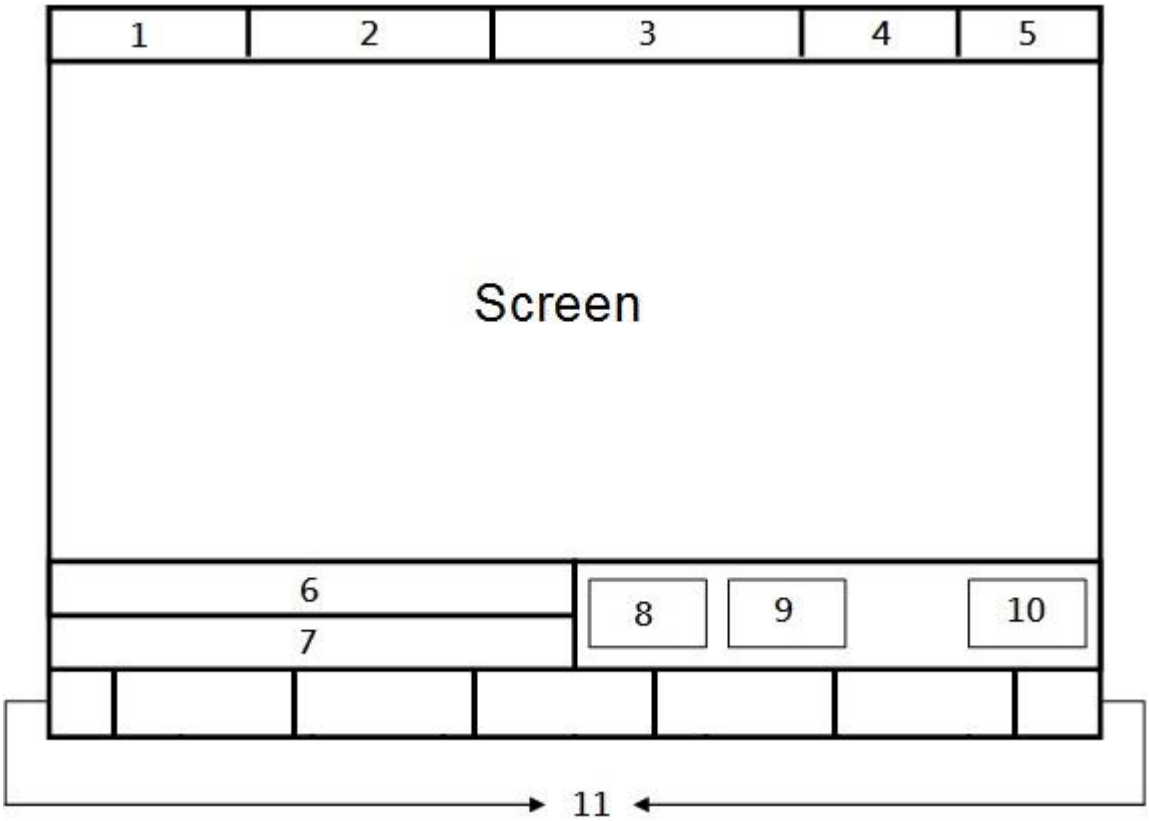
# 6 Series Lathe Operation Manual

**Date: 2015/11/13**

**Version: 1.3**

# 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

4.Date

5.Time

6.Data input

7.Display(Hint)

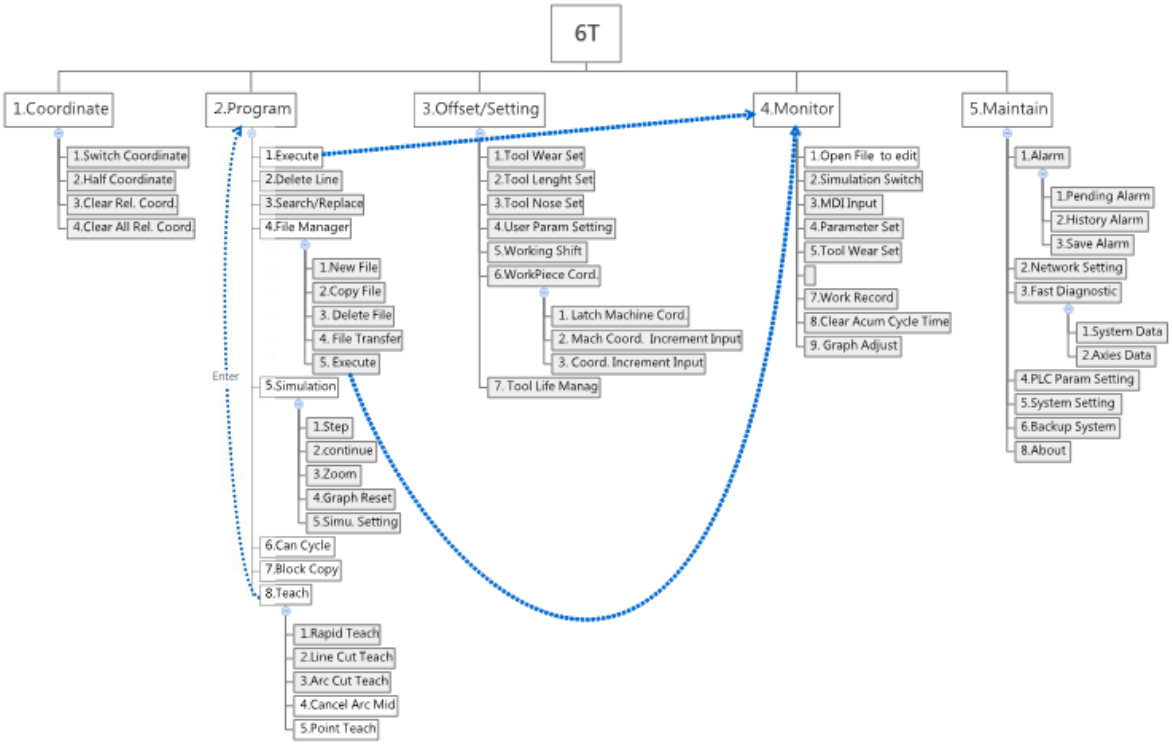
8.Status

9.Mode

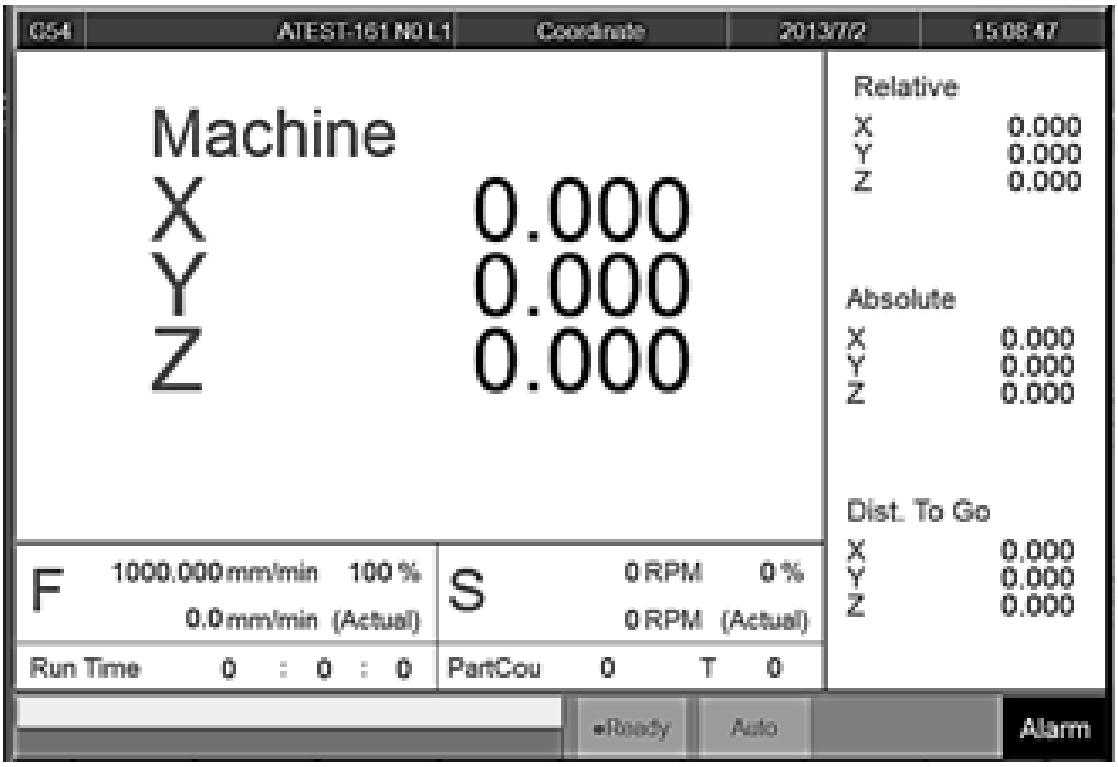
10. Alarm

11. Function Key Switch

## 1.2 CNC System Configuration



## 1.3 Coordinate




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

## 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 defines Feedrate (mm/min).
- ✧ Actual Feedrate of cutting tool (mm/min).
- ✧ Percentages of Feedrate that user define.

### 1.3.1.3 S(Rotating Speed of Spindle )

- ✧ User defines Spindle speed (RPM).
- ✧ Actual speed of spindle (mm/min).
- ✧ Percentages of Spindle speed that user define.

### 1.3.1.4 Run Time

- ✧ Timer for the execution of program.

### 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

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

### 1.3.3 Half Coordinate

- Path
  - ✧ F1 Coordinate→F2 Half Coordinate
- Function
  - ✧ Relative coordinate devided by 2.
  - ✧ Co-operate with 「Clear Rel. Coord.」function , this function can quickly 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.

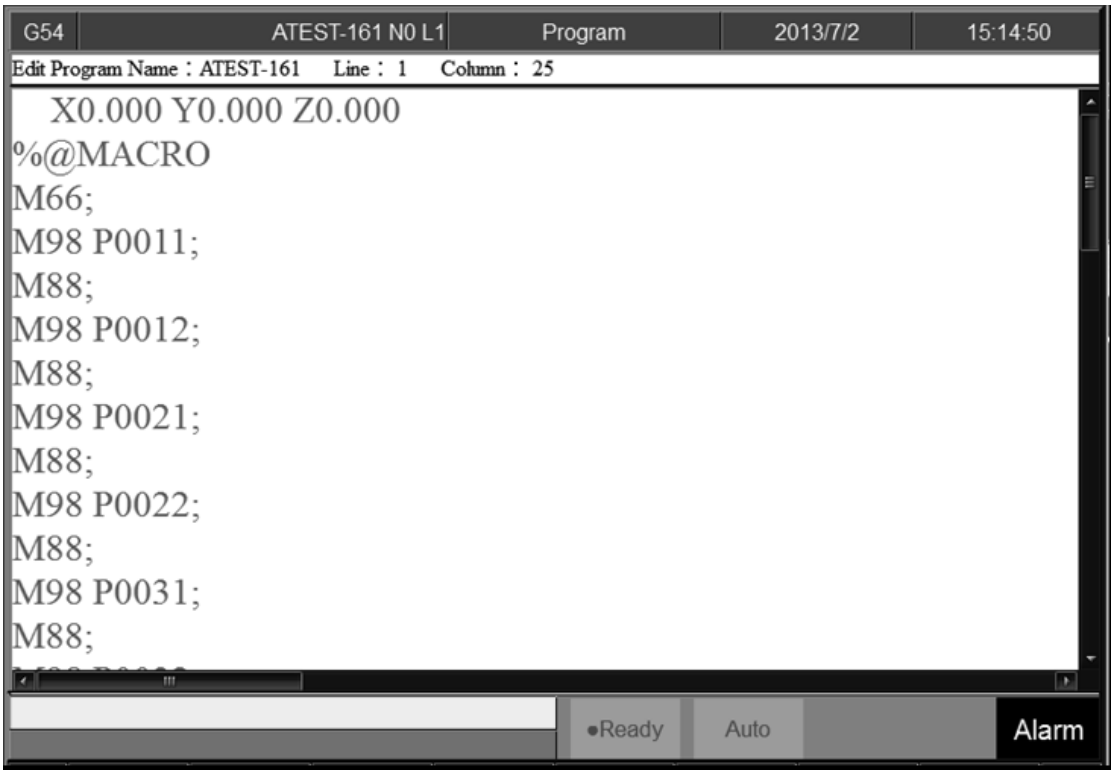
### 1.3.4 Clear Rel. Coord.

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

### 1.3.5 Clear All Rel. Coord.

- path
  - ✧ 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 become 0.000.

## 1.4 Program




- Path
  - ✧ 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 switch between the top and end of the line.

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

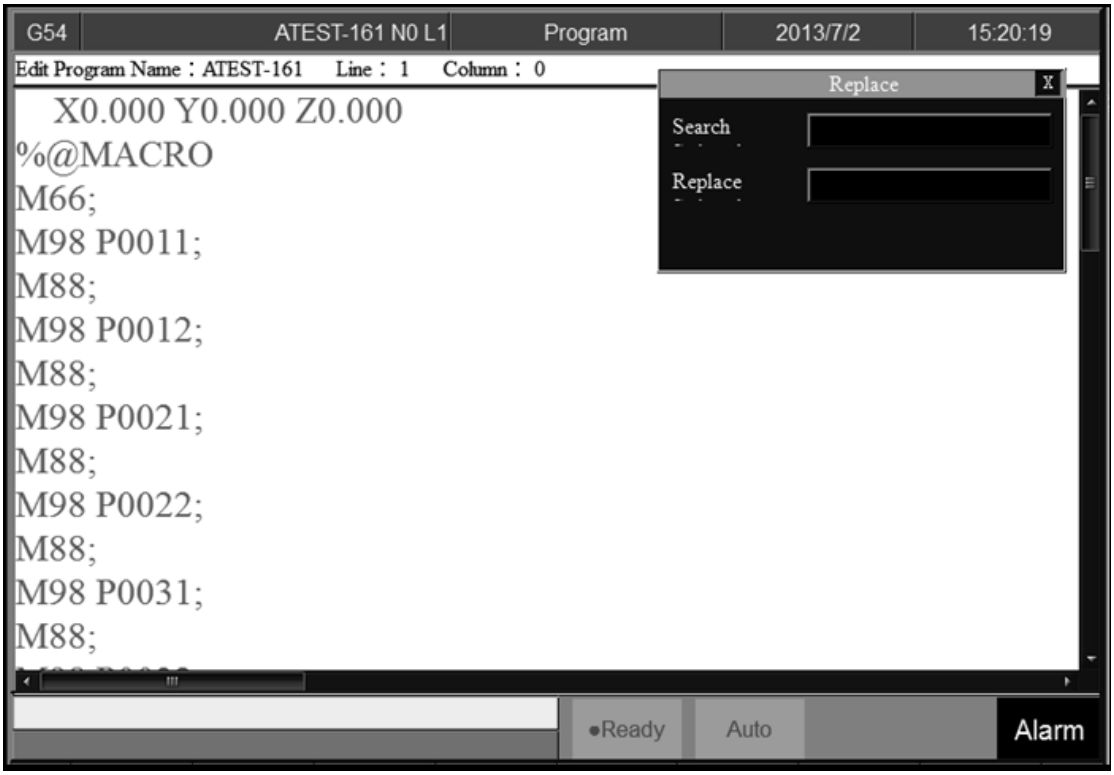
### 1.4.1 Execute

- Path
  - ✧ F2 Program→F1 Execute
- Function
  - ✧ This function can assign the current editing program as the executive program and also change the minitor to the 「Monitor」 page.
- Note
  - ✧ This function will out of control when machining.

### 1.4.2 Delete Line

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

## 1.5 Search/Replace




- Path
  - ✧ F2 Program→F3 Search/Replace
- Function
  - ✧ This function can help to find and replace the string within the editing program.
  - ✧ Press「Search/Replace」function key, a dialog box will appear to ask users input a string that need to find and replace for.
- Operation Method
  - ✧ With【ENTER】on the key pad, can alter the cursor location between「Search」and 「Replace」.

#### 1.5.1.1 Find Next

- Path
  - ✧ F2 Program→F3 Search/Replace→F1 Find Next
- Function
  - ✧ According to the「Search String」, find the next same string within the current editing program.
  - ✧ Cursor will move the next same string and the string will be highlight.

#### 1.5.1.2 Replace

- Path
  - ✧ F2 Program→F3 Search/Replace→F2 Replace
- Function
  - ✧ According to the「Search String」, find the next same string within the current editing program and replace with the「Replace String」
- Operation Method
  - ✧ If there's no highlight string on the screen, press「Find Next」.
  - ✧ Press F2「Replace」will replace the current highlight string with new string and also cursor will go to the next same「Search String」.
  - ✧ If you want to skip the current highlight stirng, press F1「Find Next」.

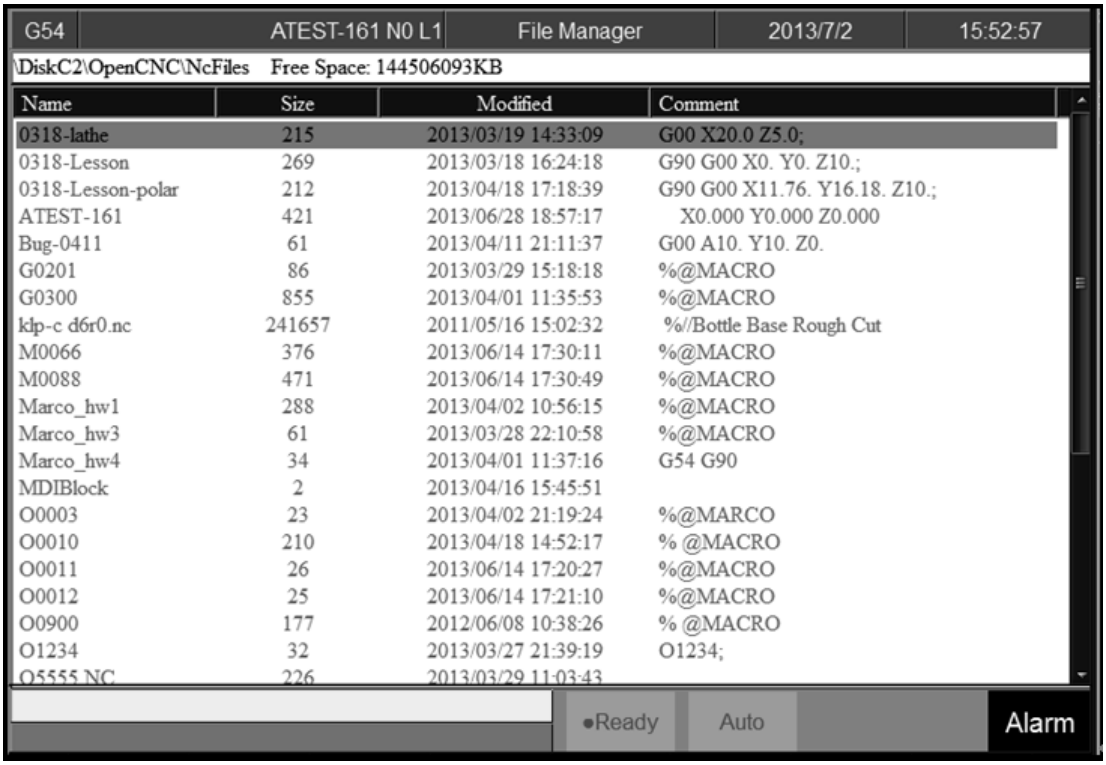
#### 1.5.1.3 **Replace All**

- Path
  - ✧ F2 Program→F3 Search/Replace→F3 Replace All
- Function
  - ✧ Replace all of the string that define in「Search String」with the new string and move the cursor to the line of last string.

#### 1.5.1.4 **Modify Setting**

- Path
  - ✧ F2 Program→F3 Search/Replace→F4 Modify Setting
- Function
  - ✧ Reset the 「Search/Replace」 string.
- Operation Method
  - ✧ After Executed 「Find Next」/ 「Replace」/「Replace All」cursor will change to the program edit place.
  - ✧ Press F4「Modify Setting」to reset the 「Search/Replace」string.

## 1.5.2 File Manager




- Path
  - ✧ 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.

#### 1.5.2.1 New File

- Path
  - ✧ 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 format of file name does not have file extension. If user want to create a new file with file extension such as *.NC, enter *.NC is ok.
  - ✧ The size of file name cannot be longer than 32 characters(include file extension)

#### 1.5.2.2 Copy File

- Path
  - ✧ 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 format of file name does not have file extension. If user want to create a new file with file extension such as *.NC, it is also ok.
- ✧ The size of file name must not be longer than 32 characters(include file extension)

### 1.5.2.3 Delete File

- Path
  - ✧ F2 Program→F4 File Manager→F3 Delete File
- Function
  - ✧ Delete file remarked by cursor.
- Operation Method
  - ✧ Press F3「Delete File」, selection box will show up in fornt of the NC file within the「File Manager」minitor page. Use【↑】【↓】to select the delete file.
- Sub-function Key
  - ✧ Select: Select file, can select more than one file and also can cancel the selection of one file.
  - ✧ Select All: Select all file.
  - ✧ Cancel Select: Cancel all of the selected files.
  - ✧ Delete File: Delete all of the selected files.
  - ✧ Delete All: Delete all file within data storage device.
- Note
  - ✧ Current Programming and machining file cannot be deleted.

#### 1.5.2.4 File Transfer

- Path
  - ✧ F2 Program→F4 File Manager→F4 File Transfer
- Function
  - ✧ Transmit data between controller and outer device.

#### 1.5.2.4.1 File Import

- Path
  - ✧ F2 Program→F4 File Manager→F4 File Transfer→F1 File Import
- Function
  - ✧ Import outer file into controller
- Function Page Explanation
  - ✧ The upper block is the outer devices with the following choice.
    - ◆ USBDisk
    - ◆ DiskA
    - ◆ Network
    - ◆ USBDisk2
  - ✧ Left column are data storage in the outer device.
  - ✧ Right column are current data storage in the controller.
- Sub-function Explanation
  - ✧ Copy: Copy the remarked file from the outer device to the controller.
  - ✧ Select: Select file, can select more than one file and also can cancel the selection of one file.
  - ✧ Select All: Select all file.
  - ✧ Cancel Select: Cancel all of the selected files.
  - ✧ Device Change: Change outer device.
- 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 data structure will change, according to the 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.

#### 1.5.2.4.1 File Export

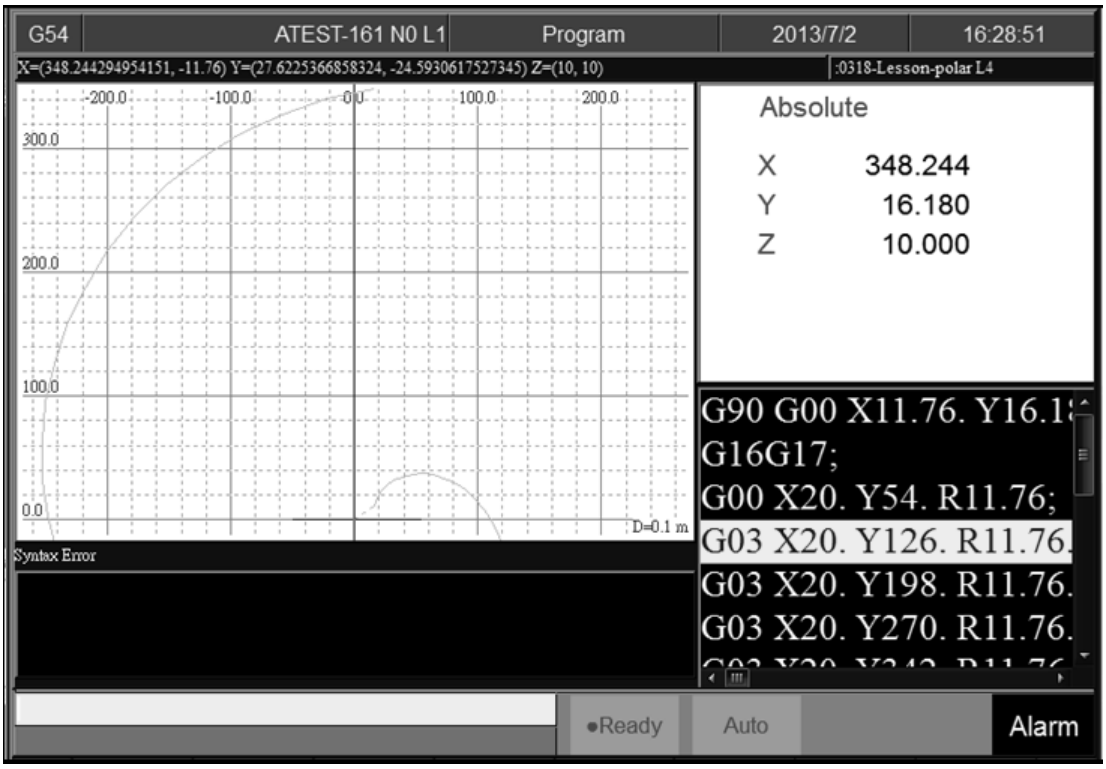
- Path
  - ✧ F2 Program→F4 File Manager→F4 File Transfer→F2 File Export
- Function
  - ✧ Export file within controller to outer device.
- Function Page Explanation
  - ✧ The upper block is the outer devices with the following choice.
    - ◆ USBDisk
    - ◆ DiskA
    - ◆ Network
    - ◆ USBDisk2
  - ✧ Left column are data storage from controller.
  - ✧ Right column are current data storage in outer device.
- Sub-function Explanation
  - ✧ Copy: Copy the remarked file from the outer device to the controller.
  - ✧ Select: Select file, can select more than one file and also can cancel the selection of one file.
  - ✧ Select All: Select all file.
  - ✧ Cancel Select: Cancel all of the selected files.
  - ✧ Device Change: Change outer device.
- Operation Method
  - ✧ Press「File Export」, a dialog box will appear.
  - ✧ Default outer device is USBDisk.
  - ✧

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

### 1.5.2.5 Execute

- Path
  - ✧ F2 Program→F4 File Manager→F5 Execute
- Function
  - ✧ This function can assign the current cursor located program as the executive 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.
  - ✧ The screen will change to the 「monitor」 page.
- Note
  - ✧ This function will be out of control when machining.

