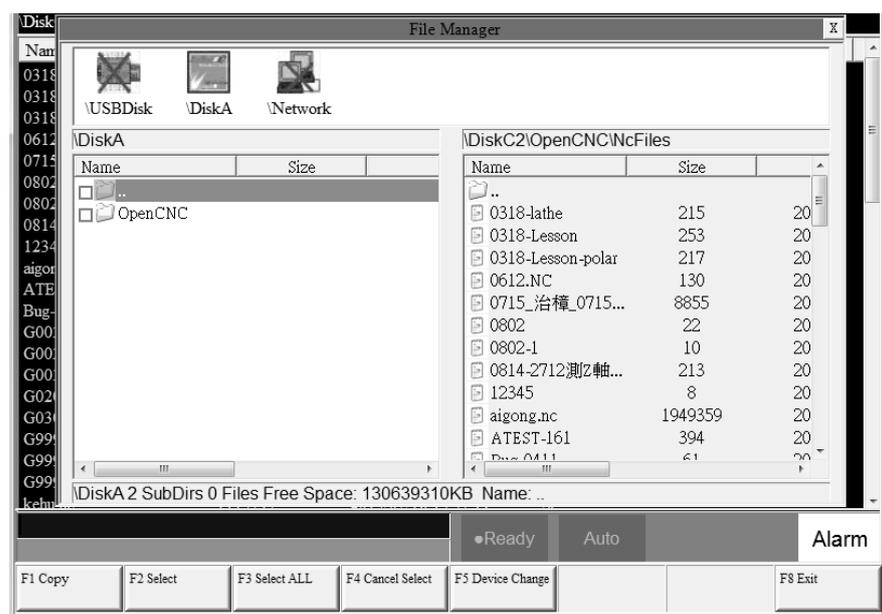
3.5.2 File Export

✧ 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

4 Appendix

4.1 Release Note

Doc. Ver.	Content	Release Date	Author	Reviewer	CNC Ver.
1.0	1st Version	2013/08/15	Tsai Chiju	Sandy Duan	1.0.6
1.1	Modify the content	2013/10/25	Sandy Duan	Yulius.Duma	1.0.6
1.2	Modify [WorkPiece Cord] key	2014/03/02	Andy Ngo	Changhungan	2.2.2

4.2 Contact Window

Hsinchu SYNTEC TECHNOLOGY CO., LTD. (Headquarters) No. 21 Industry E. Rd. 4, Hsinchu Science Park, Hsinchu City 30077, Taiwan, ROC TEL：+886-3-6663553 FAX：+886-3-6663505 E-Mail： sales@syntecclub.com.tw customerservice@syntecclub.com.tw	Jiangsu-Suzhou SYNTEC Equipment CO.,Ltd. D 1-8,Teng Fei New Industrial Zone, 5# Xing Han Street, Suzhou Industrial Park, China TEL：+86-512-69008860 FAX：0512-69560818
Taichung Taichung Service Agency No.31, Alley 9, Lane 271, Shepi Rd., Fengyuan City, Taichung County 420, Taiwan TEL：+886-4-25337731 FAX：+886-4-25349224	Taichung Taichung Service Agency No.42, Jingming St., West Dist., Taichung City 403, Taiwan TEL：+886-4-23102626 FAX：+886-4-23102636
Tainan Tainan Service Agency No.218, Denan Rd., Rende Dist., Tainan City 71756, Taiwan TEL：+886-6-2796707 FAX：+886-6-2796705	Jiangsu-SuZhou Suzhou Huaxin Numerical Control Technology Development CO.,LTD Xing Han Street, Suzhou Industrial Park, Jiangsu Province, to take off the new Su Industrial Square D, 2nd Floor 01-08 unit TEL：0512-69560828 FAX：0512-69560818
Zhejiang-Hangzhou SuZhou SYNTEC Equipment CO.,Ltd. Hangzhou Branch Room1202 Unit2 Buiding1, Edifice DiKai , Jincheng Road, Beigan Street,Xiaoshan District , Hangzhou City,Zhejiang Province, China TEL：+86-571-82751187 FAX：+-86-571-82751186	Zhejiang-Ningbo SuZhou SYNTEC Equipment CO.,Ltd. Ningbo Branch Room 12-07,No.262,416 Alley,Zhaohui Road,Jiangdong District, Ningbo City. TEL：+86-574-87750305 FAX：+86-574-87750306