### 1.5.3 Simulation




- Path
  - ✧ F2 Program→F5 Simulation
- Function
  - ✧ This function can simulate the program and make a prediction for the actual machining route.
  - ✧ Capability of debug.
  - ✧ Default display range will be the largest limit of the program.
  - ✧ Relative simulate function can be modify by changing 「simulate Setting」.

#### 1.5.3.1 Step

- Path
  - ✧ F2 Program→F5 Simulation→F1 Step
- Function
  - ✧ Simulate the program step by step.
  - ✧ Minitor the variation of the coordinate step by step.

#### 1.5.3.2 Continue

- Path
  - ✧ F2 Program→F5 Simulation→F2 Continue
- Function
  - ✧ Kernel will scan all of the program and then do the simulation.

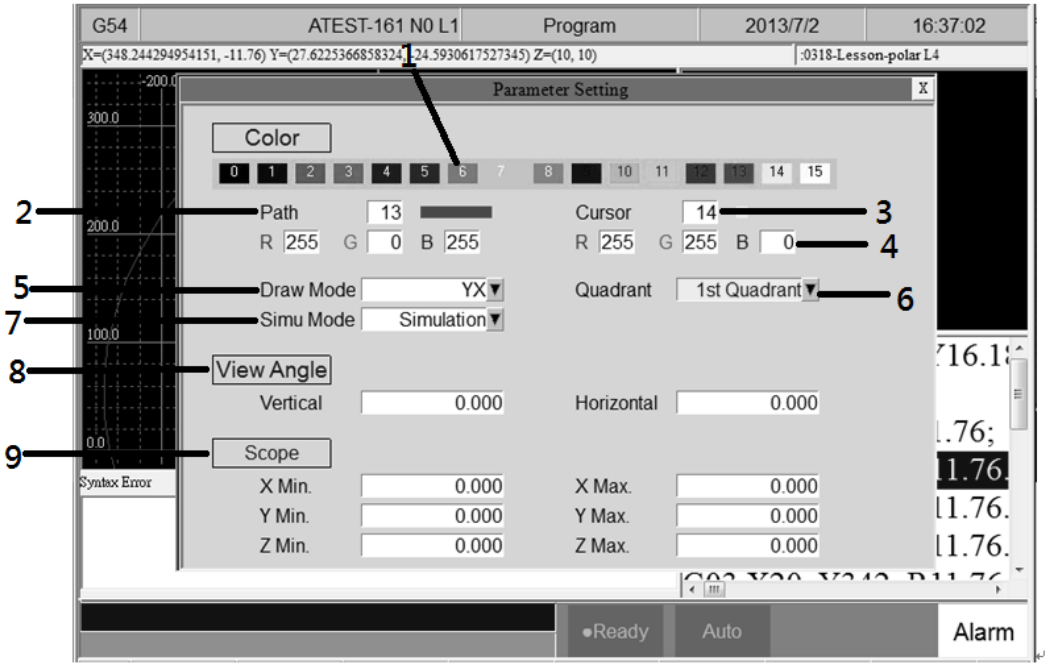
#### 1.5.3.3 Zoom

- Path
  - ✧ F2 Program→F5 Simulation→F3 Zoom
- Function
  - ✧ Enlarge/contract the simulation result
- Operation Method
  - ✧ Press F3「Zoom」,there will be a block show up. Use 【↑】【↓】【←】【→】can move the block to up and down, left and right.
  - ✧ Use【Page Up】【Page Down】to enlarge/contract the area within the block.
  - ✧ After the area is selected, press【ENTER】to check the result.

### 1.5.3.4 **Graph Reset**

- Path
  - ✧ F2 Program→F5 Simulation→F4 Graph Reset
- Function
  - ✧ According the program reset the simulation result to the normal condition.

### 1.5.3.5 **Simu. Setting**




- Path
  - ✧ F2 Program→F5 Simulation→F5 Simu. Setting
- Function
  - ✧ Setting the relative simulation item.
- Simulation parameter
  1. Color
     - ◆ Provide 0~15 total 16 different kind of color.
  2. Path Color
     - ◆ Simulation path color.
     - ◆ Provide 0~15 total 16 different kind of color.

3. Cursor Color
    - ◆ Color of cursor point
    - ◆ Provide 0~15 total 16 different kind of color.
4. RGB Value
    - ◆ Except the provided 16 different kind of color, user can define the color by theirself.

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 profile simulate plane is on which quadrant.

■Quadrant can define are as below.

- First
- Second
- Thrid
- Fourth

7. Simulate Mode

◆ Setting profile simulate method.

● Simulation

◆ When user change to the 『Monitor』 page, simulation will become automatically.

◆ When the system had scan all of the programs, simulation will execute, user do not need to define the simulate boundary.

● Direct Draw

◆ When user change to the 『Monitor』 page, cursor will show up but simulation will not execute automatically.

◆ User need to define the simulation scope first.

◆ When the machining is start, following to the program cursor will begin simulation.

● Not Simulation

◆ Close the simulation function.

8. View Angle Setting

■Under XYZ draw mode, by setting this parameter, simulation will become 3D simulation.

■View angle can define are as below.

◆ Vertical

◆ Horizontal

9. Scope

■Scope can define are as below.

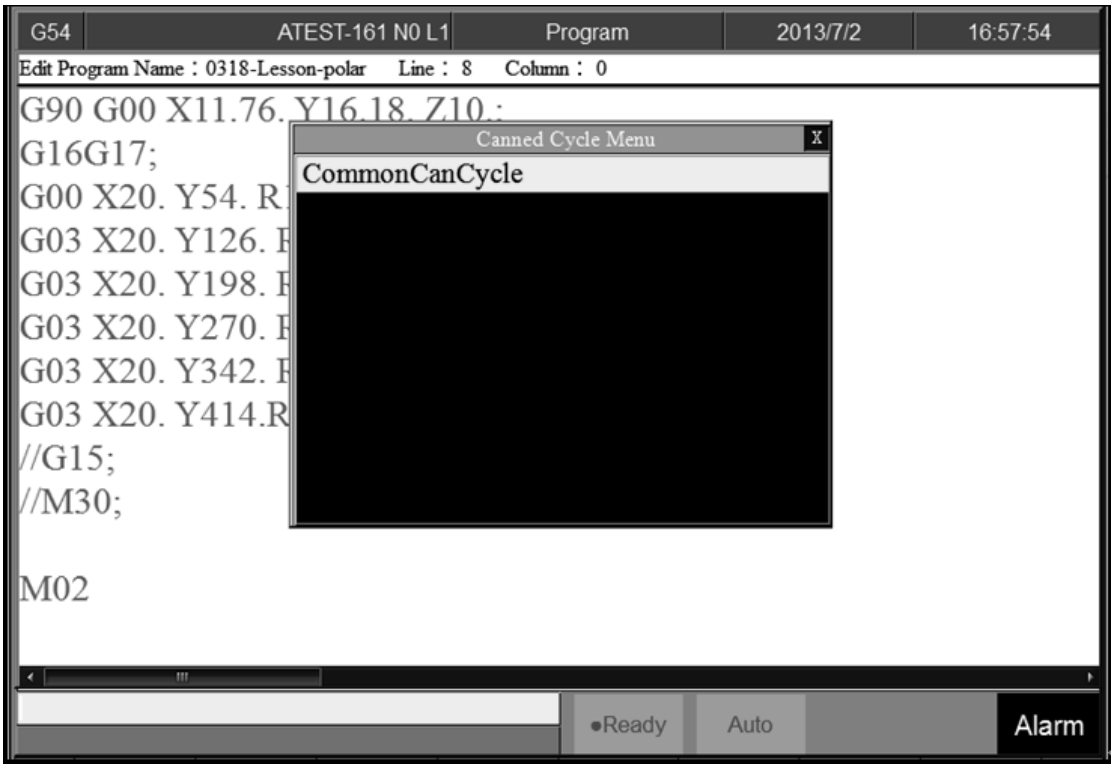
■Minimum

- X-Axis
- Y-Axis
- Z-Axis

■Maximum

- X-Axis
- Y-Axis
- Z-Axis

## 1.5.4 Can Cycle




- Path
  - ✧ F2 Program→Next→F1 Can Cycle
- Function
  - ✧ Because Syntec system provides many kind of G code and different G code have different kind of function. When editing the program, this function can help user to edit G code easily.

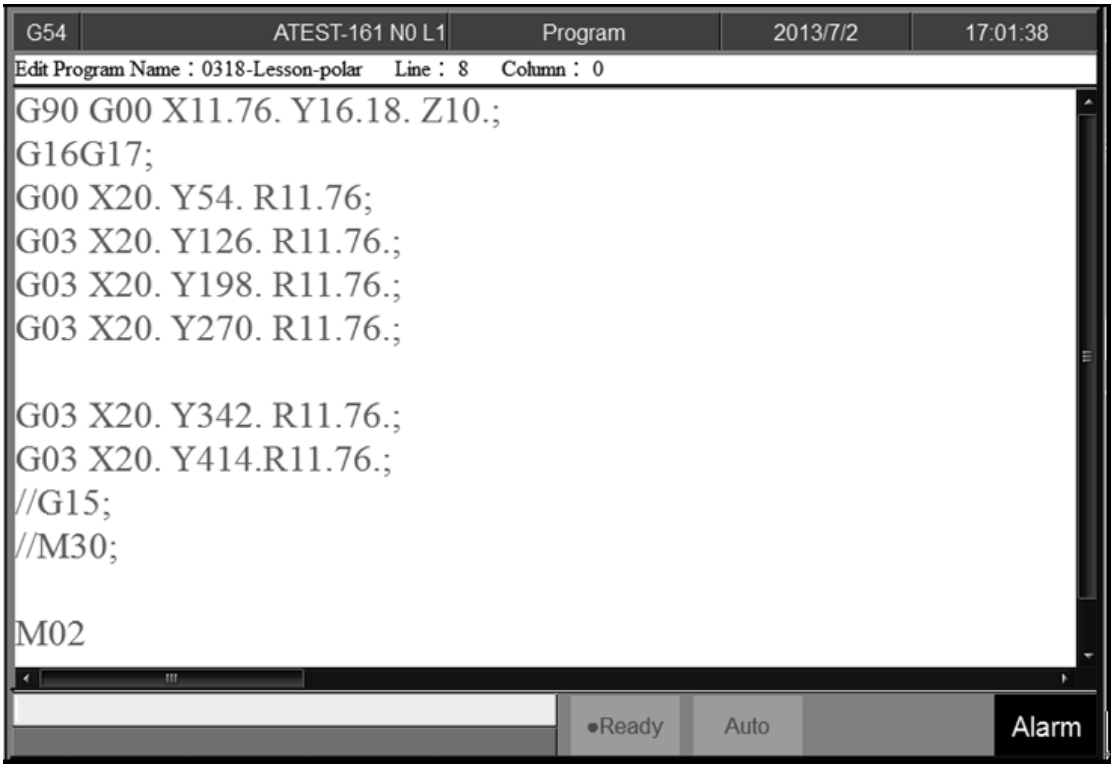
#### 1.5.4.1 Insert Cycle

- Path
  - ✧ F2 Program→Next→F1 Can Cycle→F1 Insert Cycle
- Operation Method
  - ✧ Under the program edit page, move the cursor to the insert location and press 「Insert Cycle」, follow the instruction and insert the require G code.
  - ✧ Press 「OK」, the desire G code will insert into the next line of the current cursor.

#### 1.5.4.2 Edit Cycle

- Path
  - ✧ F2 Program→Next→F1 Can Cycle→F2 Edit Cycle
- Function
  - ✧ Edit the cycle of current cursor.
- Operation Method
  - ✧ Move the cursor to the intend modify location, press 「Edit Cycle」,a modify page will show up. Modify the content and press「OK」, the content of the current cursor will change.

### 1.5.5 Block Copy




- Path
  - ✧ F2 Program→Next→F2 Block Copy
- Function
  - ✧ In current program, this function provides more than one line of program select, cut, copy and paste action.

#### 1.5.5.1 Start Line

- Path
  - ✧ F2 Program→Next→F2 Block Copy→F1 Start Line
- Function
  - ✧ Select the beginning block.

#### 1.5.5.2 End Line

- Path
  - ✧ F2 Program→Next→F2 Block Copy→F2 End Line
- Function
  - ✧ Select the end block.

#### 1.5.5.3 Block Cut

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

#### 1.5.5.4 Block Copy

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

#### 1.5.5.5 Block Paste

- Path
  - ✧ F2 Program→Next→F2 Block Copy→F5 Block Paste
- Function

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

- Operation Method

  ✧ Move the cursor to the desire block and press「Start Line」and function key「End Line」enable.

  ✧ Use the key on panel【↑】【↓】【Page Up】【Page Down】to select the block, the block being select will be highlight.

  ✧ Confirm the block that had been selected and press 「End Line」.

  →Function key 「End Line」disable.

  →Function key 「Block Copy」enable.

  →Function key 「Block Cut」enable.

  →If 「Block Cut」being use, the whole block that had been highlight will be cut off.

  →Function key 「Block Copy」/「Block Cut」disable.

  →Function key 「Block Paste」enable.

  ✧ Move the cursor to the desire location and press 「Block Paste」, the content that had been cut or copy will paste at the cursor location.

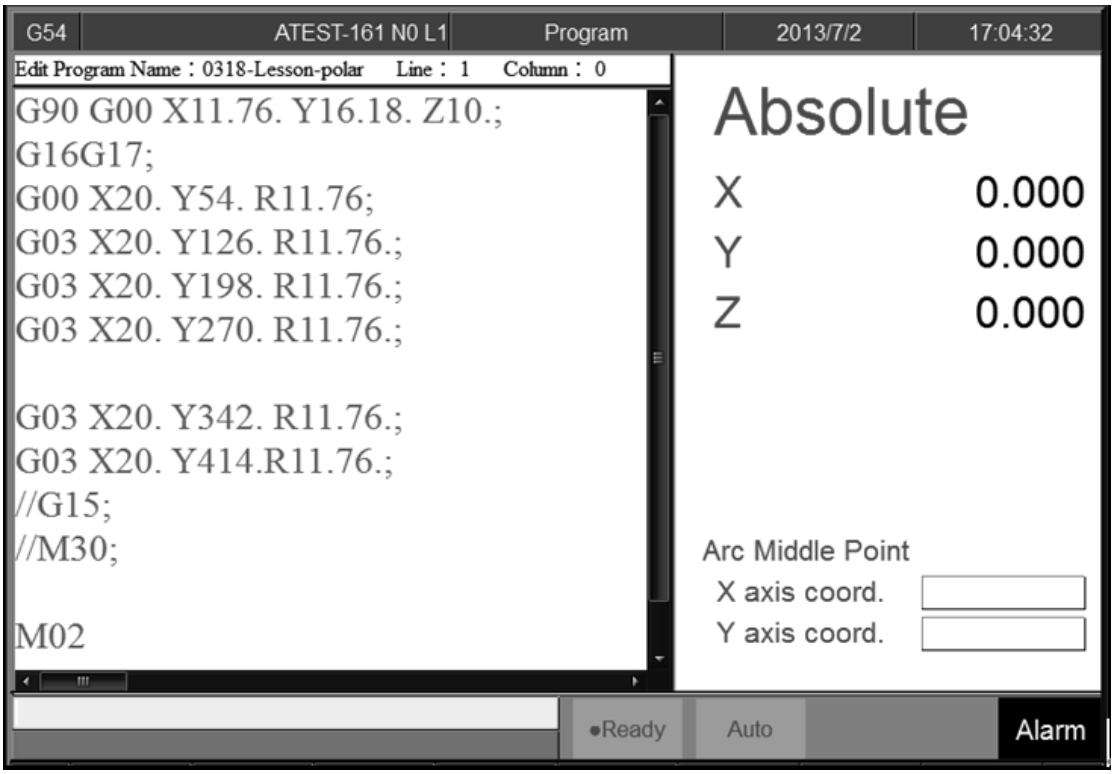
  ✧ If 「Block Copy」is use, the block that had been selected will not disappear.

- Note

  ✧ If 「Block Cut」is use, but the cutted programs do not paste, the program that had been cutted will disappear.

✧ 「Block Cut」can paste only one time and 「Block Copy」 can paste many time.

### 1.5.6 Teach




- Path
  - ✧ F2 Program→Next→F3 Teach
- Function
  - ✧ Use function 『MPG』/『JOG』/『INJOG』, move the machine to destination and use 『Teach』function, to teach the current absolute coordinate value to the NC program.
  - ✧ Omit the manual input problem.

#### 1.5.6.1 Rapid Teach

- Path
  - ✧ F2 Program→Next→F3 Teach→F1 Rapid Teach
- Function
  - ✧ Add the current absolute coordinate as the value of 「G00 Rapid Traverse」function in current program.

#### 1.5.6.2 Line Cut Teach

- Path
  - ✧ F2 Program→Next→F3 Teach→F2 Line Cut Teach
- Function
  - ✧ Add the current absolute coordinate as the value of 「G01 Linear Cutting」function in current program.

#### 1.5.6.3 Arc Cut Teach

- Path
  - ✧ F2 Program→Next→F3 Teach→F2 Arc Cut Teach
- Function
  - ✧ Add the current absolute coordinate as the machining 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 added in current cursor location of program.

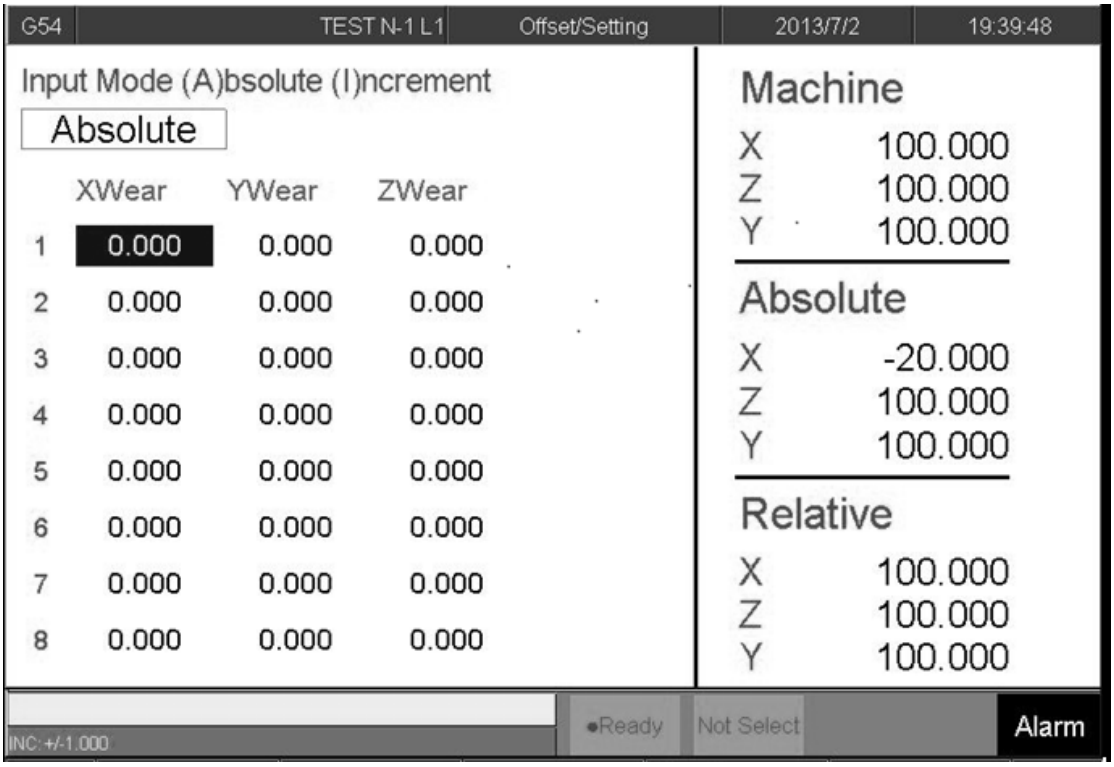
#### 1.5.6.4 Cancel Arc Middle

- Path
  - ✧ F2 Program→Next→F3 Teach→F4 Cancel Arc Middle
- Function
  - ✧ Clear the arc middle that had been set.
  - ✧ If the arc middle were not been set, this function will not enable.

#### 1.5.6.5 Point Teach

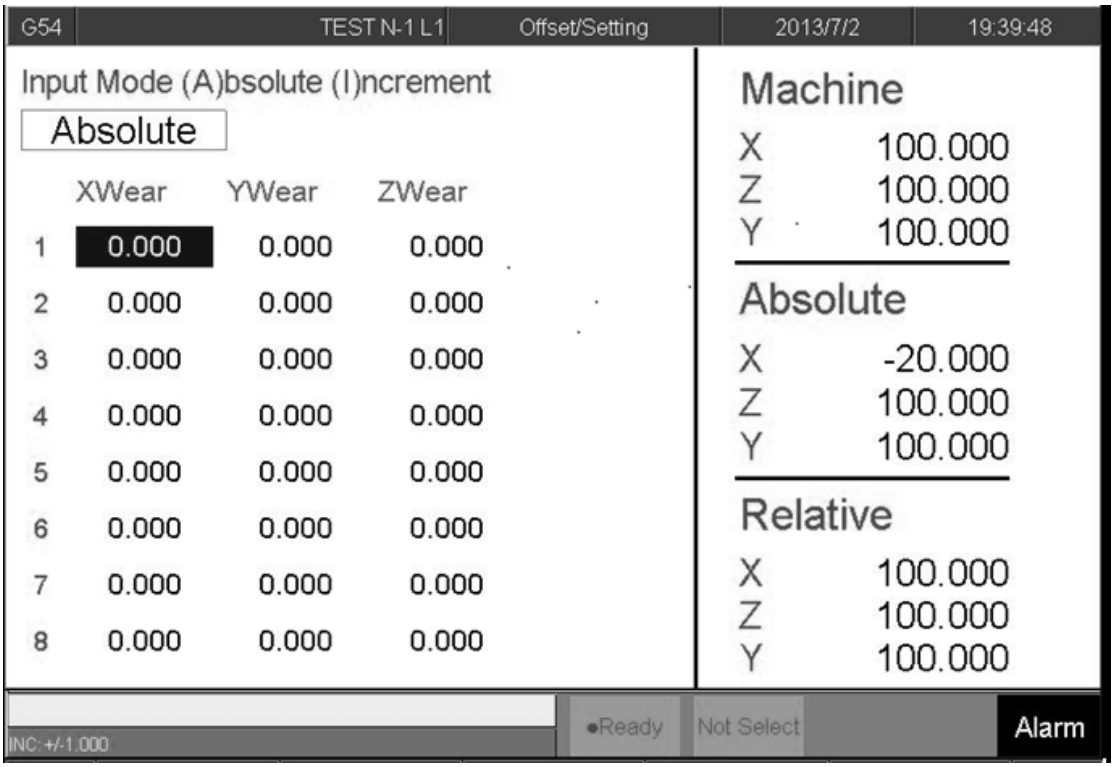
- Path
  - ✧ F2 Program→Next→F3 Teach→F5 Point Teach
- Function
  - ✧ Move the worktable to the arc center and press「Point Teach」, current absolute coordinate will be added in current cursor location of program.

## 1.6 Offset/Setting



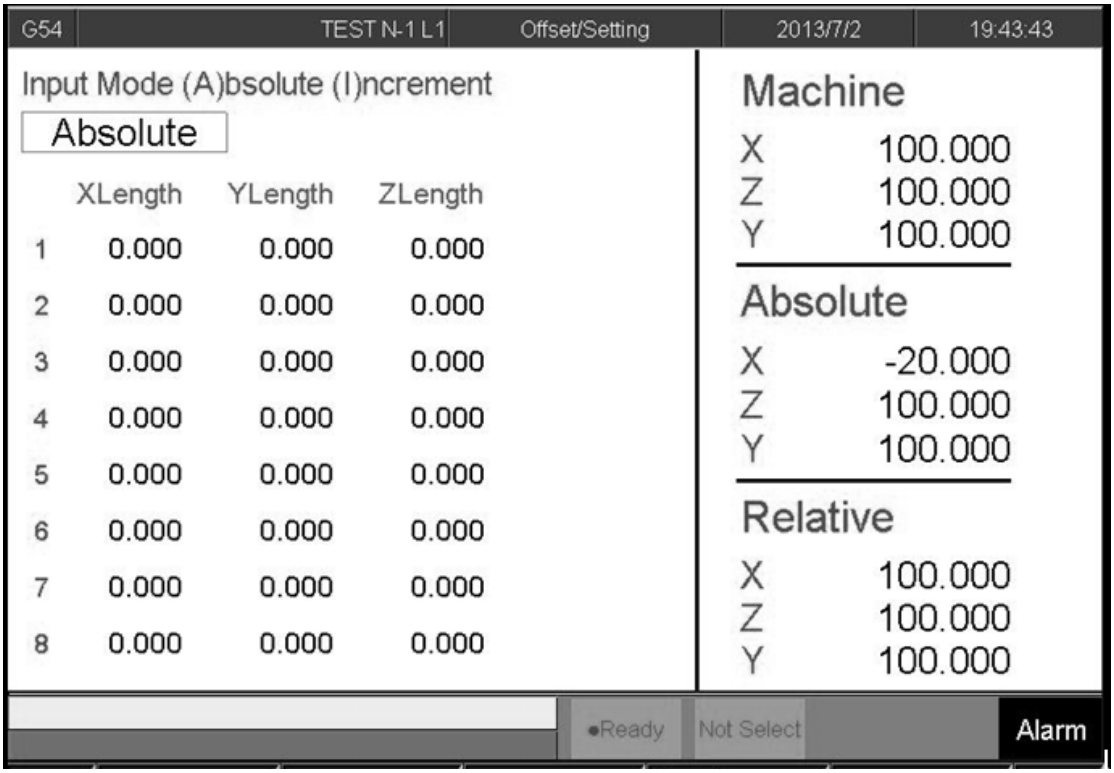

- Path
  - ✧ F3 Offset/Setting
- Function
  - ✧ Under this function group, user can do the offset/Setting.
  - ✧ Use the group function key 【Offset/Setting】can switch to the current page quickly.

### 1.6.1 Tool Wear Set



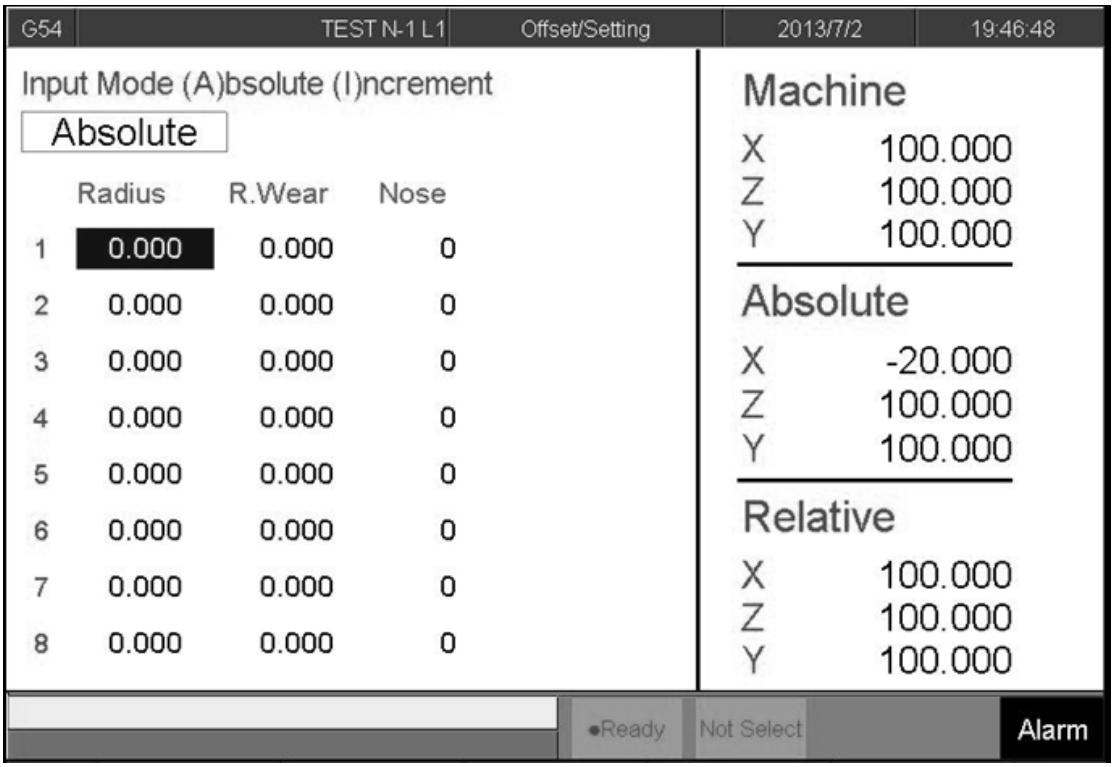

- Path
  - ✧ F3 Offset/Setting→F1 Tool Wear Set
- Function
  - ✧ Set the wear of tool.
  - ✧ Actual tool length=Tool length+Tool wear
- Function of parameter
  - ✧ Wear: Tiny modification of tool length.
- Note
  - ✧ When the tool length had been set, related tool wear will become zero.
  - ✧ If the tool wear is set under the machining condition, new tool wear setting will be effective in next compensation time.

## 1.6.2 Tool length Set




- Path
  - ✧ F3 Offset/Setting→F2 Tool Length Set
- Function
  - ✧ Set the length of tool.
  - ✧ Actual tool length=Tool length+Tool wear
- Function of parameter
  - ✧ Tool length: G43/G44 tool length compensation setting.
- Note
  - ✧ When the tool length had been set, related tool wear will become zero.
  - ✧ Setting is prohibited in machining condition.

### 1.6.3 Tool Nose Set

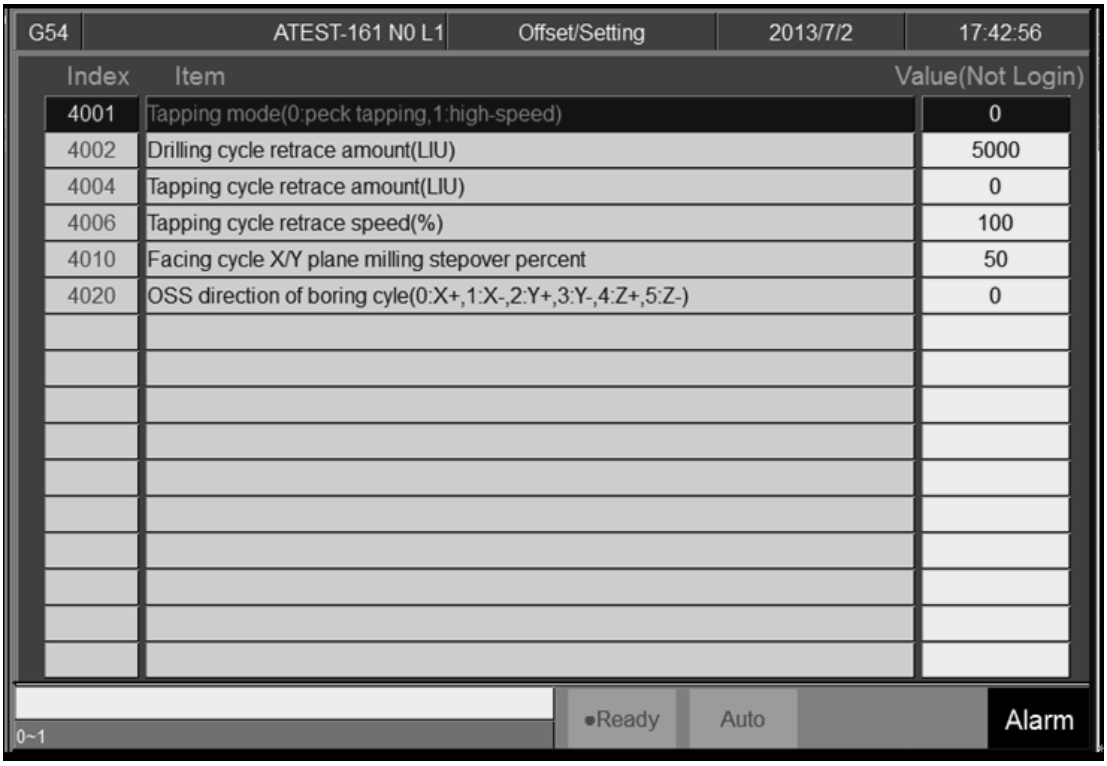



- Path
  - ✧ F3 Offset/Setting→F3 Tool Nose Set
- Function
  - ✧ Set the top position of Tool .
  - ✧ Real Tool Nose= Tool Nose Radius+Tool Nose Radius wear
- Function of parameter
  - ✧ Tool Nose Radius: G41/G42 Tool Nose Radius compensation (not diameter).
  - ✧ Tool Nose Radius Wear : G41/G42 Tiny modification of Tool Nose Radius.
  - ✧ Tool Nose direction: Set the machining direction of Tool Nose.

- ✧ Syntec provide total 8 different kind of Tool Nose direction option, see 《Syntec Lathe Programming Manual—G41/G42》。

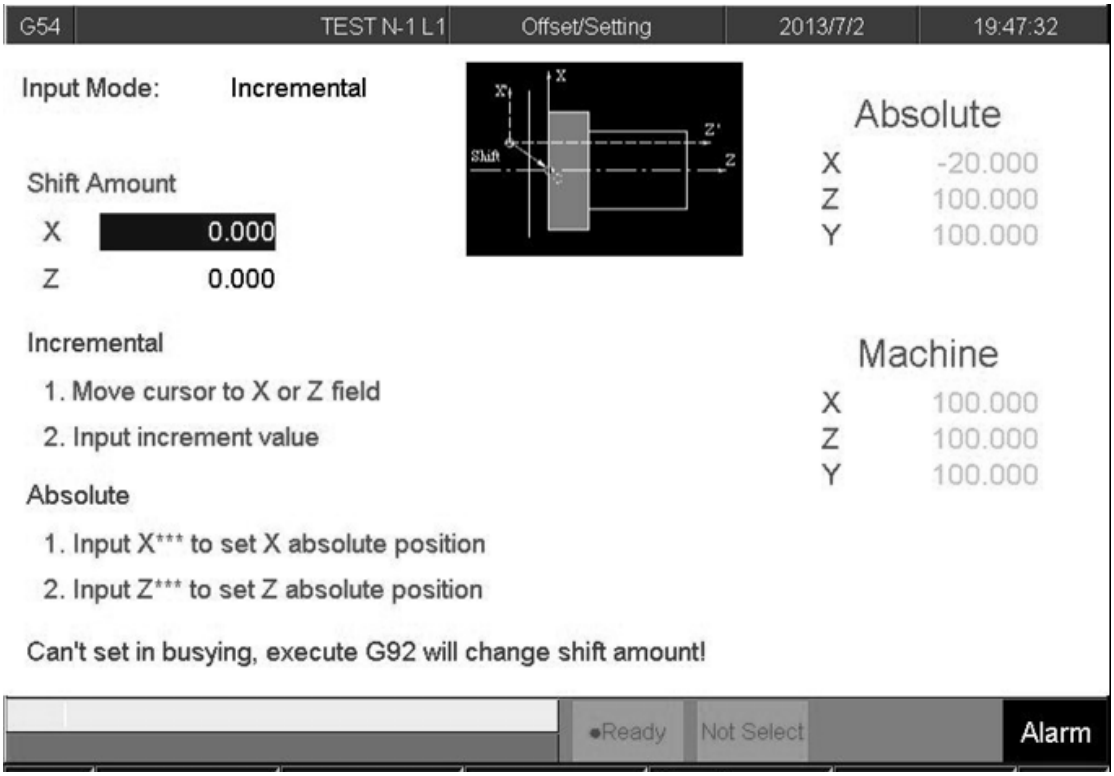
- Note
  - ✧ Setting is prohibited in machining condition.

## 1.6.4 User Parameter Setting




- Path
  - ✧ F3 Offset/Setting→F4 User Parameter Setting
- Function
  - ✧ Syntec controller provided user to set the related machining parameter.
- Function of parameter
  - ✧ Detail explanation can be seen in 「Lathe Parameter Manual」.

## 1.6.5 Working Shift




- Path
  - ✧ F3 Offset/Setting→F5 Working Shift
- Function
  - ✧ Set G92 coordinate offset.
- Operation Method

  ■Absolute Input

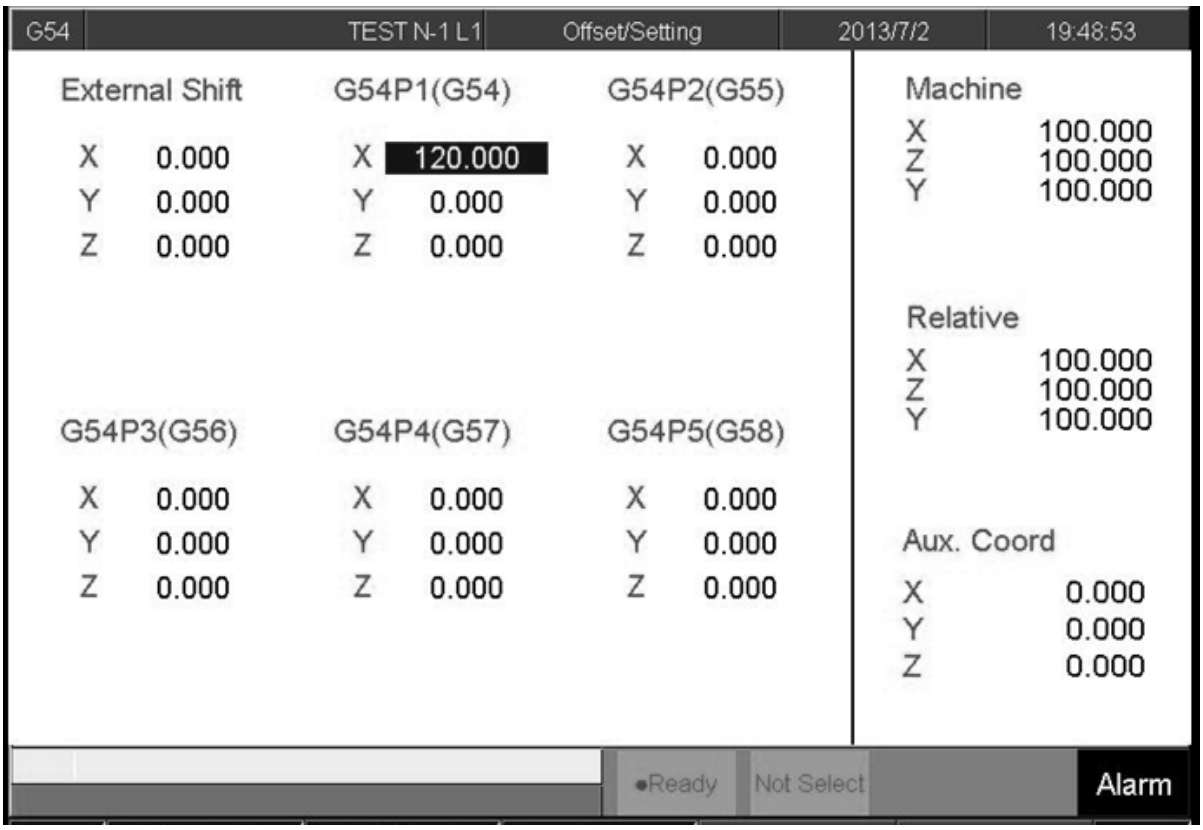
  ✧ Key in 【X】/【Z】and then key in value, press 【ENTER】.

  ■Increment Input

  ✧ Move the cursor to the coordinate that want to setting.

  ✧ Key in value and press 【ENTER】.

### 1.6.6 Work Piece Cord.




- Path
  - ✧ F3 Offset/Setting→Next→F1 Work Piece Cord.
- Function
  - ✧ This function key switch the screen to the 『Work Piece Cord』setting page.
  - ✧ If user do not define the coordinate system as G54~G59.10, the default system is G54.
  - ✧ 『External Shift』: All setting of 『External Shift』 will be effective in G54~G59.10.
- Operation Method
  - ✧ Use the key on panel【↑】【↓】to move the cursor.
  - ✧ Use【Page Up】【Page Down】to switch page.

- Note
  - ✧ After Setting 『Work Piece Cord』, Tool Length compensation need to re-setting again.

### 1.6.6.1 Mach. Coord. Teach

- Path
  - ✧ F3 Offset/Setting→Next→F1 Work Piece Cord.→F1 Mach. Coord. Teach
- Function
  - ✧ By the absolute input method, set the current cursor located work piece coordinate value as the mechanical coordinate value.
- Operation Method
  - ✧ Move the worktable to the destination.
  - ✧ Move the cursor to the relate work piece coordinate and press 「Mach. Coord. Teach」
  - ✧ Current cursor located work piece coordinate will be replaced by the mechanical coordinate.
- Example
  - ✧ Current mechanical coordinate of X axis is 5.000
  - ✧ Current workpiece coordinate G54 of X axis is 0.000
  - ✧ Move the cursor to G54 X axis.
  - ✧ Press「Mach. Coord. Teach」,the coordinate of G54 X axis become 5.000

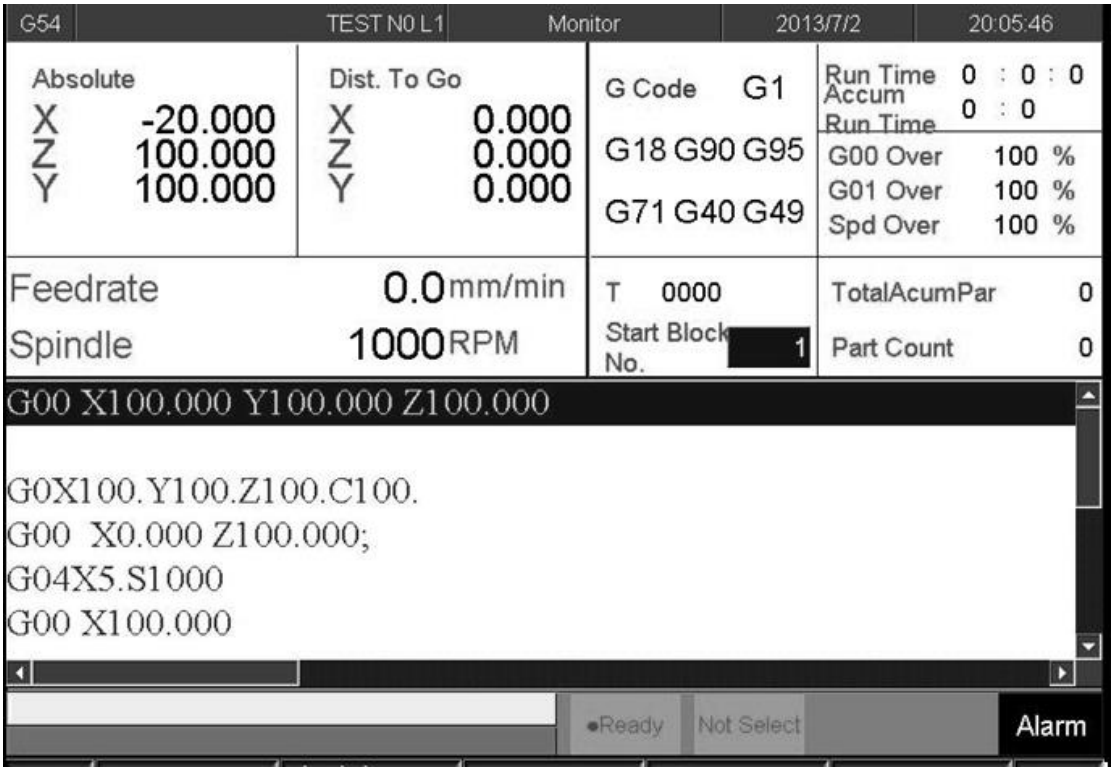
### 1.6.6.2 Mach. Coord Inc. Teach

- path
  - ✧ F3 Offset/Setting→Next→F1 Work Piece Cord.→F2 Mach. Coord Inc. Teach
- Function
  - ✧ Current cursor located work piece coordinate will be replaced by the 「Mechanical coordinate 」plus 「Mach. Coord Inc. Teach」 value.
- Operation Method
  - ✧ Move the worktable to the destination.
  - ✧ Move the cursor to the relate work piece coordinate, key in teach value. And then press 「Mach. Coord Inc. Teach」
  - ✧ Current cursor located work piece coordinate will be replaced by the mechanical coordinate plus Increment value.
- Example
  - ✧ Current mechanical coordinate of X axis is 5.000
  - ✧ Current workpiece coordinate G54 of X axis is 0.000
  - ✧ Move the cursor to G54 X axis.
  - ✧ Key in 10.
  - ✧ Press 「Mach. Coord Inc. Teach」,the coordinate of G54 X axis become 15.000

### 1.6.6.3 Inc. Input

- Path
  - ✧ F3 Offset/Setting→Next→F1 Work Piece Cord.→F3 Inc. Input
- Function
  - ✧ Current cursor located work piece coordinate will be replaced by current cursor value plus teach value.
- Operation Method
  - ✧ Move the worktable to the destination.
  - ✧ Move the cursor to the relate work piece coordinate, key in teach value. And then press 「Inc. Input 」function key.
  - ✧ Current cursor located work piece coordinate will be replaced by the current coordinate value plus teach value.
- Example
  - ✧ Current G54 X axis value is 5.000
  - ✧ Move the cursor to G54 X axis.
  - ✧ Key in 10.
  - ✧ Press 「Inc. Input 」function key ,the coordinate of G54 X axis become 15.000

## 1.7 Monitor




- Path:
  - ✧ F4 Monitor
- Description
  - ✧ This function key provides all necessary required information in machining process.

- Menu description

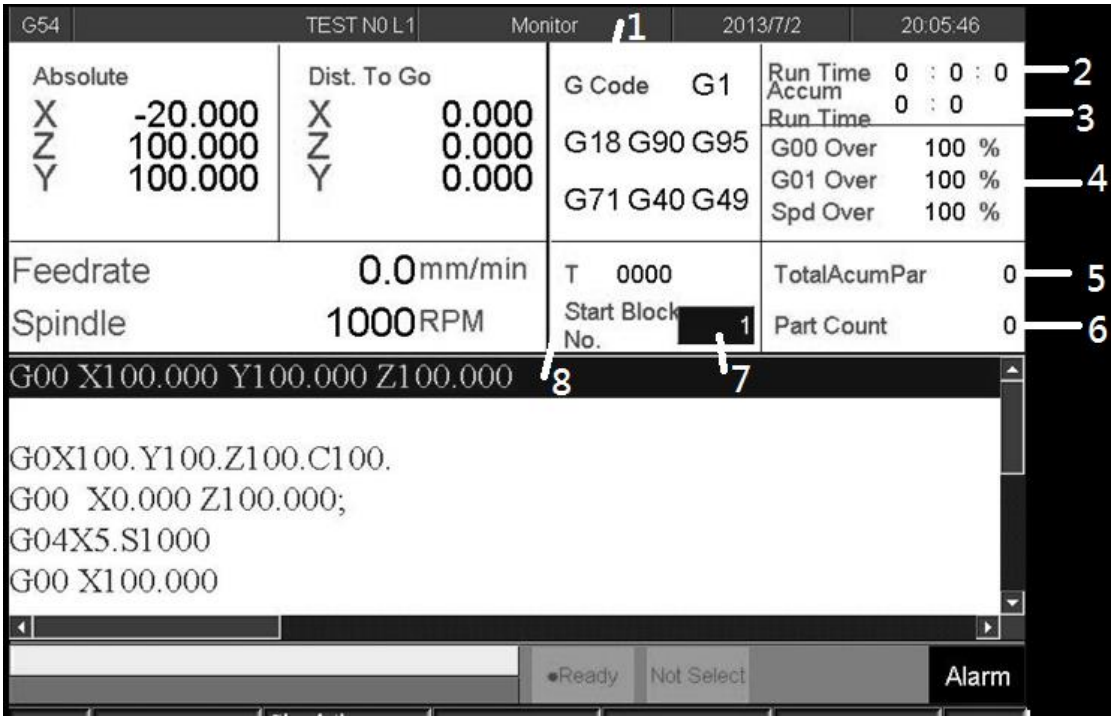
**1. Machine monitor area**

- This area displays current machining data
- Absolute coordinate
- Dist. To go
- Feedrate
- Spindle speed,

**2. Program monitor area**

- Display current machining program
- The yellow single block indicates the current machined one

### 1.7.1 Monitor area of machining information




- Description
  - ✧ This area and machining setting are overlapped, users can press ”F4-parameter set” to exchange between those two displays
    1. G-code status
       - ◆ Display the current G-code that system are running.
    2. Run time
       - ◆ Display machining time of one workpiece.
       - ◆ When machining program is started, it will start to count.
    3. Accum run time
       - ◆ Display the total machining time
    4. Override

- G00 Overide
- G01 override
- Spindle override

5. Total AcumPar
   - Total workpieces that CNC machined
   - The system can not automatically reset this value to zero
   - If users want to reset manually, please switch to the "machining setting area" and set Total AcumPar equal to 0
6. Part count
   - It will is reset to 0 when machining other machining program
   - Display the No. workpieces machined currently

7. Start block
   - Users can set the starting single block in machining program.

● Operation:

- n: Specify the starting single block is the $n^{th}$ row in program. (Ex: 20)
- L+ n: Specify the starting single block is the $n^{th}$ row in program. (Ex: L20)

- N+ n: Search the single block that has N+n sign and specify that single block as the starting line. (Ex: N3)
- T+ n: Search the single block that has T+n sign and specify that single block as the starting line.. (Ex: T01)
- If users specify the number of single block exceeding the maximum row number of program, system will set the last single block as default value

- Machining tool information

8. T

- 4 codes displayed.
- First 2 codes is the tool number currently machining
- 2 subsequent codes specify the tool number executing compensation.