Zhejiang-Wenling	Guangdong-Guangzhou
The Suzhou new generation of CNC Equipment Co., Ltd. Wenling Branch Room 1206, Department A, Zhenxing Plaza, Taiping Subdistrict, Wenling City TEL：0576- 86138372 FAX：0576-86119106	SuZhou SYNTEC Equipment CO.,Ltd. Guangzhou Branch Room 403 Stairs1 Youyi Building!ALuojia Village 8,Fuyi Road,ShijiTown,Panyu Distrct,Guangzhou City,Guangdong Province, China TEL：+86-20-34583040 FAX：+86-20-34583220
Guangdong-Dongguan	Shaanxi-Xi'an
SuZhou SYNTEC Equipment CO.,Ltd. Dongguan Branch Room 705 JinGuo Business affairs centre Xin an District Chang an TEL：+86-769-81660318 FAX：+86-769-81660328	The Suzhou new generation of CNC Equipment Co., Ltd. Xi'an Office Room 2503,Unit 1,Rancho Santa Fe,No.36, Dianzi Third Road, Yanta District, Xi'an, Shaanxi TEL：029-88287423 FAX：029-88287423
Shaanxi-Baoji	Shandong-Jinan
The Suzhou new generation CNC Equipment Co., Ltd. Baoji offices Room 2401, Building 1, Xinjian Road, Weibin District, Baoji TEL：18700712118	SuZhou SYNTEC Equipment CO.,Ltd. Jinan Branch Room 1112, Unit A, Jiahui Global Plaza, No.548, Beiyuan Street, Tianqiao District, Jinan, China TEL：+86-53185907208 FAX：+86-53185905708
Fujian-Xiamen	Tianjin
SuZhou SYNTEC Equipment CO.,Ltd. Xiamen Branch No. 100,B Chuang Chang Fang 3F East,Jin Fu Road,Tong An District,Xiamen City, Fujian Province, China TEL：+86-592-7191901 FAX：+86-592-7220536	The Suzhou new generation of CNC Equipment Co., Ltd. Tianjin Branch Haiyi International 3-1416, No.2, Rongyuan Road, Huayuan Industries Park, Binhai Hi-and-New Tech Park, Tianjin TEL：+86-22-87134111 FAX：022-87134111
Henan-Luoyang	Chongqing
The Suzhou new generation of CNC Equipment Co., Ltd. Luoyang Company	The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing

601B, Runsheng Building, Cross of Sanshan Road & Heluo Road, Hi-and-New Tech Park of Luoyang, Henan TEL：0379-65110352 FAX：0379-65110352	Branch Room 41-4, Building A, Hongding International, Guanyinqiao, Jiangbei District, Chongqing TEL：023-67913296 FAX：023-67634382
Jiangsu-Nanjing The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch Room 505, Unit 3, Building 12,Wuyi Luzhou Guan Chu Yuan, No.99, Tianyuan Middle Road, Jiangning District, Nanjing TEL：0512-69008860-300	Hubei-Wuhan Suzhou new generation of CNC Equipment Co., Ltd. Wuhan Office Room2003-2006,A#Guangguguoji,456 #Luoyu Road,Donghukaifaqu,Wuhan City, China TEL：027-87638876 FAX：027-87204137
Liaoning-Shenyang The Suzhou new generation CNC Equipment Co., Ltd. Shenyang Office No.3 A 1-12-3, South Wenhui Street, Shenhe District, Shenyang, Liaoning TEL：15002491395 FAX：18698806526	Anhui-Hefei The new generation of CNC Equipment Co., Ltd. in Suzhou, Hefei Office Room 704, Building 13,Wanzhen Xiaoyao Garden Ⅳ, Taihu East Road, Hefei TEL：15951989576
Thailand SIAM RADERMEN Co., Ltd. 52/4,MOO 5,THEPARAK RD.(KM11.5)BANGPLEEYAI,BANGPLEE, SAMUTPRAKRAN 10540 TEL：+66-27553536 FAX：+66-27575476 E-Mail：lee_cheahow@yahoo.com	Malaysia SURE FIRST ENTERPRISE (M) SDN BHD. 47000 NO 19.JALAN TSB 6,TAMAN INDUSTRI SUNGAI BULOH.47000 SELANGOR DARUL.EHSAN TEL：+603-61577632 FAX：+603-61574632
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	France INTEGRATION CNC 6, Rue l'arché de Copigny, 76630 BAILLY, FRANCE TEL：+33 2 35 06 07 83 FAX：+33 2 35 06 07 83 http://www.integrationcnc.fr/ E-Mail：vaque.j@online.fr

TEL：+961-76-883778 FAX：+961-1-901807 E-Mail：kesmeyan@rockwell.tw	
Turkey KASIKCIOGLU ELEKTROMEKANIK OTOMASYON Besevler K.S.S. 17 BLOK NO:72 Nilufer/BURSA/TURKYIE TEL：+90-224-4434684 FAX：+90-224-4434685 E-Mail：murat@kasikcioglu.com	ISTANBUL/TURKEY AKSIS MAKINA Yeni doğan mahallesi karakaş Sk. Emintaş Erciyes Sanayi Sitesi no:12/206 TEL：+90 212 613 87 83 FAX：+90 212 613 87 85 http://www.aksismakina.com E-Mail：info@aksismakina.com