### 1.7.2 Display area of machining setting

- Description
  - ✧ This area and machining information are overlapped, users can press ”F4-parameter set” to exchange between those two display.
  - ✧ Display description
  - ✧ Interrupt Line No.
  - ✧ Display the line number interupted at last time (N)
  - ✧ Interupt Colum No.
  - ✧ Display the Colum No. interrupted at last time (N)
  - ✧ Spindle speed
  - ✧ Set the speed of spindle
  - ✧ Can be set when system is on busy status. Moreover, it will be enalbled immediately
- Feedrate
  - ✧ Set the speed of the feed.。
  - ✧ Can be set when system is on busy status
  - ✧ This value will be updated after completely excuting current single block
- Total AcumPar
  - ✧ Total workpieces that CNC machined

    The system can not automatically reset this value to zero
- Part count
  - ✧ It will is reset to 0 when machining other machining program

- ✧ Display the No. workpieces machined currently
- ✧ When CNC executes one M code defined by parameter 3804-part count M code, part count would be added 1 automatically and run time will be reseted to 0. When part count is reached, system will change to pause status.

- Required part
  - ✧ Set number of workpieces wants to machine
  - ✧ Once the number of required part is reached, an alarm will be issued and system will change to pause status.

### 1.7.3 Simulation graph

- Description
  - ✧ Display the tool trajectory of current machining program.
  - ✧ Related setting, please see "F2-program→ F5-simulaiton→ F5- Simu. Setting."
  - ✧ Use "F2-simulation switch" to change display content

#### 1.7.3.1 **Open file to edit**

- Path:
  - ✧ F4 monitor→F1 Open file to edit
- Description
  - ✧ Enter and edit the current machining program, also switch to “F2-program” interface
  - ✧ Note: once system is on busy status, user cannot edit current maching program

#### 1.7.3.2 **Simulation switch**

- Path:
  - ✧ F4 monitor→F2 Simulation switch
- Description
  - ✧ Check whether simulation function works properly
  - ✧ Only modifying simulation properties when entering simulation setting interface.

#### 1.7.3.3 **MDI input**

- Path:
  - ✧ 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 to return referent point (HOME)
  - ✧ Press F3 -MDI input and type in G or M code.

- ✧ Press F1 (OK) to confirm the input command.
- ✧ The typed-in command line will show on right upper corner of the screen.
- ✧ Press 【CYCLE START】on the operator panel to execute the single-line command.

- Note:
  - ✧ must select MDI mode first

#### 1.7.3.4 **Parameter set**

- Path:
  - ✧ F4 monitor→F4 Parameter set
- Description
  - ✧ Switching between “machining setting” and “machining information”

#### 1.7.3.5 **Tool wear set**

- Path:
  - ✧ F4 monitor→F5 Tool wear set
- Description
  - ✧ Display the tool wear setting interface, user can setting tool wear here.

#### 1.7.3.6 **Work record**

- Path:
  - ✧ F4 monitor→ Next→F2 work record
- Description
  - ✧ Check current machining record, can export to external storage devices

### 1.7.3.7 Clear Acum Cycle time

- Path:
  - ✧ F4 monitor→ Next→F3 Clear Acum Cycle time
- Description
  - ✧ Clear the accumulative time

### 1.7.3.8 **Graph adjust**

- Path:
  - ✧ F4 monitor→ Next→F4 Graph adjust
- Description
  - ✧ Zoom in/out simulation graph
- Operation
  - ✧ See “Zoom ” in “simulation”

## 1.8 Maintain

- Path:
  - ✧ F5 Maintain
- Description
  - ✧ This function key displays alarm, network setting, fast diagnostic, PLC param setting, system setting

### 1.8.1 Alarm


- Path:
  - ✧ F5 Maintain→F1 Alarm
- Description
  - ✧ Whenever the system or the program stops due to errors, alarm messages will be shown on the screen

#### 1.8.1.1 **Pending alarm**

- Path:
  - ✧ F5 Maintain→F1 Alarm→ F1 Pending Alarm
- Description
  - ✧ Display system alarm content at present

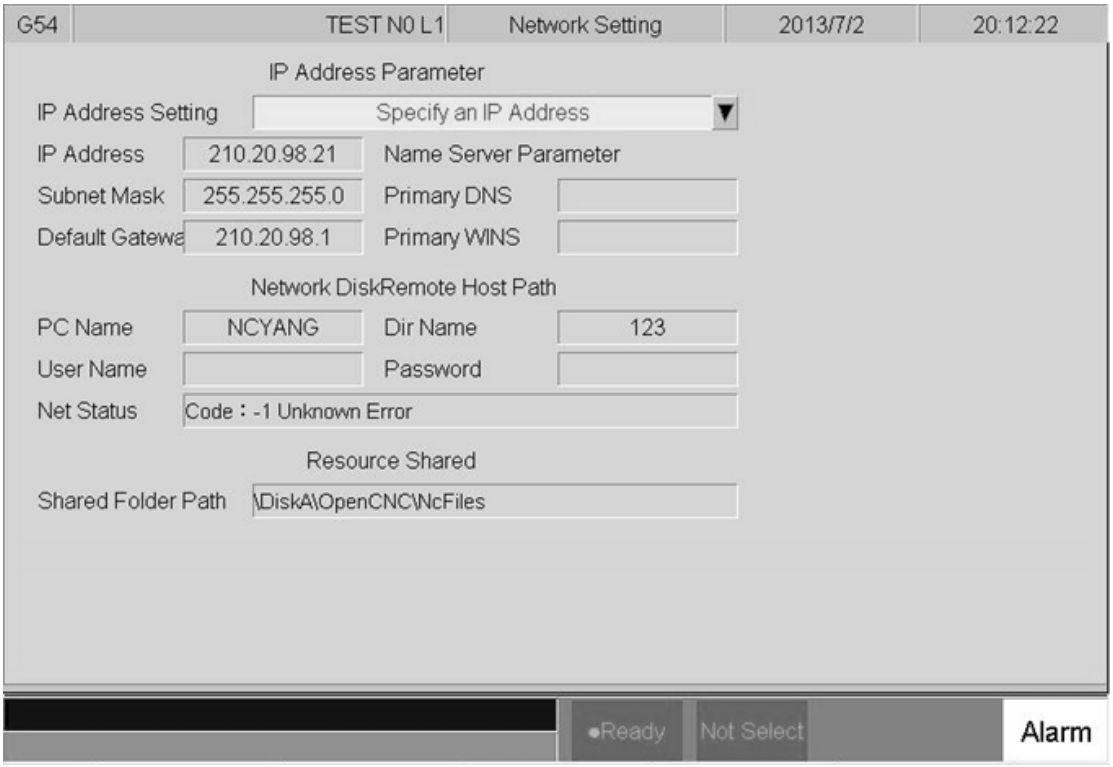
#### 1.8.1.2 **History alarm**

- Path:
  - ✧ F5-Maintain→F1 Alarm→ F2 History Alarm
- Description
  - ✧ Show all the alarm history of the system.
- Note:
  - ✧ Some alarm is not displayed here, ex: MACRO alarm

#### 1.8.1.3 **Save Alarm**

- Path:
  - ✧ F5 Maintain→F1 Alarm→ F3 Save alarm
- Description
  - ✧ Save Alarm History to external disk.
  - ✧ Default export file name:
  - ✧ Actual alrm: actalm.txt.
  - ✧ History: histalm.txt.

## 1.8.2 Network setting

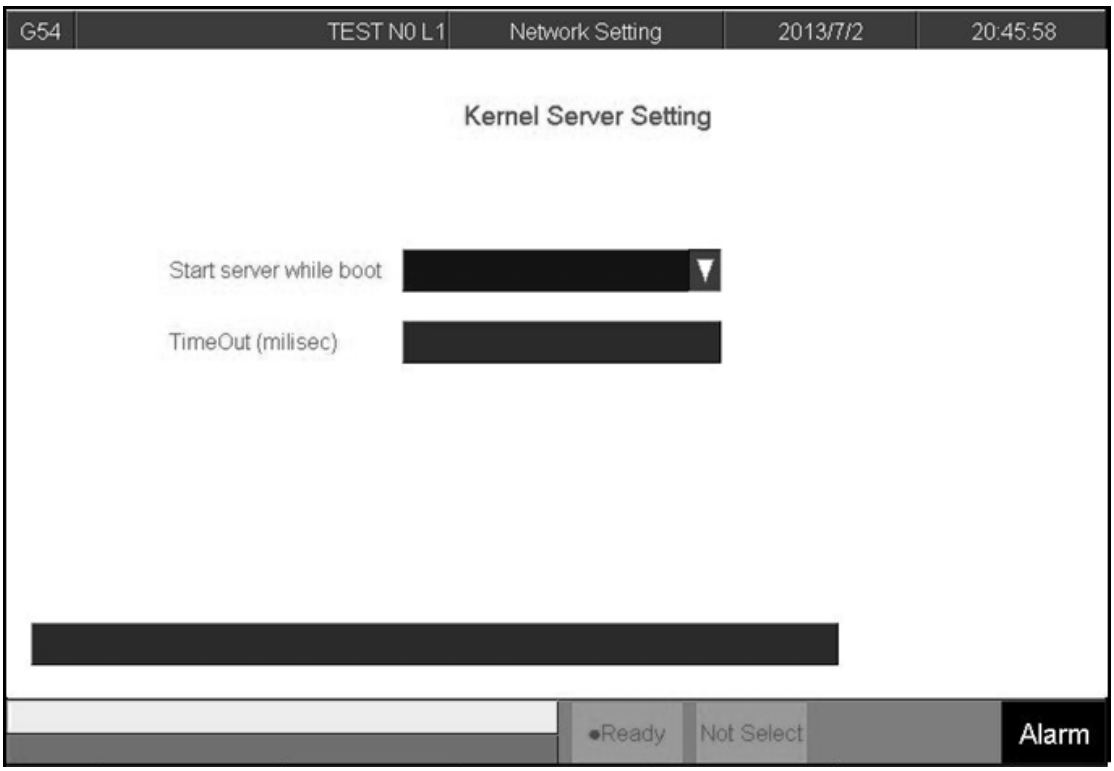



- Path:
  - ✧ F5 Maintain→F2 Network setting
- Description
  - ✧ Set system network setting
- Related infor.
  - ✧ IP address setting
  - ✧ Network cable(with HUB), select ”Obtain an IP address automatically”
  - ✧ Jumper cable(without HUB), select “use the following IP address” and enter IP address(the forth number is different from controller setting) and Subnet mask(same with controller setting)
- IP address
  - ✧ Only used for “use the following IP address” option.
- Sunet mask

- ✧ Enter the IP address for subnet mask (the same with PC subnet mask).

- PC name
  - ✧ Enter the full computer name of your PC.
  - ✧ Need to be the same with PC
- Dir name
  - ✧ Enter the sharing folder name (the same name with PC sharing folder )
  - ✧ User name and password
  - ✧ If the shared folder is not set the password, user do not need to enter user name, if yes, please enter the same password.

### 1.8.2.1 Set Kernel Server




- ✧ Path:
- ✧ F5 Maintain→F2 Network Setting→ F5 Set Kernel Server

- Description
  - ✧ Setting function related to kernel server
  - ✧ Related infor.
  - ✧ Start server whie boot
  - ✧ Setting whether server is started when booting
  - ✧ Timeout(Milisec)
  - ✧ Set the acceptable time out when connecting to the Kernel server unsuccessfully

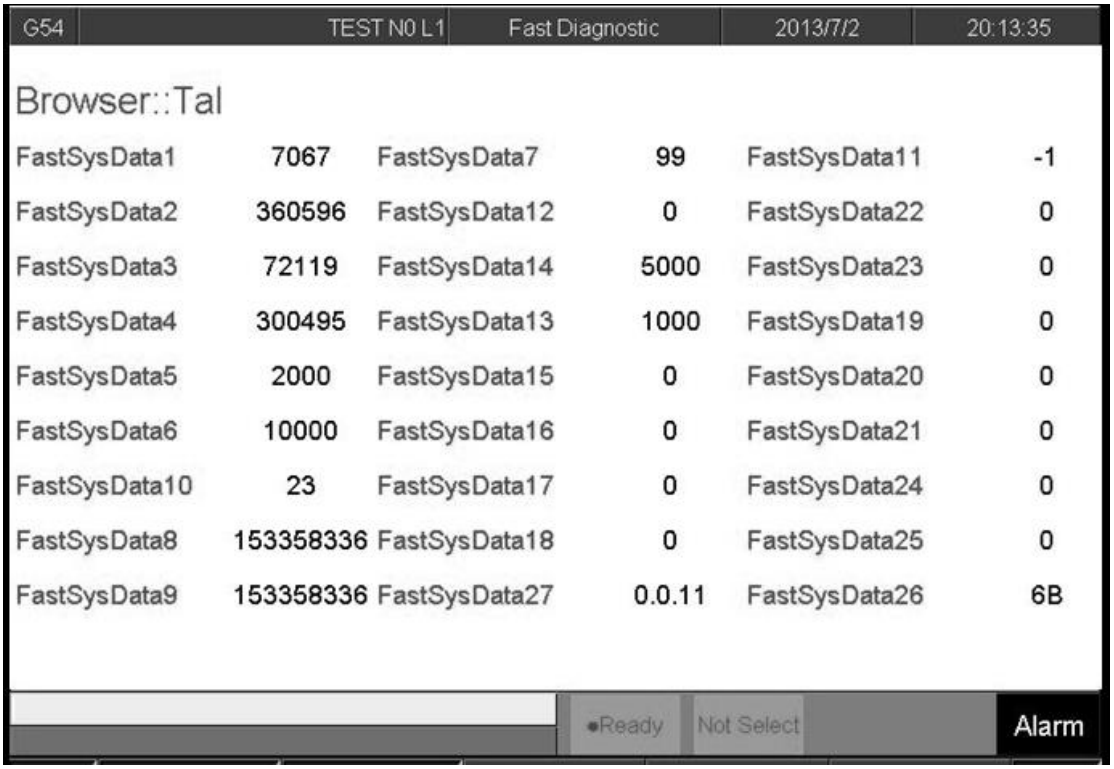
#### 1.8.2.1.1 Start server

- Path:

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

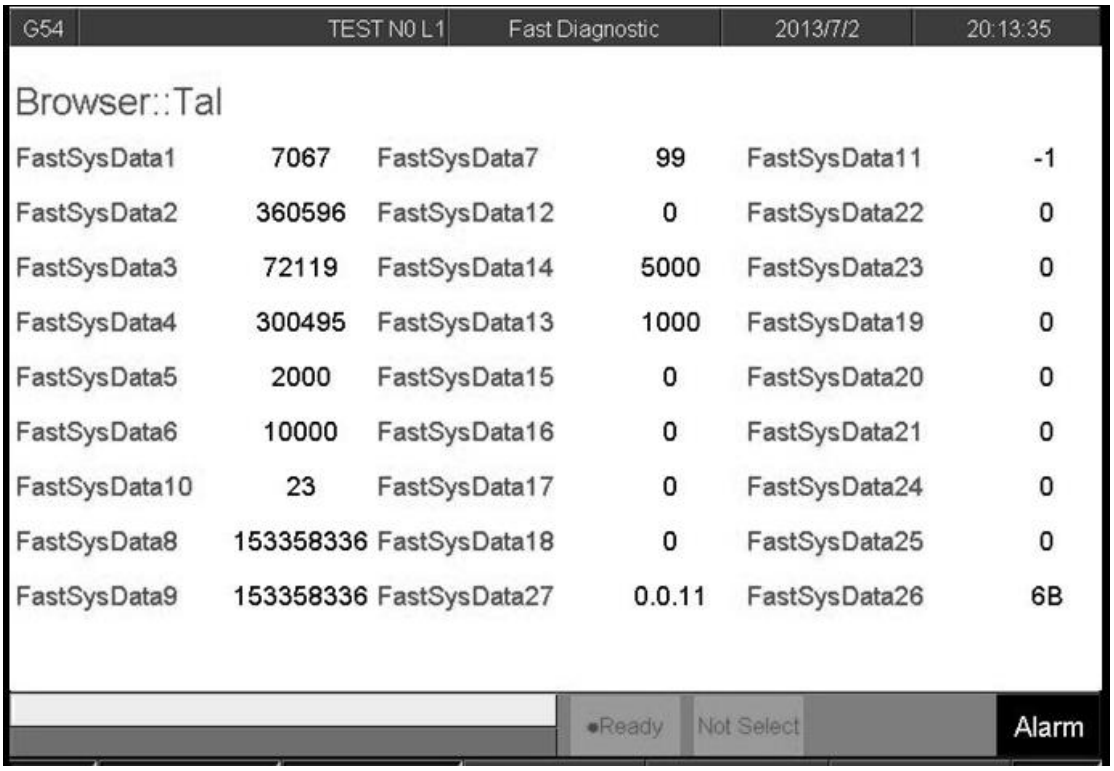
- Description
  - ✧ Start server immediately

### 1.8.3 Fast Diagnostic



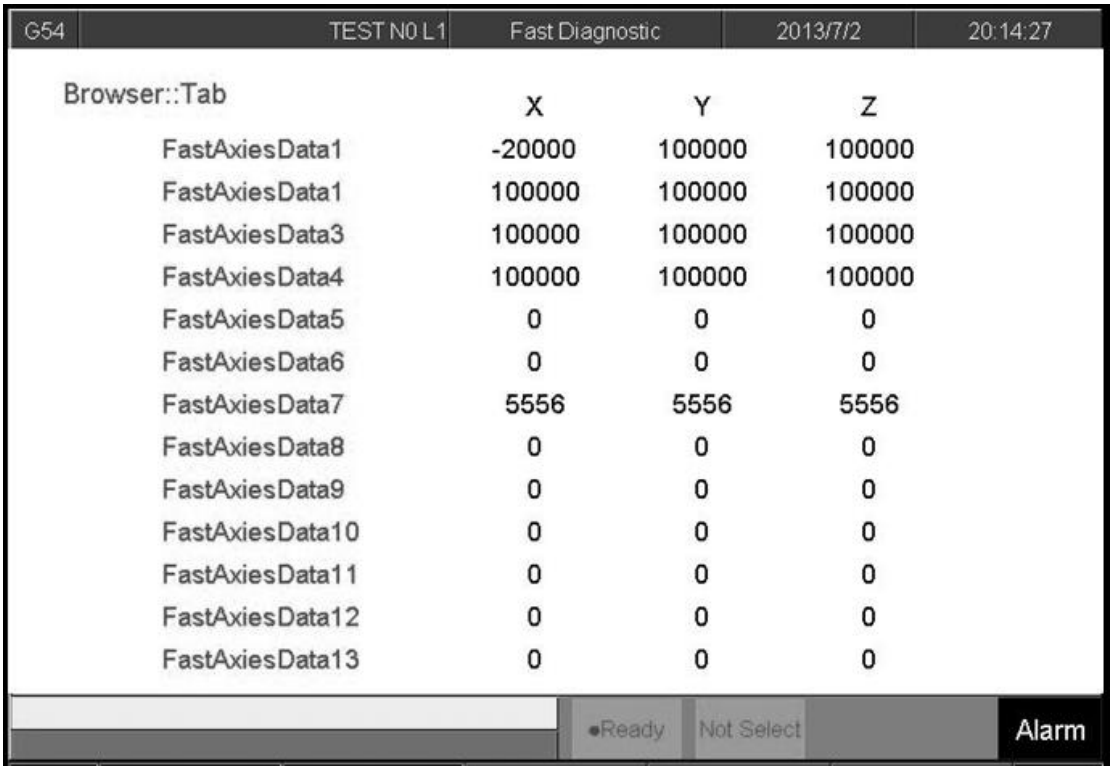

- Path:
  - ✧ F5 Maintain→F3 Fast diagnostic
- Description
  - ✧ Display simple diagnostic information of system and axies

### 1.8.3.1 System data



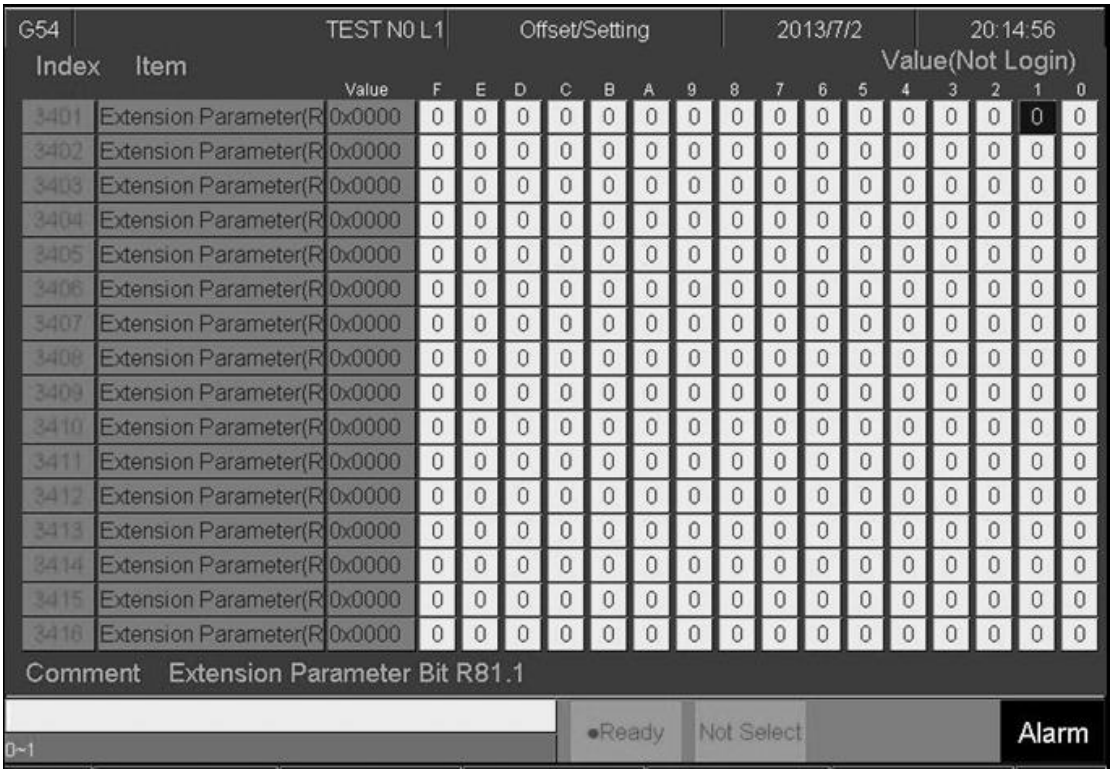

- Path:
  - ✧ F5 Maintain→F3 Fast diagnostic→ F1 System Data
- Description
  - ✧ Display simple diagnostic information of system

### 1.8.3.2 **Axies data**




- Path:
  - ✧ F5 Maintain→F3 Fast diagnostic→ F2 Axies data
- Description
  - ✧ Display simple diagnostic information of Axies

## 1.8.4 PLC param setting



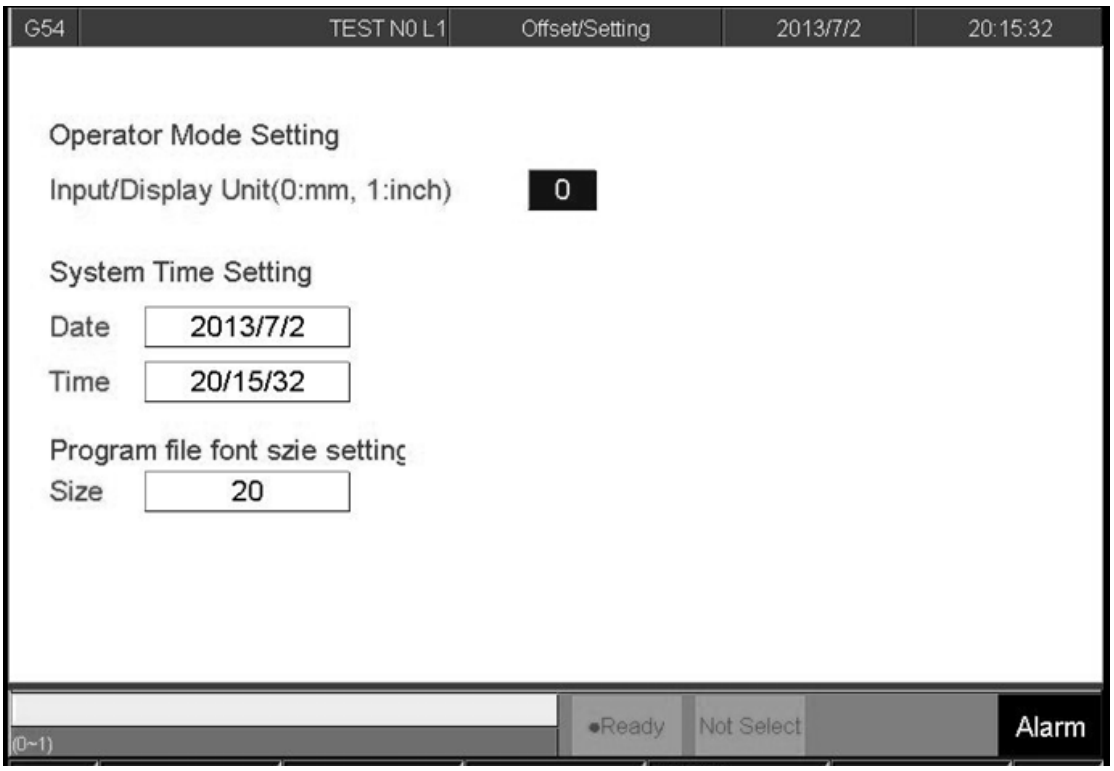

- Path:
  - ✧ F5 Maintain→ F4 PLC param setting
- Description
  - ✧ SYNTEC's controller provides R81 ~ R100, totally 20 sets of registers for machinerymanufacture use, with 16Bits of each register.
  - ✧ Machinery manufacture can use these 20 sets of registers to provide user execute control flag in PLC specific functions.
- Operation
  - ✧ Use 【↑】【↓】【←】【→】 to move cursor and 【PageUp】【PageDown】to switch to next page
  - ✧ Only can enter [0] or [1].
  - ✧ Available some bits for comment

- ✧ Corresponding file name for comment:

  ParamExt_RBit_(L).xml

- ✧ (L)=COM/CHT/CHS/language

## 1.8.5 System setting




- Path:
  - ✧ F5 Maintain → F5 System setting
- Description
  - ✧ This part is used to set system environment
- Operation
  - ✧ Use 【↑】【↓】【←】【→】 to move cursor and 【PageUp】【PageDown】to switch to next page

### 1.8.5.1 Operator mode setting

- ✧ Setting system unit
- ✧ 0：mm
- ✧ 1：Inch
- ✧ Note: reboot to enable setting

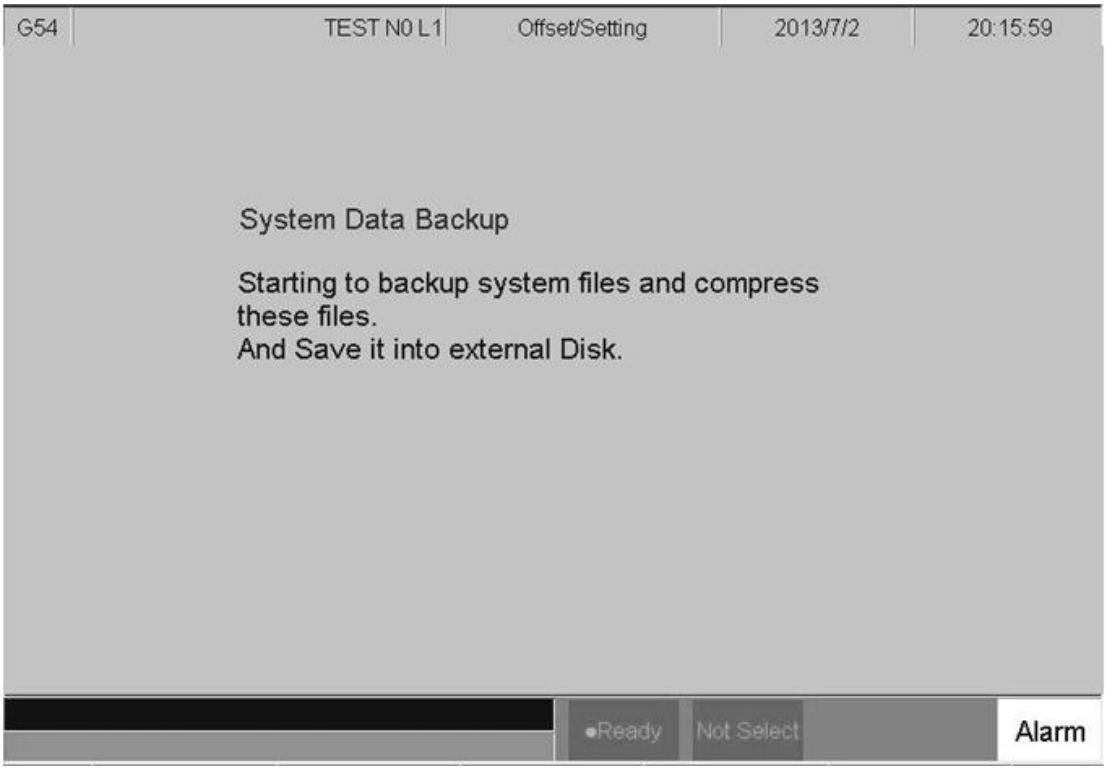
### 1.8.5.2 System time setting

- ✧ Date: input format YYYY/MM/DD
- ✧ YYYY: year
- ✧ MM: month
- ✧ DD: day
- ✧ Time: input format HH/MM/SS
- ✧ HH: hours
- ✧ MM: minute
- ✧ SS: second

### 1.8.5.3 Program file font size setting

- ✧ Setting display of font size in program
- ✧ Note: reboot to enable setting

## 1.8.6 Backup system




- Path:
  - ✧ F5 Maintain → Next→ F1 Backup system
- Description
  - ✧ This area executes backup system data.

### 1.8.7 About

- Path:
  - ✧ 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 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** 

- Description:
  - ◆ When CNC power is on, please implement home search
- Operation:
  - ◆ Select HOME mode
  - ◆ Press axis manual key X+ or X-,Y+ or Y-,Z+ or Z- or press cycle start
  - ◆ CNC would start to return reference position (HOME)

■ **JOG mode (Rapid JOG)** 

- Operation:
  - ◆ Select JOG mode

- ◆ Press X+,X-,Y+,Y-,Z+,Z- key, machine table will move according to axis name and direction
- ◆ Operators can use JOG% or G01% to adjust Jog feedrate
- ◆ When operator press both above key and rapid Key "⁓" at the same time, CNC will move the machine table with "RAPID speed".

■ **Incremental JOG (IN JOG)** 

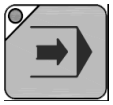
- ● Operation:
  - ◆ Select INC JOG mode
  - ◆ Press X+,X-,Y+,Y-,Z+,Z- key, moving distance of machine table equal to the [incremental distance × pressing number of operator]
  - ◆ Operators can set the incremental distance by rotary switch

    *1 : 1um ,*10 : 10um ,*100 : 100um

■ **MPG JOG** 

- ● Description:
  - ◆ User can use MPG (Manual Pulse Generator) mode to move the machine table
- ● Operation:
  - ◆ Select MPG mode

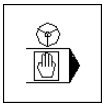
- Select incremental rate
- Select corresponding axis X, Y, Z by rotary switch on MPG device, rotate MPG, machine table will move with velocity according to rotation speed of MPG device.

■ **AUTO mode** 

- Description:
  - Users use this function to execute machining NC file
- Operation:
  - Select AUTO mode after returning reference point (HOME)
  - Set workpiece coor.(G54..G59), CNC will default G54 in case user do not set any workpiece coor.
  - Excute Tool setting by setting tool wear, tool length, and tool nose
  - Press “CYCLE START” key to run the NC program.
  - Press “Feedhold” key to feedhold the NC program if necessary

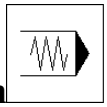
■ **MDI mode** 

- Description**:**
  - Users use this function to execute a block without NC file
- Operation:
  - Select MDI mode
  - MDI only is enabled after machine returns reference point (HOME)
  - Select F4”Monitor”
  - Press F3 “MDI Input”, another interface screen would appear to input program.
  - Press OK key after finishing input program
  - Press “CYCLE START” key to execute the MDI block.
  - If MDI block SYNTAX is incorrect, data in MDI menu will disappear

## ■ MPG Simulation 

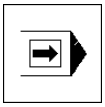
- Description:
  - Users can use this function to check NC file
- Operation:
  - Select AUTO mode
  - Press MPG simulation button, and the corresponding led will be “ON”
  - Press “CYCLE START” key to start running NC file.

- ◆ CNC would change machine status from “READY” to “BUSY”
- ◆ Machine does not move
- ◆ Operator can rotate MPG to run NC file
- ◆ The faster MPG rotates, the faster machining speed is
- ◆ MPG stops, CNC stops too.
- ◆ This function can be “Enable” “ Disable” immediately
- ◆ P.S. This function is easy to use for checking purpose

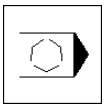
■ **Dry Run** 

- ● Description:
  - ◆ User can use this function to check NC file
- ● Operation:
  - ◆ Select AUTO mode
  - ◆ Press Dry run button, and the corresponding led will be “ON”
  - ◆ Press “CYCLE START” key to start running the NC file.
  - ◆ CNC will change machine status from “READY” to “BUSY”

- This function can be “Enable” “ Disable” immediately

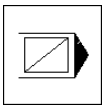
## ■ Single block 

- Description:
  - User can use this function to check NC file
- Operation:
  - Select AUTO mode
  - Press single block button, and corresponding led will be “ON”
  - Press “CYCLE START” key to start running the NC file
  - CNC will execute NC file only one block and STOP
  - CNC will change machine status from “BUSY ” to “B_STOP”
  - Press “CYCLE START” again ,then CNC execute next block
  - This function is used to check NC file by checking each Block of NC file.

## ■ Optional Stop 

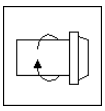
- Description:
  - Users can use this function to decide NC file M01 is STOP or not
- Operation:
  - Select AUTO mode

- Press Optional stop button, and corresponding led will be “ON”
- Press “CYCLE START” key to run the NC file.
- When CNC run “M01” code, CNC would STOP
- CNC would change machine status from “BUSY ” to “Feedhold”
- This function is used to change tool or check workpiece.

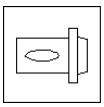
■ **Optional Skip** 

- Description:
  - Users can use this function to decide whether program skips or not when program is run to ‘/” sign in NC file.
- Operation:
  - Select AUTO mode
  - Press Optional skip button, and corresponding led will be “ON”
  - Press “CYCLE START” key to run the NC file.
  - When CNC execute to “/” sign in program ,CNC would Skip this block
  - If this key is not pressed, CNC will execute this block

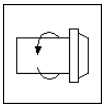
● **Spindle control**



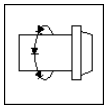
Spindle CW rotation



Spindle stop



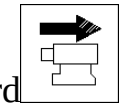
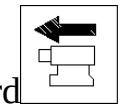
Spindle CCW rotation

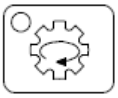


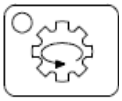
Low spindle speed: When spindle rotates, if this button is pressed, spindle will rotate with low speed

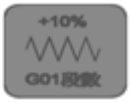
- **Spindle speed increases of 10%** 
- **Spindle speed decreases of 10%** 

- Working led & Blower
  - **Working led ON/OFF** 
  - **Blower ON/OFF** 

- Working Liquid
  - Flush working liquid ON/OFF 
    - Aux machine table forward 
    - Aux machine table backward 
- Chip Removal Device
    - Positive rotation of chip removal device 
  - Negative rotation of chip removal device 
- Tool magazine control


Tool magazine CW


Tool magazine CCW

- G00&G01 override switch
  - Increase G00 speed. 
  - Decrease G00 speed. 
  - Increase G01speed. 
  - Decrease G01speed. 

## 2.2 Text key description

A ~ Z ： 26 English character keys

0 ~ 9 ： Numerical keys


: Delete key


: Insert /replace key


: Switch keyboard to use smaller key on keyboard


: Space key


: ”Backspace” key to delete a character


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


: To input current data to input box


: Enable Help function/message on screen

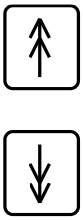

: For optional skip key input


: End of block

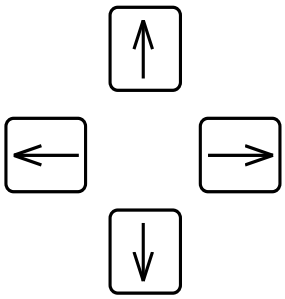

: Decimal fraction

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

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



：Switch cursor to Page Up /Page Down



：Edit cursor control key

# 3 How to operate 6 series controller

## 3.1 System status

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

1. Not Ready

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

- ✧ Emerency 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

12. Ready

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

- ✧ "Emerency 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.

13. Busy

System is operating program
Conditions:

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

14. Pause

In operational process, system pauses 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

15. 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

- Manual Function

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

### 3.2.1 JOG

- Description:
  - ✧ 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%

### 3.2.2 Rapid JOG

- Description:
  - ✧ 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.

### 3.2.3 INC JOG (incremental JOG)

- Description:

✧ 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

### 3.2.4 MPG

- Description:

✧ 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
- ✧ Those percentage movements is shared with INC JOG mode

## 3.3 Machining process

### 3.3.1 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

### 3.3.2 Single block

- Description:
  - ✧ 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

## 3.4 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.

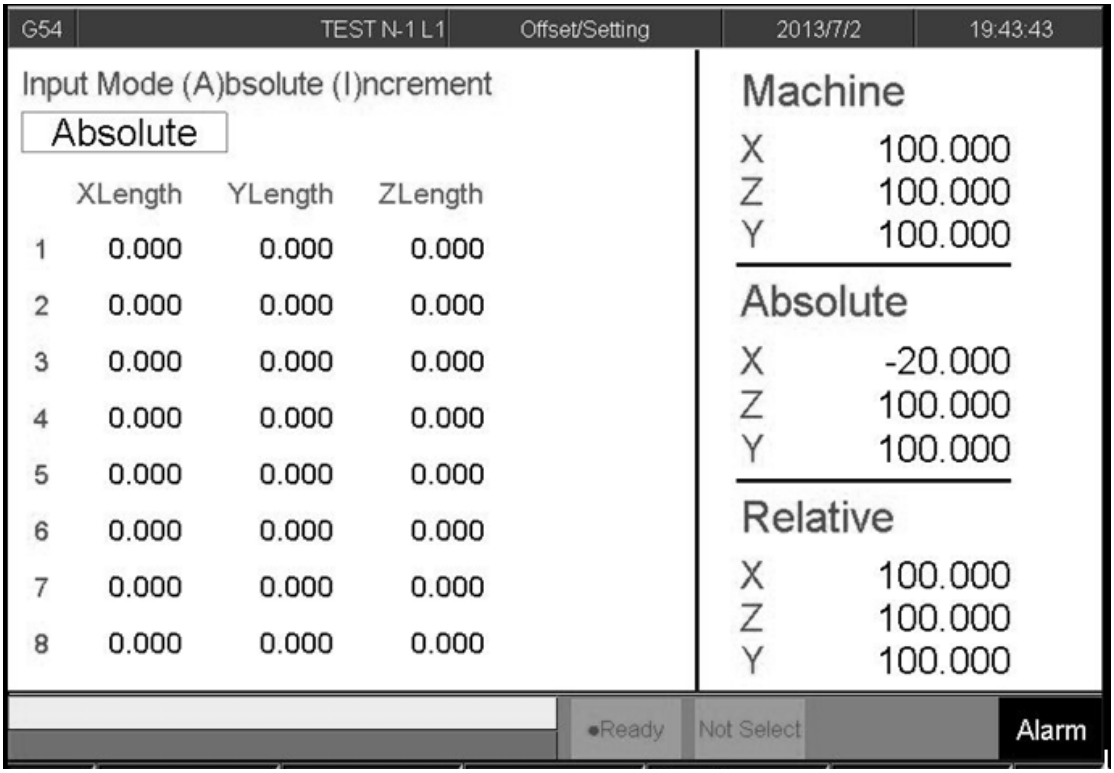
- Operation:
  - ✧ Release emergency stop button, CNC status will change “NOT READY” to “ READY ”
  - ✧ Select HOME mode
  - ✧ Press axis direction key(X+,X-,Y+,Y-,Z+…) ,each axis would start HOMING
  - ✧ Home direction is defaulted in the CNC parameter
  - ✧ Home function can run 3 axes at the same 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.

## 3.5 Tool Preparation

When programming and editing machining program, user usually wite Gcode related to tool and workpiece, and does not care about the real position. Moreover, tool length is different between each cutting tool, so tool preparation is necessary to avoid disturbing caused by different tool length.

Syntec lathe controller provides: tool length set, tool wear set, tool nose set and working shift.

### 3.5.1 Tool length set




- Path:

  ■F3 Offset/Setting → F2 Tool length set

  ✧ Switch the cursor to the position wants to set by using page up/down keys and arrow keys

  ✧ There are 3 methods to input: absolute, incremental and teaching

  ✧ Enter「A*** 」to input absolute value, here *** is input value; generally, this method is used to input tool radius compensation or tool length compensation.

  ✧ Enter 「I*** 」 to input incremental value, here *** is input value; generally, this method is used to input tool radius wearing compensation or tool length wearing compensation.

- ✧ Enter「X***」 or 「Z***」 to input teaching value.

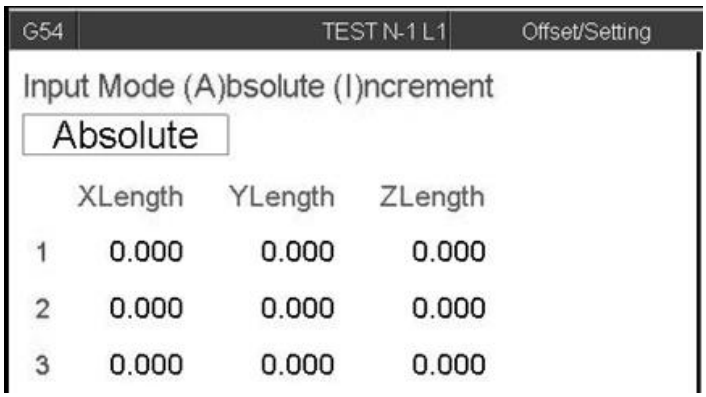
  User can input other axis name as Y, X1, X2, here *** is teaching value; generally, this method is used to input tool length compensation.
- ✧ Fomula calculated compensation value
- ✧ Tool radius + radius wear ) is real G41/G42 compensation value
- ✧ (Tool length + length wear) is real G43/G44 compensation value
- ✧ Using teaching method, user does not need to move cursor to desired input position, only need move cursor to right tool number and enter axis name, controller will automatically enter setup value according to axis name.
- ✧ After setting tool length by teaching method, tool wear value is set to zero automatically.

### 3.5.2 Tool Length measurement

Once NC file has T code, G43 will enable automatically. Therefore, users must set tool length before machining.

Before measuring, please confirm whether working shift(G92)and workpiece coordinate system is correct

#### 3.5.2.1 Z axis tool length measurement

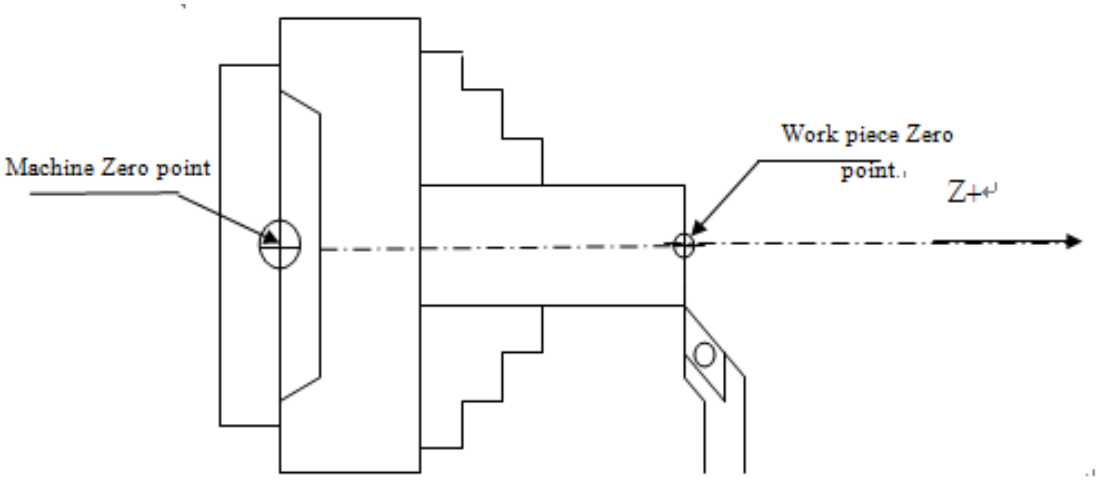



Use MDI mode and enter the tool No. want to measure. For example: T0101

- Path:
  - ✧ F3 Offset/Setting → F2 Tool length set
- Operation:
  - ✧ Move cursor to tool No. wants to set tool length, ex tool No. 1
  - ✧ Use manual mode (JOG, INJOG, MPG mode) to move the cutting tool until it touches the Z axis work piece zero point. Use teaching input method 「Z0」to set current workpiece zero point position.
  - ✧ Input Z axis cutting measuring

- ✧ Normally, it is not easy to move cutting tool until it touches the Z axis work piece zero point. In this case, user usually inputs Z axis cutting measuring

- Operation
  - ✧ Clamp the round workpiece by spindle
  - ✧ Use manual mode to move tool nose touch of workpiece end face, ensure that workpiece can be cut by moving X axis
  - ✧ Spindle rotates CW direction, using MPG mode to move X axis in direction that can cut workpiece
  - ✧ Move X axis in invert direction, do not move Z axis
  - ✧ Use teaching input method, enter 「Z0, the workpiece end face that just is cut is theworkpiece zero point.



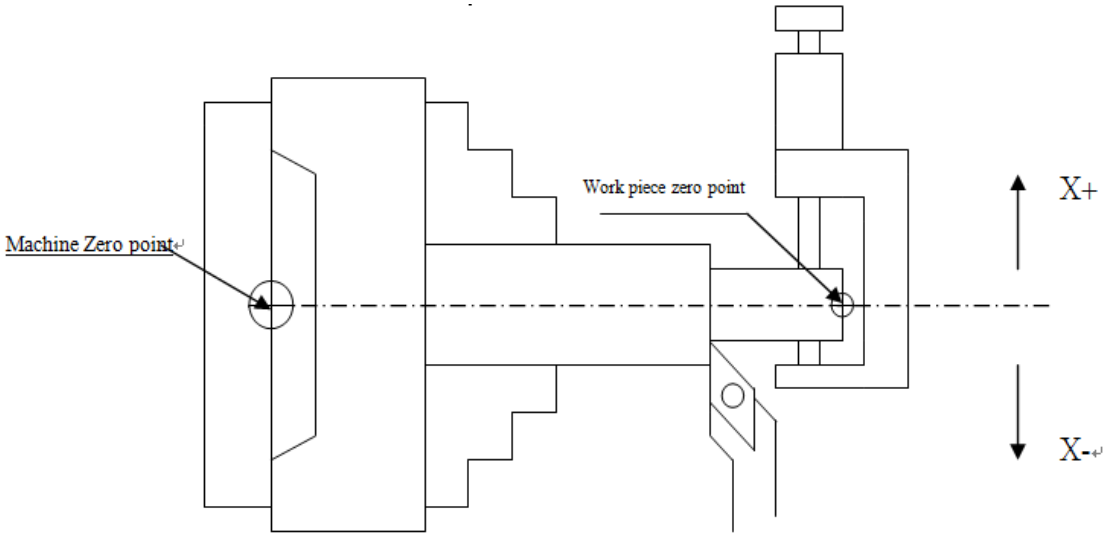

- ✧ Note: After setting Z axis tool length by teaching method, Z axis tool wear value is set to zero automatically.

### 3.5.2.2 X axis tool length measurement

- ✧ Input X axis cutting measuring
- ✧ Normally, it is not easy to move cutting tool until it touches the X axis work piece zero point. In this case, user usually inputs X axis cutting measuring

- Operation:
  - ✧ Use MDI mode and enter the tool No. wants to measure. For example: T0101
  - ✧ Path: F3 Offset/Setting → F2 Tool length set
  - ✧ Move cursor to tool No. wants to set tool length, ex tool No. 1
  - ✧ Use manual mode (JOG, INJOG, MPG mode) to move the cutting tool until it touches the X axis work piece zero point. Use teaching input method 「X0」to set current workpiece zero point position.
- Example
  - ✧ Clamp the round workpiece (diameter14mm) by spindle, ensure workpiece can be cut by moving cutting tool along Z axis
  - ✧ Rotary spindle, Cut a distance along Z axis
  - ✧ Retract cutting tool along Z axis, do not move X axis
  - ✧ Measure cutting diameter “D “
  - ✧ Input this teaching value “X+/-D”. For example, after cutting, the workpiece diameter is 13.5mm, input X13.5

✧ The plus/minus sign is decided by tool tip position, if tool tip is located at X+ (tool tip is located above central line), sign is “+”, if tool tip is located at X-, the sign is “-“



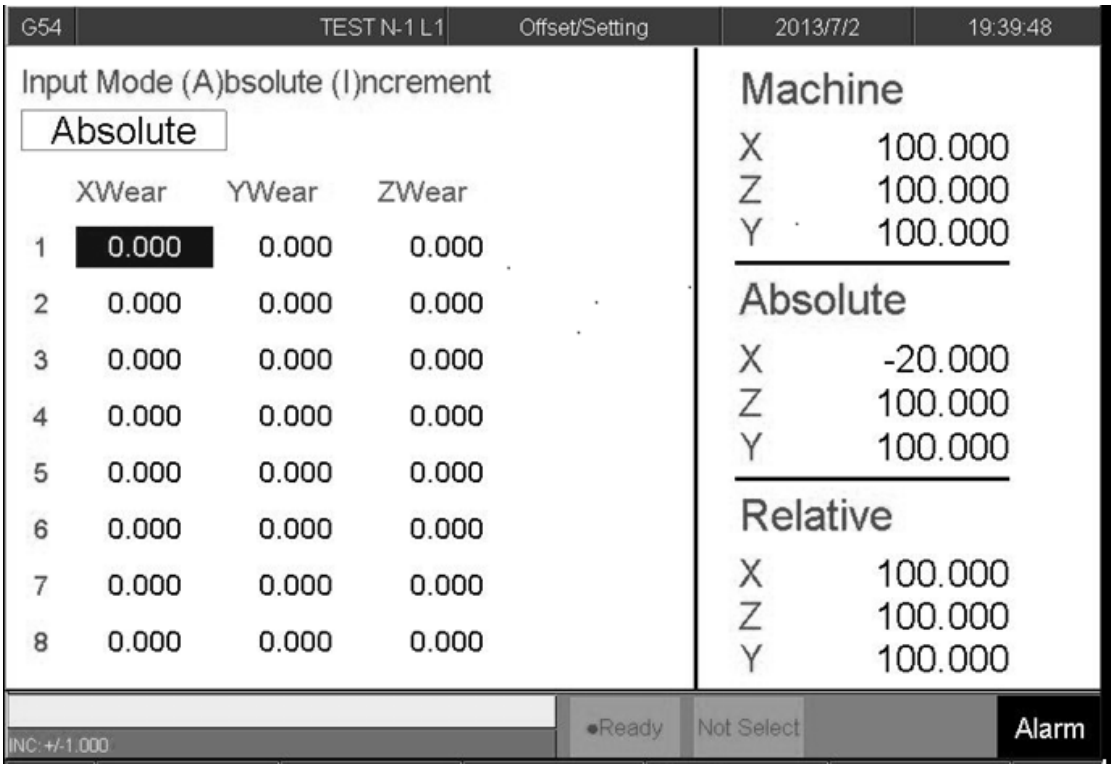

- Note:

✧ After setting Xaxis tool length by teaching method, X axis tool wear value is set to zero automatically.

### 3.5.2.3 **Tool wear setting**

Tool wear setting commonly used to compensate tool wear, tool expansion and contraction caused by thermal, or corrects the tool length in fine machining process. After completing tool length setting, do not modify tool length value randomly, only use tool wear setting to correct the size workpiece in fine machining process.

After setting tool length, tool wear value is set to zero automatically. If machining result has error, users can use tool wear function to compensate tool length. Actually, tool length = tool length + tool wear




- Path:
  - ✧ F3 Offset/Setting → F1Tool wear set
- Input method
  - ✧ Absolute: input the absolute value of tool wear by entering “A**”. Tool wear would equal this value

- ✧ Incremental: enter tool wear incremental value by entering “I** “. Tool wear will be changed to pre-value plus this value
- ✧ Input mode: only need to input “A” or “I” single character, then “absolute “or “incremental” setting would be kept, then users can directly input only setting value.
- ✧ plus/minus sign +/- :
- ✧ Input value can be assigned sign “+/-”
- ✧ The plus/minus sign is decided by adjusted direction of tool tip.
- ✧ If adjusted direction of tool tip is “Plus” direction, then enter tool wear value with plus sign
- ✧ If adjusted direction of tool tip is “minus” direction, then enter in tool wear value with minus sign

- Example:

If machining result is bigger than diameter 10um, tool tip is adjusted to X- , then input “I–0.01” to tool wear. -10um diameter will be added to current tool wear value. The next cutting time, tool tip cutting path would move toward “X-” direction 10um in diameter.

- Parameter

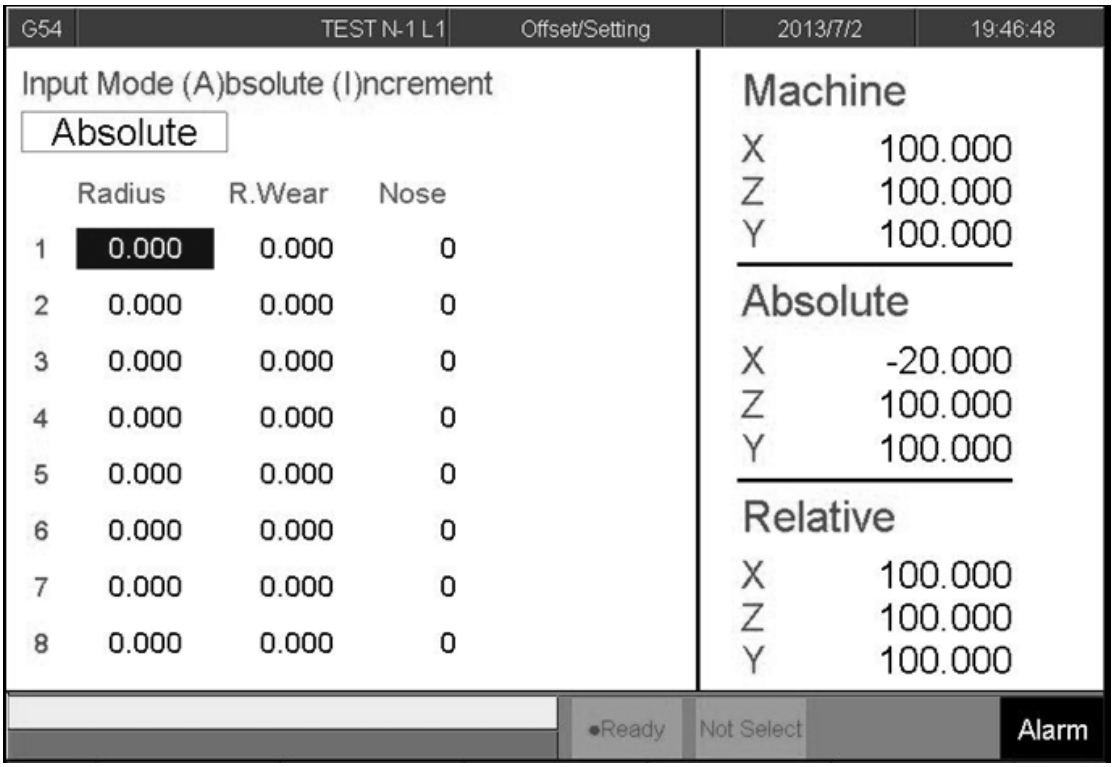
| No | Descriptions | Range | Unit | initial | occasion |
|---|---|---|---|---|---|
| 3245 | Max inc. value of input for tool wear compen.(BLU) | [1~200000] | BLU | 1000 | reset |

Set the param to limit the value of input for tool wear compention to avoid collide.

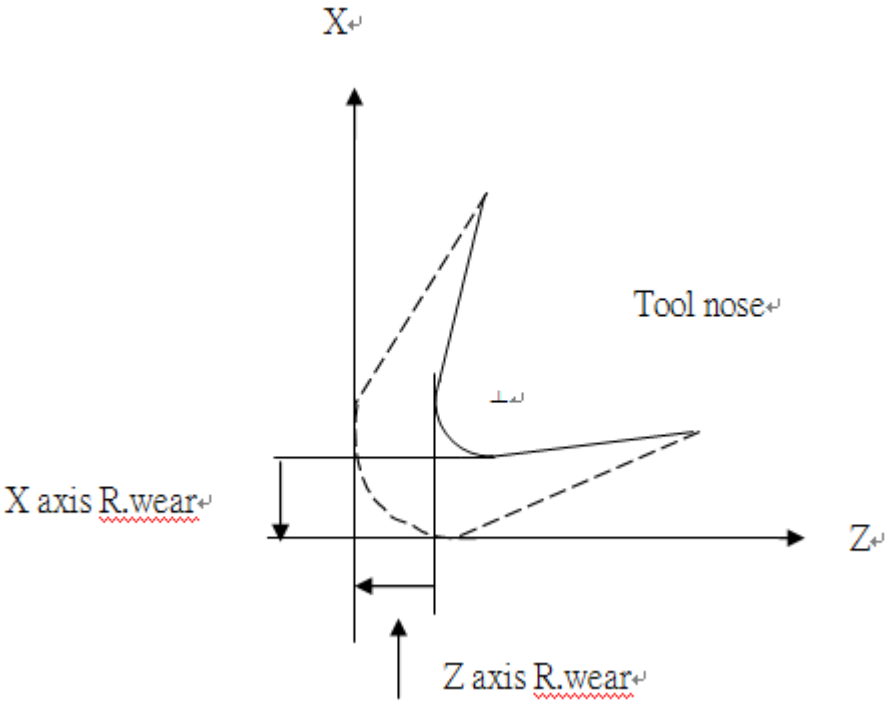
### 3.5.2.4 **Tool nose setting**

Tool nose is circular, however; tool length measurement only measures tool length, so for precision machining, users can set tool nose to compensate tool nose dimension error, operation method are shown as below:

- Path:
  - ✧ F3-Offset/setting→ F3 tool nose set



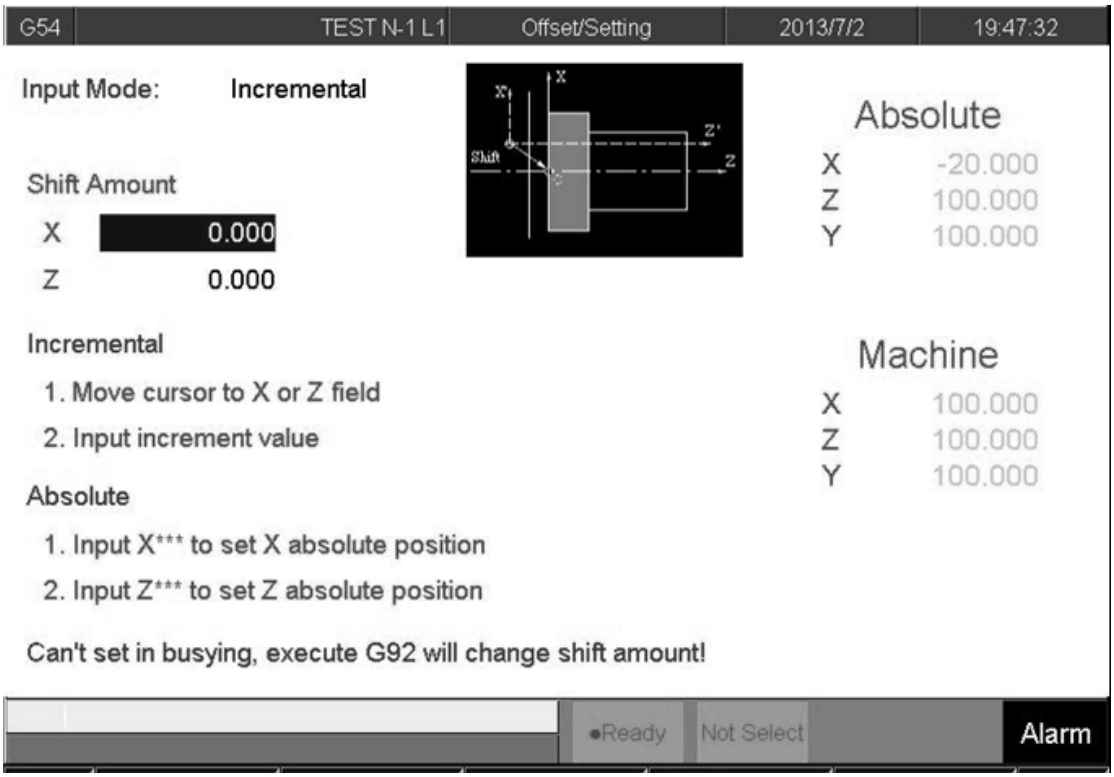

- There are 3 kinds tool nose data to input
  - ✧ Radius: tool nose radius
  - ✧ R.wear: tool nose radius wear, real tool nose equals to (Radius+R.wear)
  - ✧ Nose: epending on tool nose shape, there are 8 tool nose directions to be selected. Please see SYNTEC programming manual to know more detail (G41/G42).
  - ✧ Using G41/G42 in NC files to enable tool nose compensation


X
Tool nose
X axis R.wear
Z
Z axis R.wear

### 3.5.2.5 **Working shift**

During writing program process, user often uses the workpiece coordinate system, it needs to set properly in CNC before machining. Machining program will modify the relation between the workpiece coordinate and machine coordinate system according to different workpice coordinate system shift

- Syntec's controller provides 2 methods to set workpiece coordinate system
  - ✧ Use NC program or MDI execute G92 code
  - ✧ Use working shift function
  - ✧ Working shift function
- Path:
  - ✧ F3-Offset/Setting→ F5 working shift

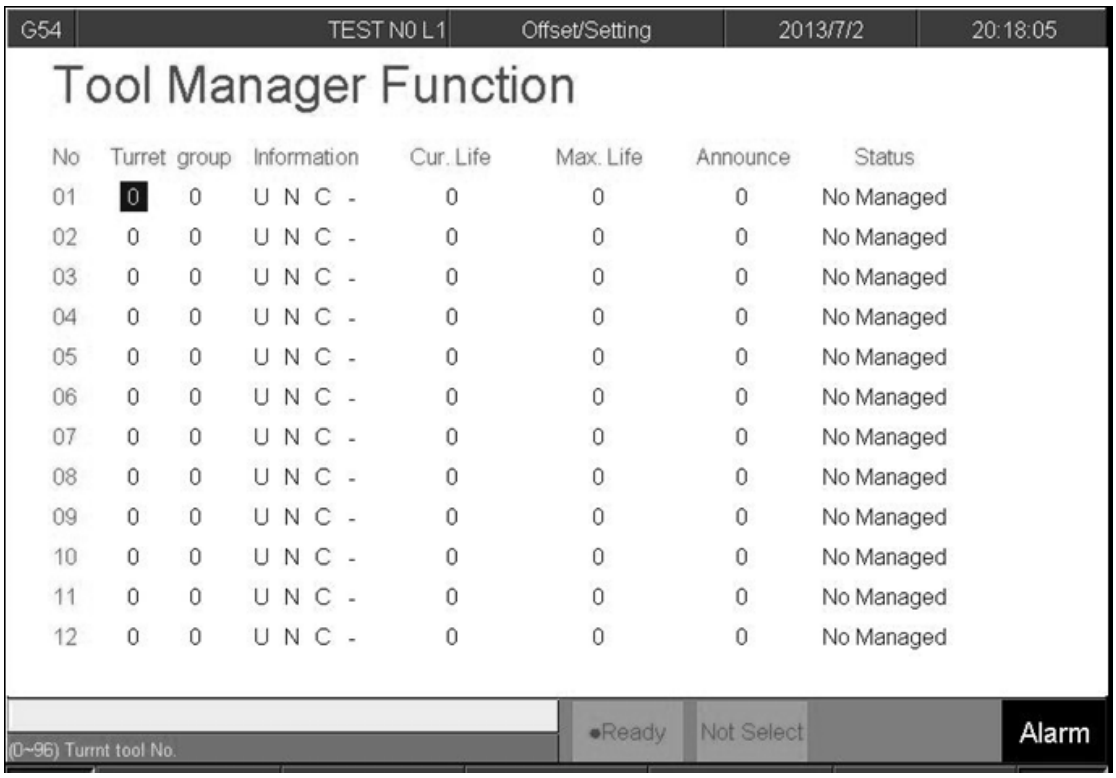



- Working shift function has 2 input methods.

- ✧ Incremental：move cursor to axis wants to set, enter offset value. This value describes the offset relation between absolute coordinate and machine corrdinate.
- ✧ Absolute：Enter「X***」or「Z***」.「***」means absolute coordinate value of current position.

### 3.5.2.6 **Tool 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, cutomers need to use this function, please contact to machine maker.




- Condition
  - ✧ Both auto and manual can be use.
  - ✧ 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 informations (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

## 3.6 Program preparation and execute machining

### 3.6.1 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 presssing F4-Monitor

### 3.6.2 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 F5-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 scaning 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

### 3.6.3 Machining test

#### 3.6.3.1 MPG simulation

- Condition
  - ✧ Only for single block and auto mode
- Operation
  - ✧ Select Auto mode
  - ✧ Press MPG simulaiton 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

#### 3.6.3.2 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

## 3.6.4 Machining monitor

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

### 3.6.4.1 Part count manager

- Description

1. Total accumpart
   - ✧ 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.

### 3.6.4.2 **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.

### 3.6.5 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.

#### 3.6.5.1 Emerency stop

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

#### 3.6.5.2 Alarm display

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

1. **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 still is not removed, press reset button, alarm window will be not displayed.
- Press F5-Maintain to display pending alarm contents.

#### 3.6.5.3 History alarm

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

- Path:
  - ✧ F5- maintain→ F1-Alarm→F2 History alarm
  - ✧ Display history alarm
  - ✧ The smaller No. alarm is, the sooner alarm occurres

#### 3.5.2.3 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.

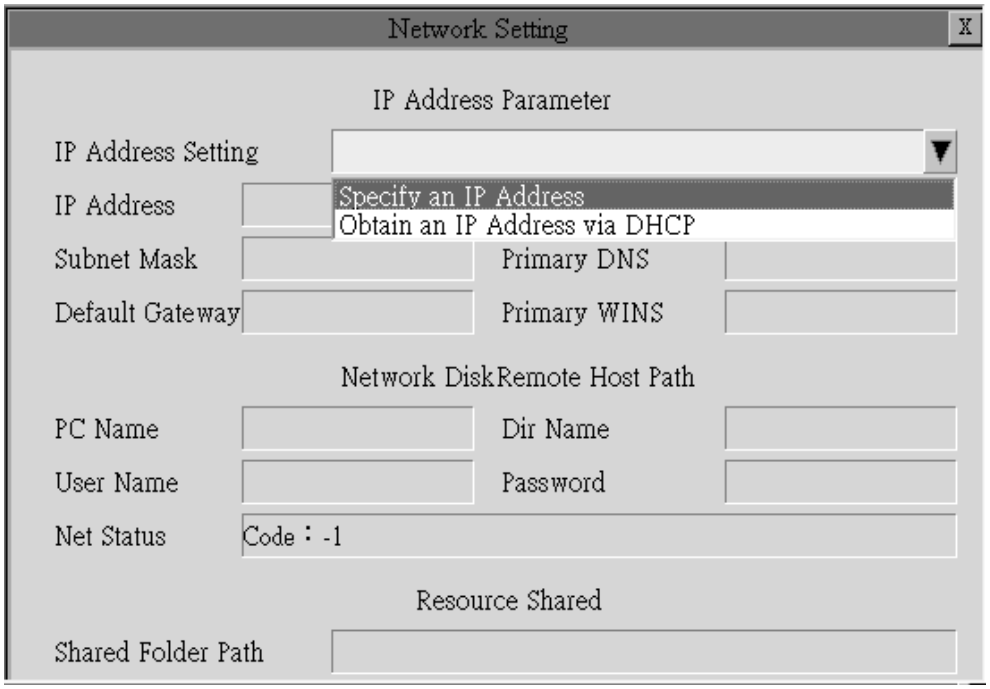
- Operation
  - ✧ Insert the external storage device into controller, or set the corresponding network folder
  - ✧ Switch to “Alarm” page(F5-maintain→F1-alarm)
  - ✧ To export the pending alarm, press F1-pending alarm→ F3-save alarm
  - ✧ To export the history alarm, press F2-history alarm→ F3-save 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

### 3.6.6 Network setting

A. On the interface screen, press down**"F5 Maintain" => "F2 Network setting"** to access IP address setting.

B. **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

C. **IP Address**: if you select "Specify an IP Address", enter the free IP address




D. **Subnet mask**: Enter the IP address for subnet mask (the same with PC subnet mask).

E. **PC Name**: Enter the full computer name of your PC.

F. **Dir Name**: Enter the sharing folder name (the same name with PC sharing folder )

G. **User Name**: Enter GUEST

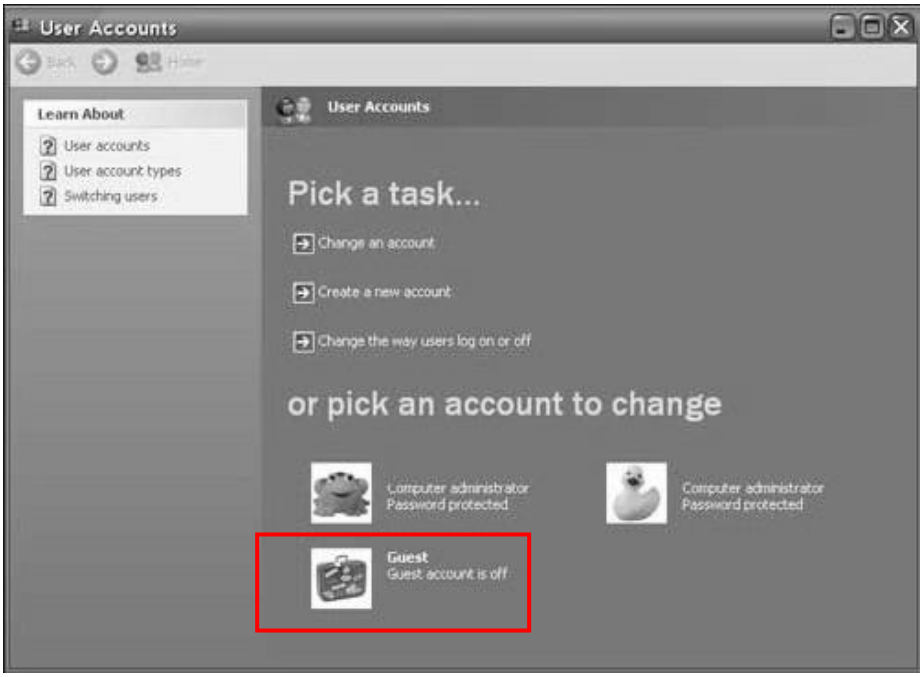
H. Press「**F1 OK**」, and then reboot controller to finish installation.

## 3.6.7 PC setting

### 3.6.7.1 XP OS

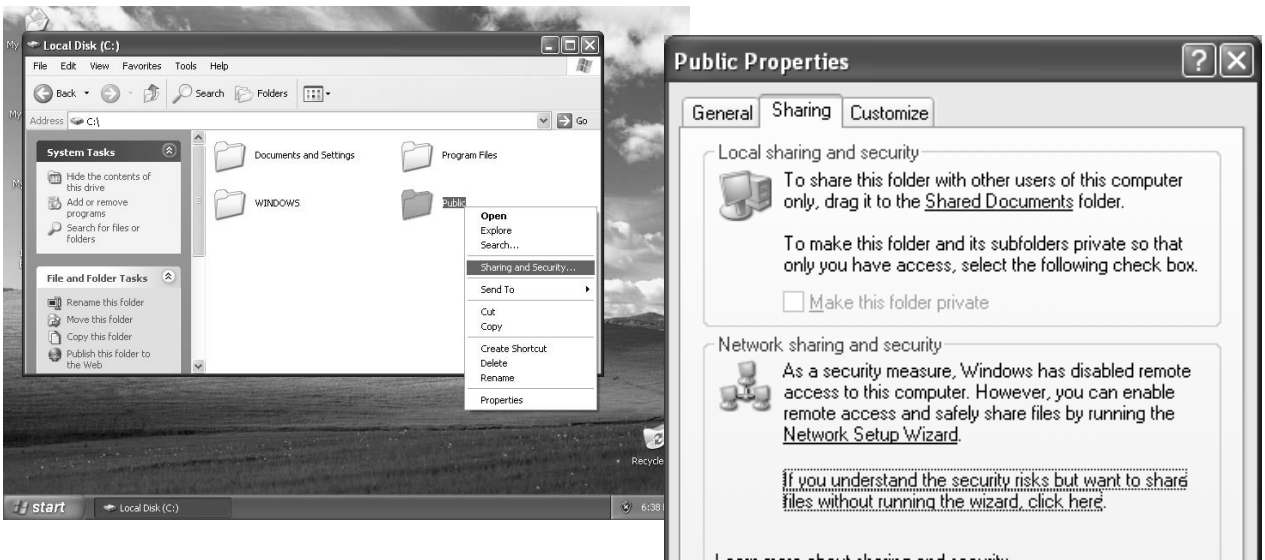
1. Guest account setting

Log in as Administrator and select “start”→ “control panel”→”user account”→ Guest

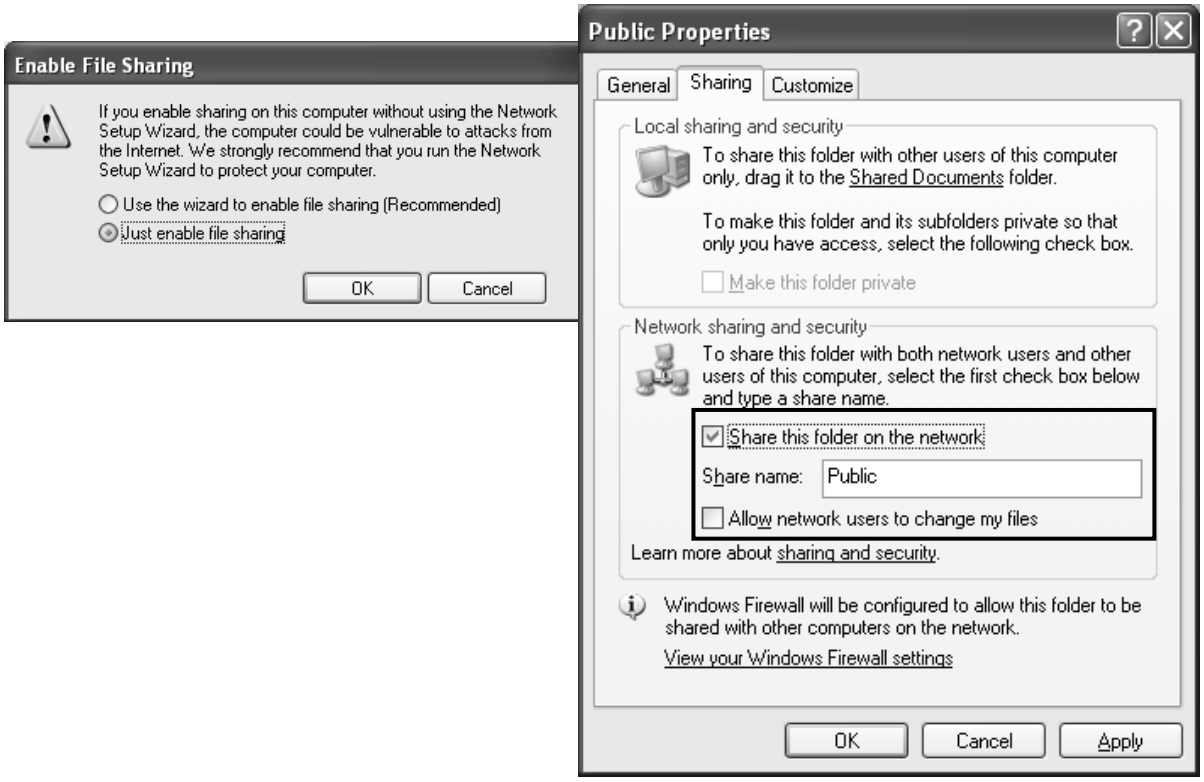



2. Sharing resource setting

- 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”

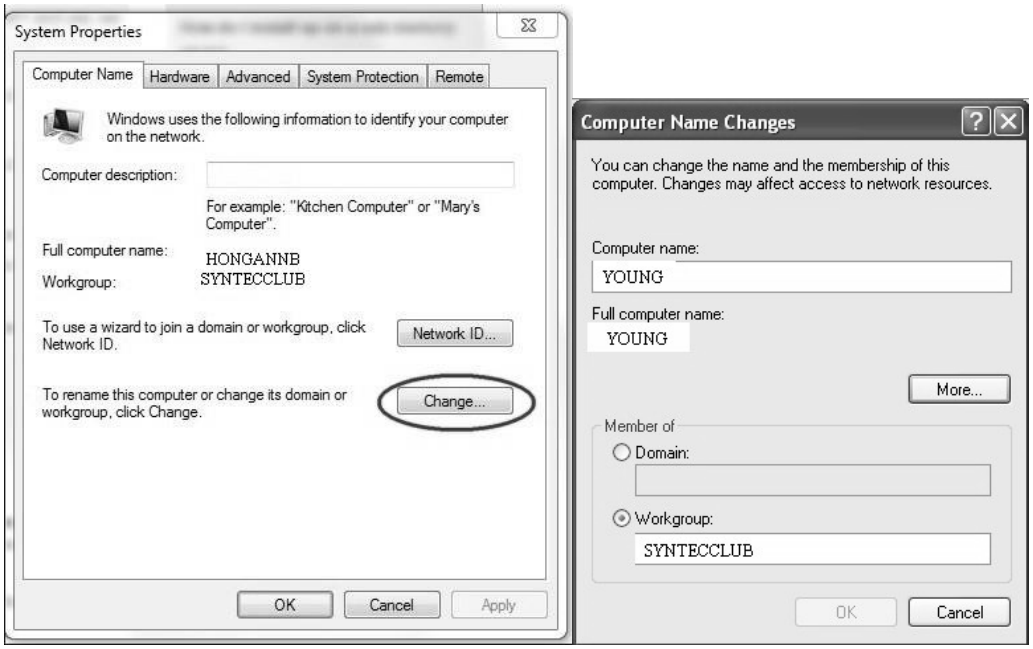


3. Click “OK” to confirm sharing setting; Select “Share this folder on the network”, and “Allow network users to change my files”.



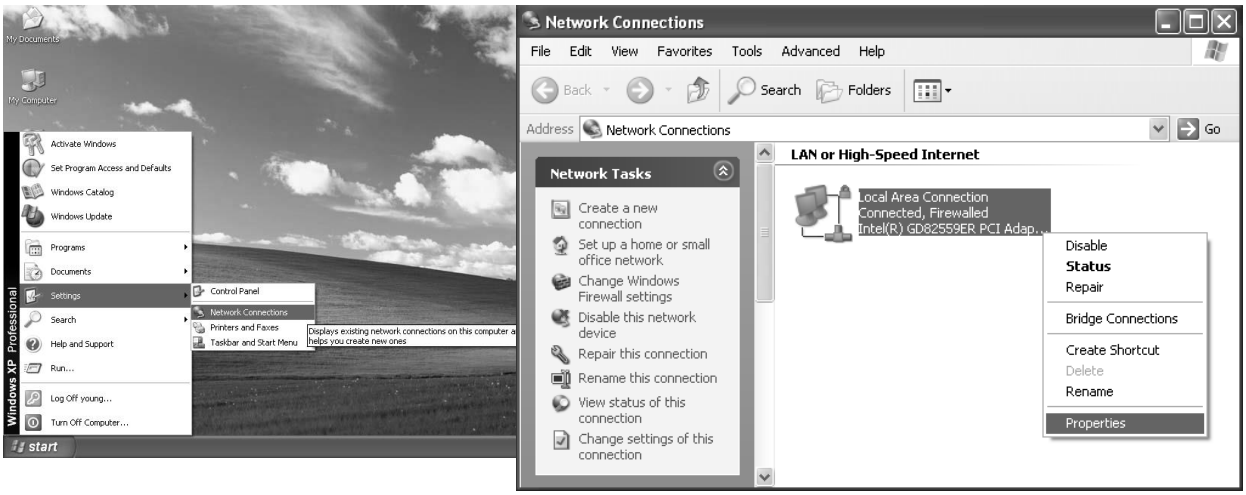
4. Setting PC name and workgroup

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



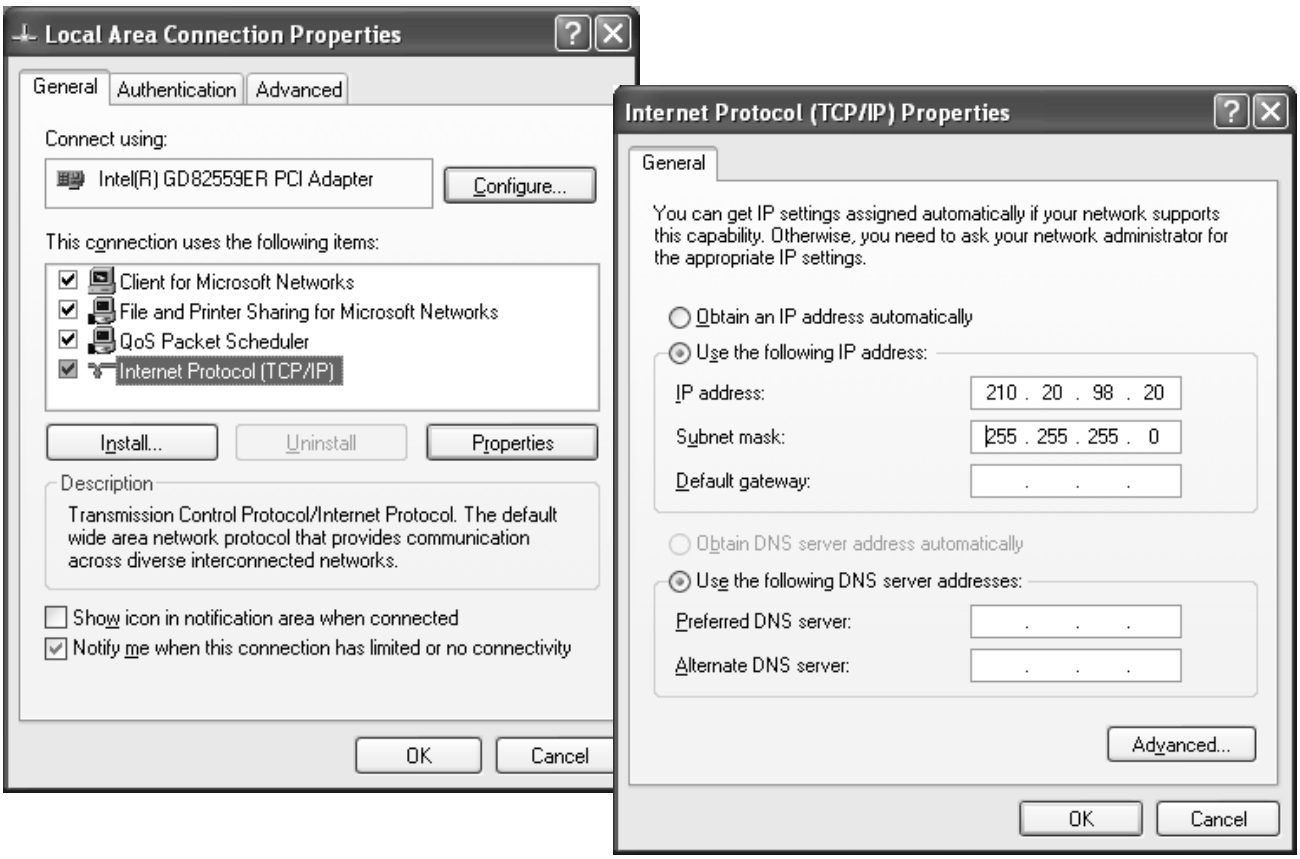
5. TCP/IP setting

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




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

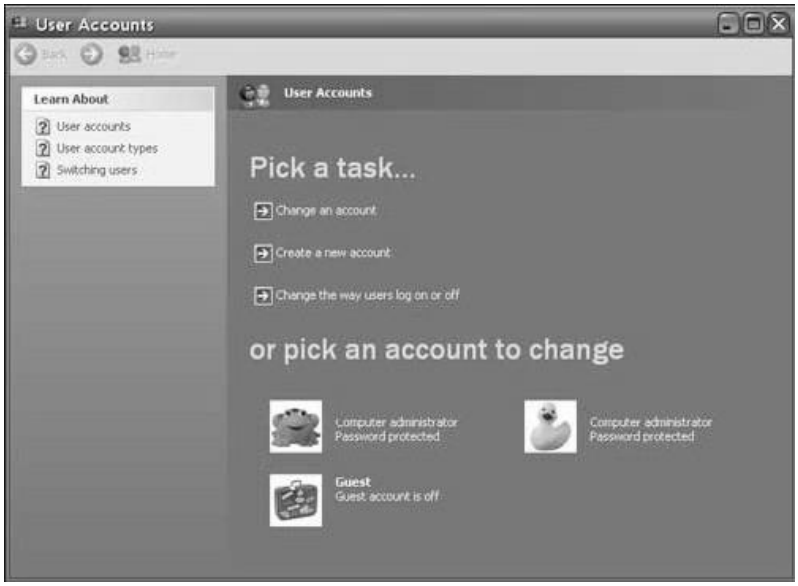
- ✧ Jumper cable (without HUB), select "use the following IP address" and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)
- ✧ Network cable (with HUB), select "Obtain an IP address automatically"



### 3.6.8 VISTA OS

1. Guest account setting

Log in as Administrator and select “start”→ “control panel”→ “user account”→ Guest

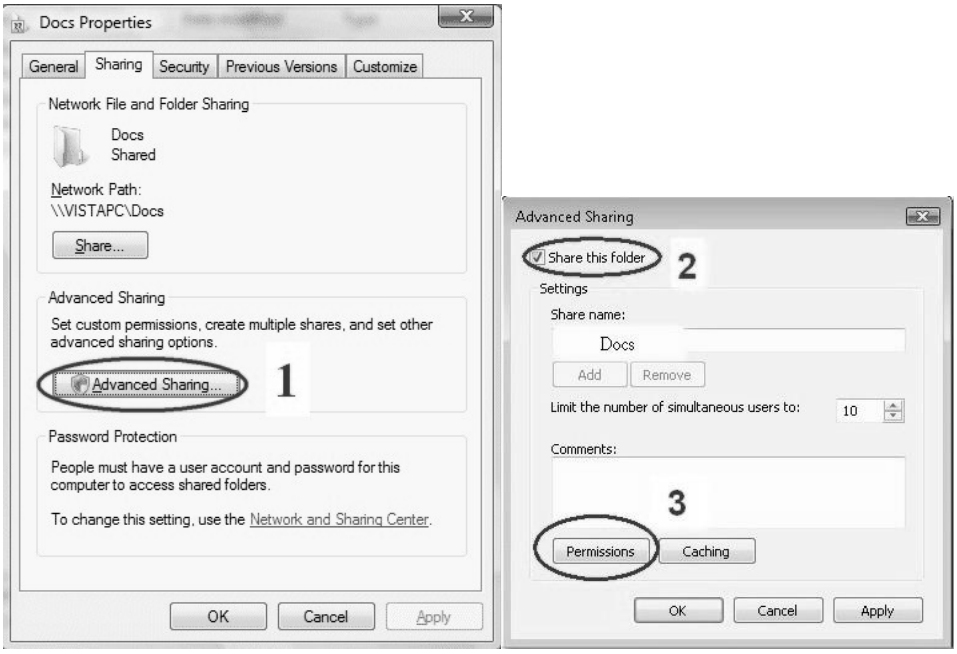


2. Sharing resource setting

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

A. Click on "advanced sharing"

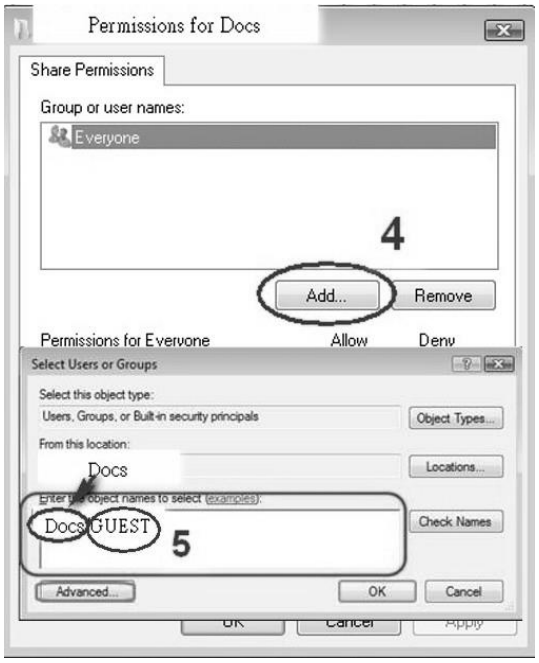
B. Click on "share this forlder"




C. Click on "permission"

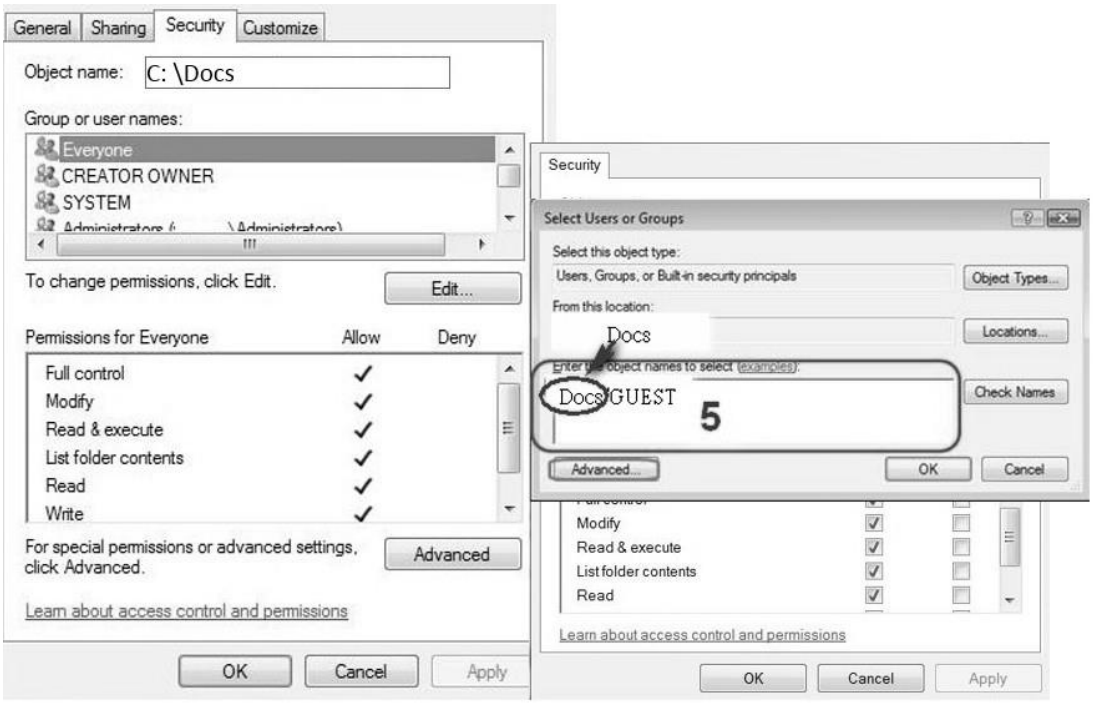
D. Click on "add"

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


Permissions for Docs
Share Permissions
Group or user names:
Everyone
4
Add...
Remove
Permissions for Everyone
Allow
Deny
Select Users or Groups
Select this object type:
Users, Groups, or Built-in security principals
Object Types...
From this location:
Docs
Locations...
Docs;GUEST
5
Check Names
Advanced...
OK
Cancel

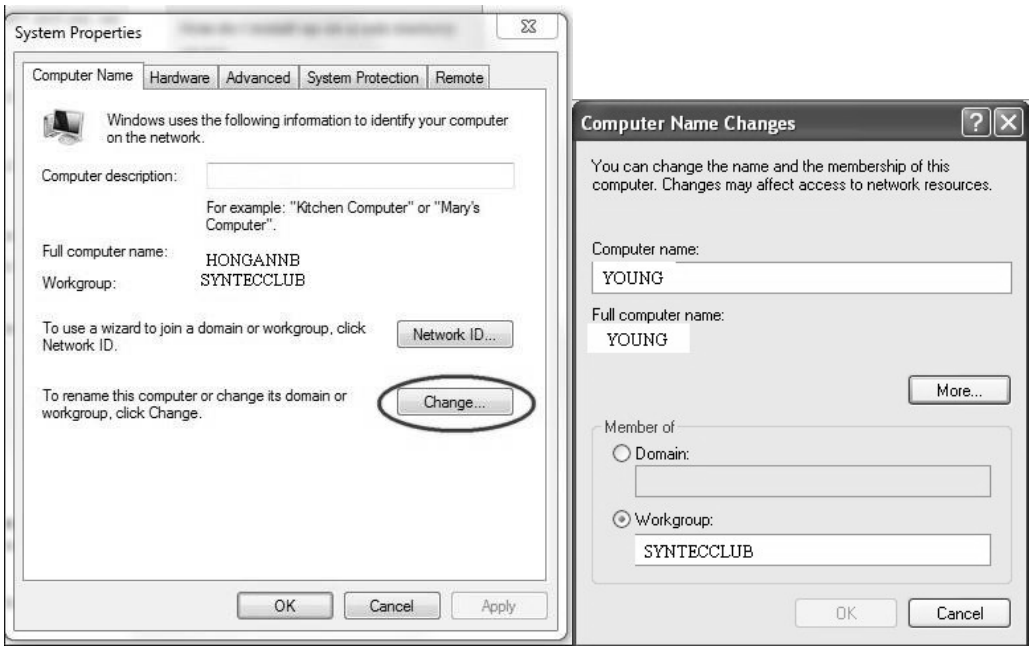
3. Security setting

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



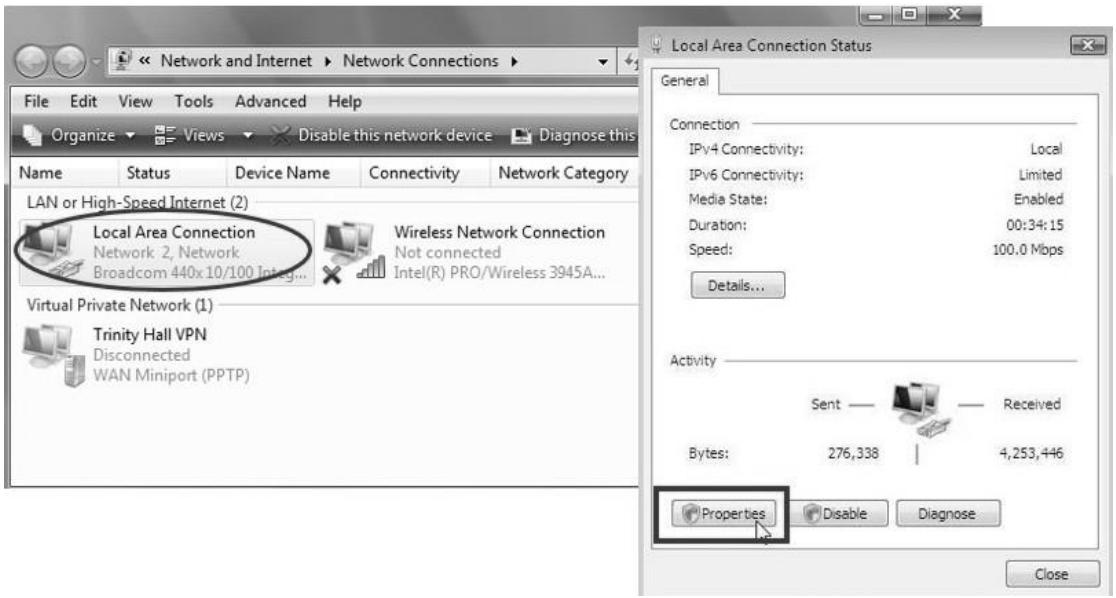

4. Setting PC name and workgroup

"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

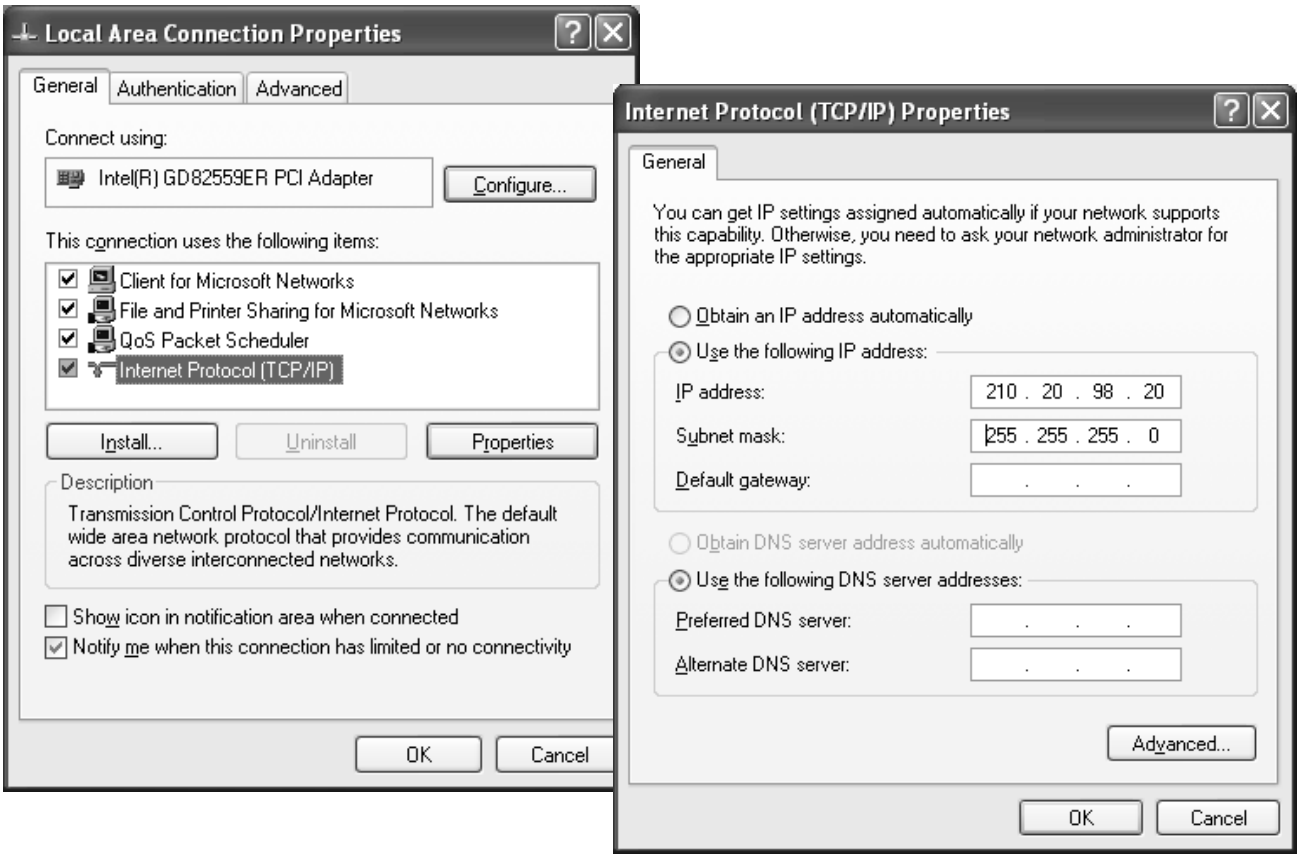
a. “START” → “control panel” → “Network and sharing center”→“Network connection manager” → “Properties”




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

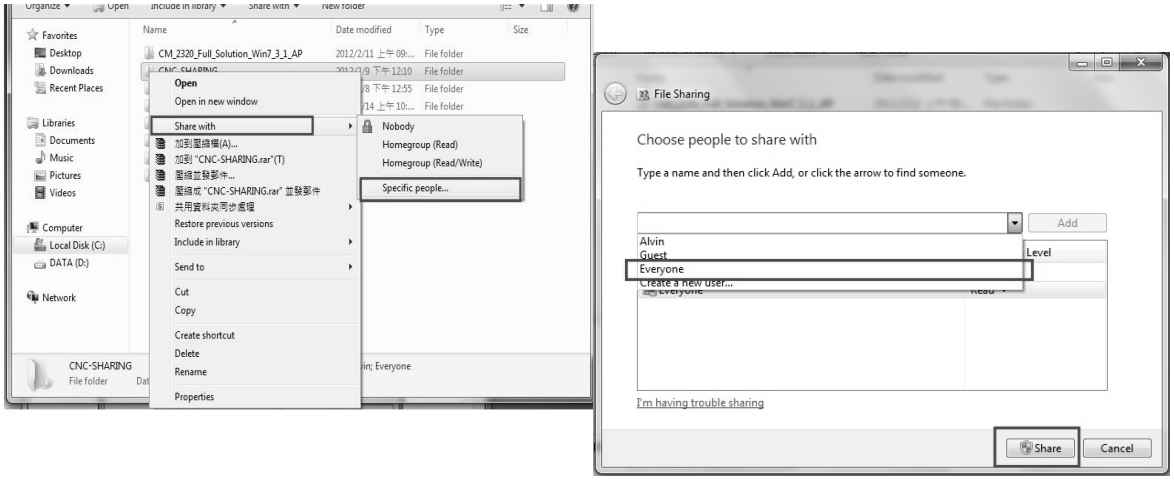
✧ Jumper cable (without HUB), select “use the following IP address” and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)

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

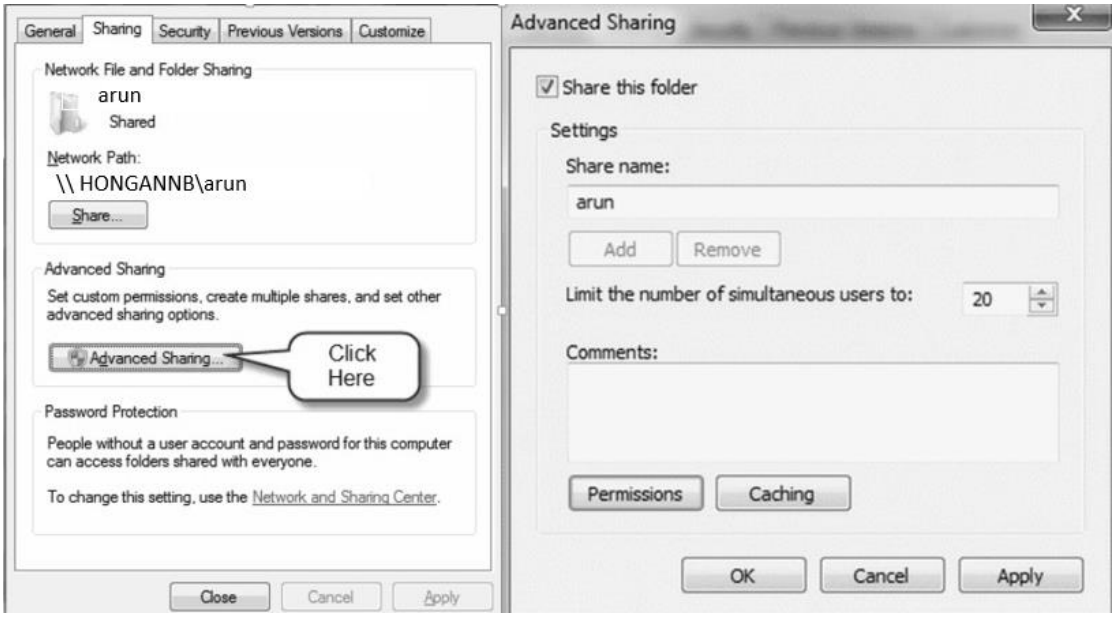


### 3.6.9 Win 7 OS

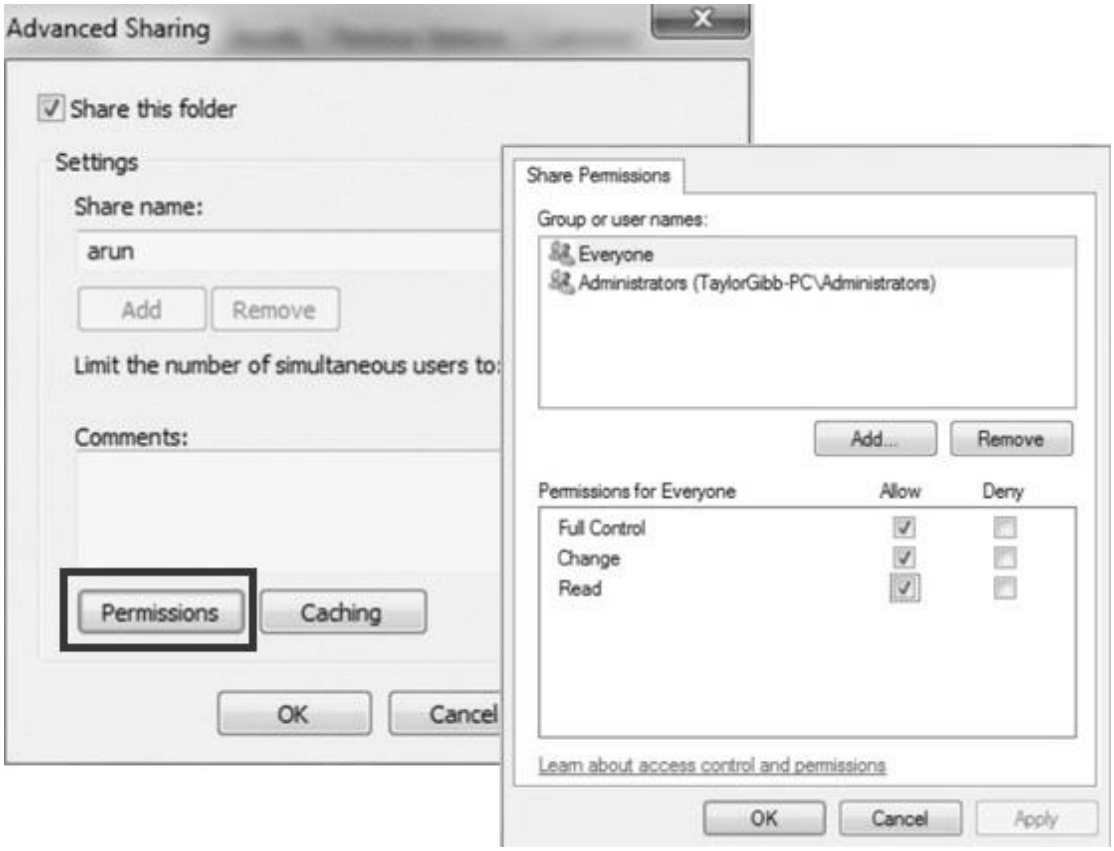
1. Sharing resource setting
   - ✧ Right-click on folder wants to share, select “share with” and “spefic people”
   - ✧ Share this folder to everyone, and then click “Share” as follows.
   - ✧ Set permission as write/read



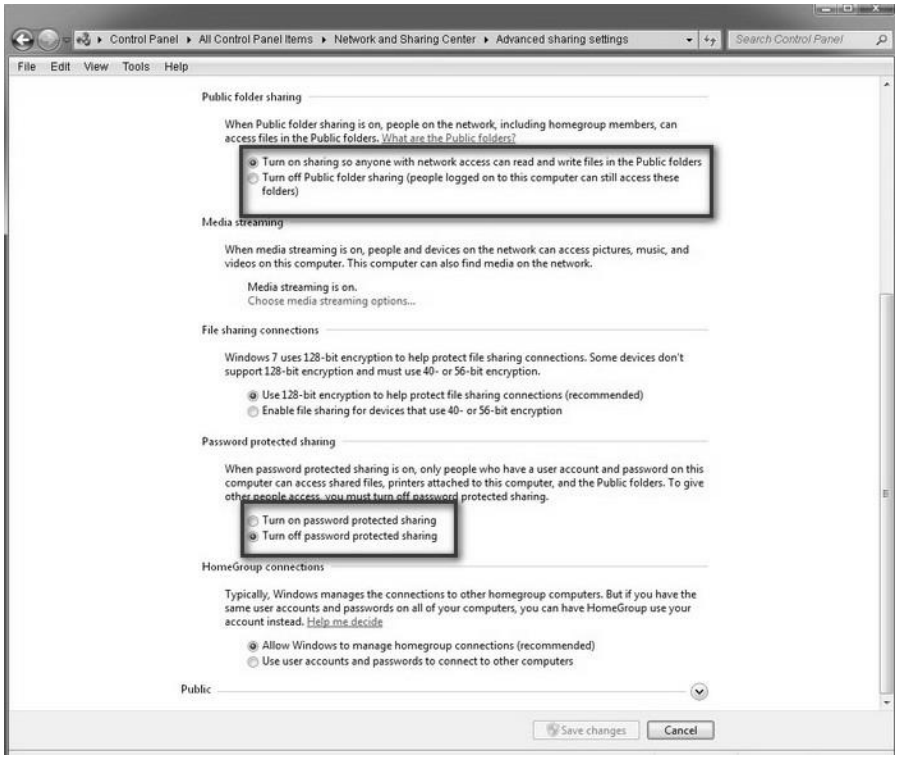

   - ✧ Left-click on “advanced sharing”and select “share this forlder”



✧ Select “permission” and select “full control” “only read”and “change”

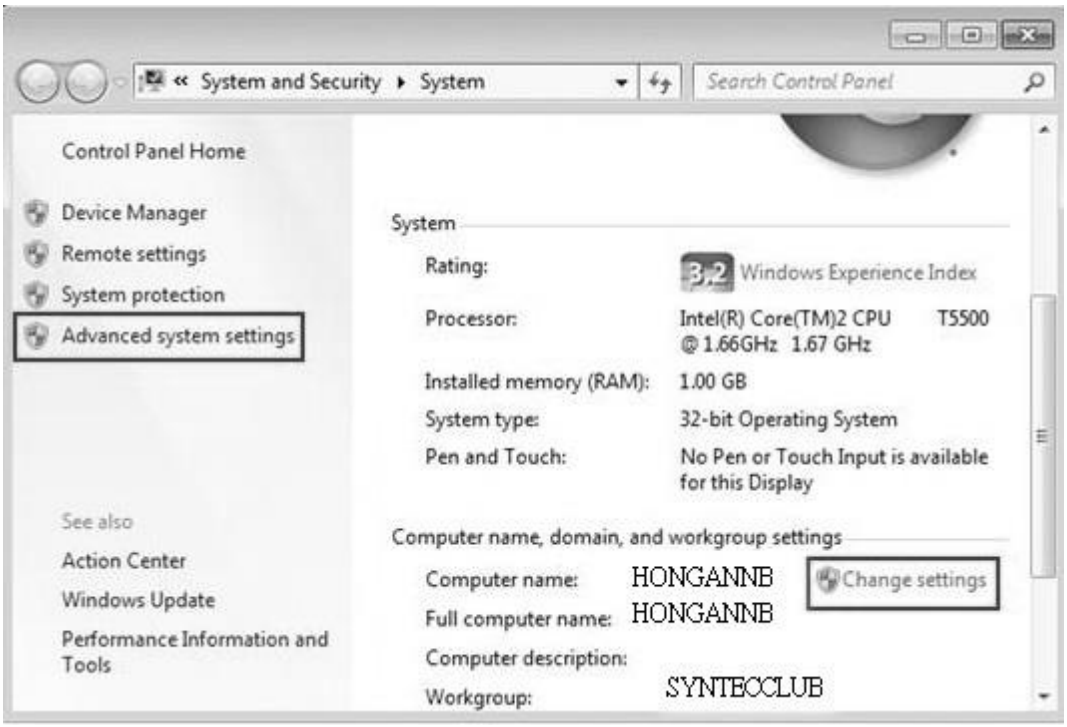



✧ Open”Network and sharing center”, select “turn off password protected sharing” and “Open sharing…..”


Control Panel ▸ All Control Panel Items ▸ Network and Sharing Center ▸ Advanced sharing settings
Search Control Panel
File Edit View Tools Help
Public folder sharing
When Public folder sharing is on, people on the network, including homegroup members, can access files in the Public folders. What are the Public folders?
Turn on sharing so anyone with network access can read and write files in the Public folders
Turn off Public folder sharing (people logged on to this computer can still access these folders)
Media streaming
When media streaming is on, people and devices on the network can access pictures, music, and videos on this computer. This computer can also find media on the network.
Media streaming is on.
Choose media streaming options...
File sharing connections
Windows 7 uses 128-bit encryption to help protect file sharing connections. Some devices don't support 128-bit encryption and must use 40- or 56-bit encryption.
Use 128-bit encryption to help protect file sharing connections (recommended)
Enable file sharing for devices that use 40- or 56-bit encryption
Password protected sharing
When password protected sharing is on, only people who have a user account and password on this computer can access shared files, printers attached to this computer, and the Public folders. To give other people access, you must turn off password protected sharing.
Turn on password protected sharing
Turn off password protected sharing
HomeGroup connections
Typically, Windows manages the connections to other homegroup computers. But if you have the same user accounts and passwords on all of your computers, you can have HomeGroup use your account instead. Help me decide
Allow Windows to manage homegroup connections (recommended)
Use user accounts and passwords to connect to other computers
Public
Save changes
Cancel

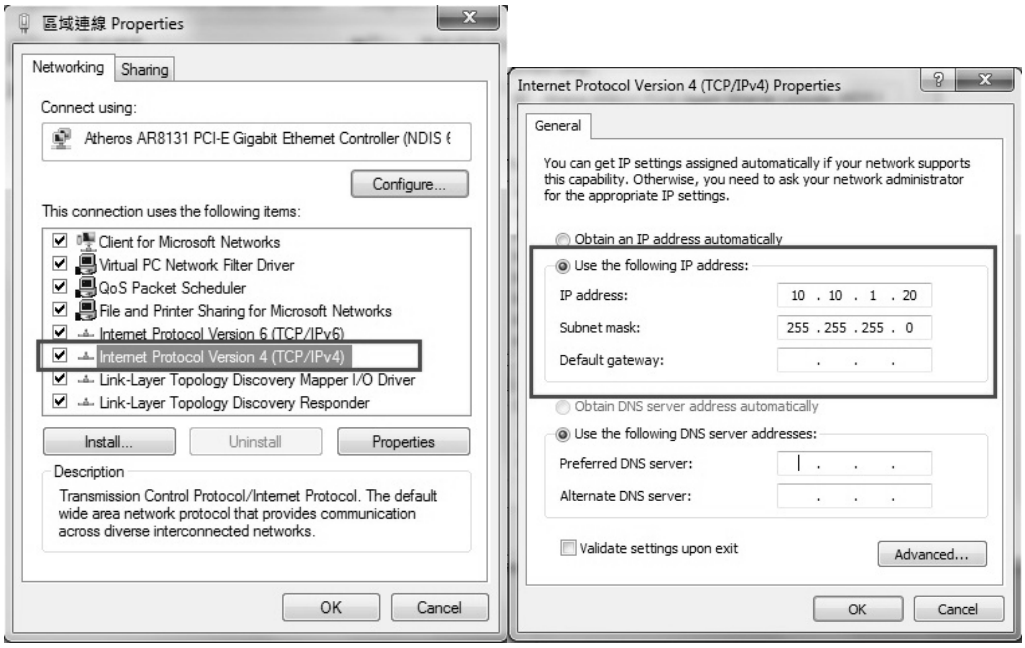
2. Setting PC name and workgroup

"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.



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”

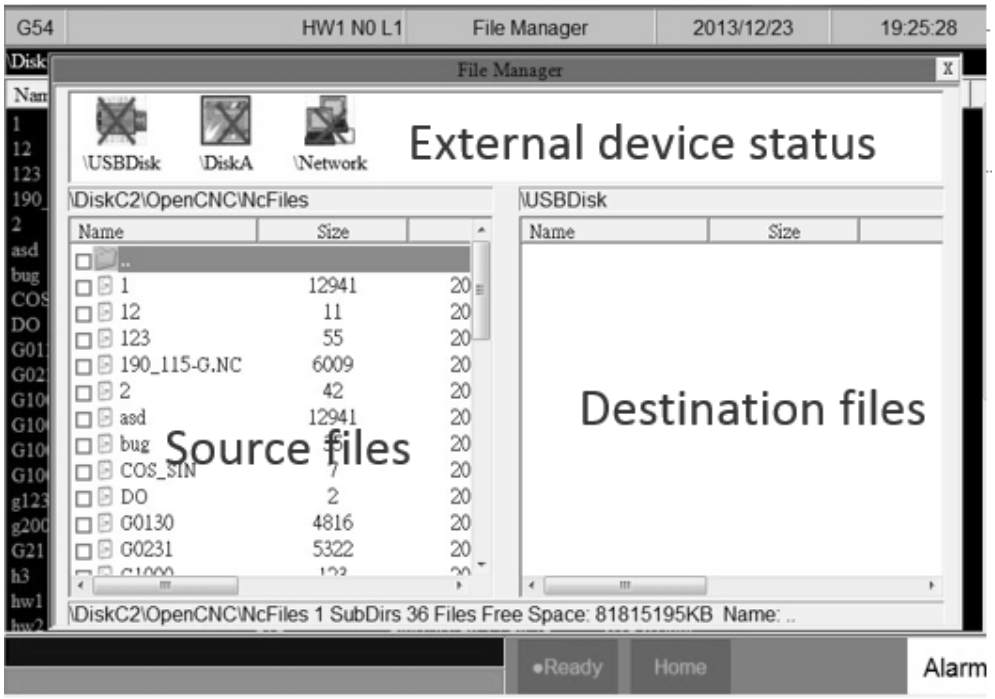


## 3.7 File transfer

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.

### 3.7.1 File import

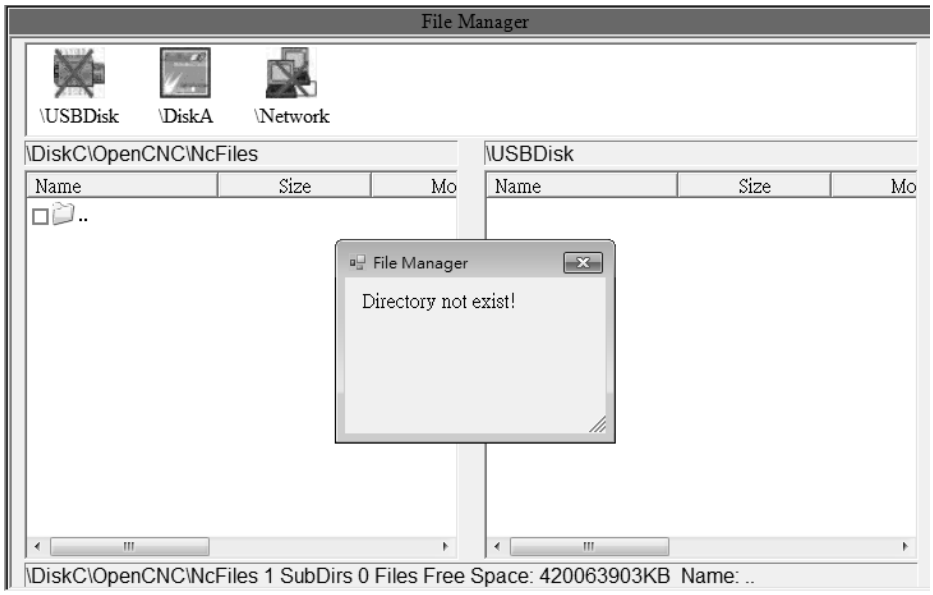
- Operation
  - ✧ Path: F2-program→ F4-file manager → F4-file transfer→ F1-File import
  - ✧ Other interface will appear on screen, press F5-“device change” to move cursor to desired external device on the status bar
  - ✧ Press [Enter] key to eccess to inside device, if the device icon has red cross, 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



### 3.7.2 File export

- Operation
  - ✧ Path: F2-program→ F4-file manager → F4-file transfer→ F2-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 eccess 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 | 6 Series Lathe Operation Manual | 2013/07/08 | 許維中<br>段生杏 | | 0.0.11 |
| 1.1 | Modify [WorkPiece Cord] key | 2014/03/02 | Andy Ngo | Changhungan | |
| 1.2 | Modify Network setting | 2014/06/27 | Andy Ngo | Changhungan | |
| 1.3 | Add Chinese topic, and increase front size | 2015/11/13 | Linda Chen | Yu-An Chiang | |
| 4 | | | | | |

## 4.2 Contact Window

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

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

| | |
|---|---|
| District,Xiamen City, Fujian Province, China | Rong Yuan Road No. 4 day science and Technology Park Building No. 1, 2 door 501 |
| Henan-Luoyang<br>The Suzhou new generation of CNC Equipment Co., Ltd. Luoyang Company<br>TEL：0379-65110352<br>FAX：0379-65110352<br>Address：<br>601B, Runsheng Building, Cross of Sanshan Road & Heluo Road, Hi-and-New Tech Park of Luoyang, Henan | Chongqing<br>The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch<br>TEL：023-67913296<br>FAX：023-67634382<br>Address：<br>Room 41-4, Building A, Hongding International, Guanyinqiao, Jiangbei District, Chongqing |
| Jiangsu-Nanjing<br>The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch<br>TEL：0512-69008860-300<br>Address：<br>Room 505, Unit 3, Building 12,Wuyi Luzhou Guan Chu Yuan, No.99, Tianyuan Middle Road, Jiangning District, Nanjing | Hubei-Wuhan<br>Suzhou new generation of CNC Equipment Co., Ltd. Wuhan Office<br>TEL：027-87638876<br>FAX：027-87204137<br>Address：<br>Room2003-2006,A#Guangguguoji,456#Luoyu Road,Donghukaifaqu,Wuhan City, China |
| Shenyang<br>The Suzhou new generation CNC Equipment Co., Ltd. Shenyang Office<br>TEL：024-25821398<br>FAX：024-25821398 | Anhui-Hefei<br>The new generation of CNC Equipment Co., Ltd. in Suzhou, Hefei Office<br>TEL：15951989576<br>Address：<br>Room 704, Building 13,Wanzhen Xiaoyao Garden Ⅳ, Taihu East Road, Hefei |
| Shenzhen<br>Shenzhen branch<br>District, Sham Chun City, Guandon Province<br>TEL：0755-84584085<br>Address： | Thailand<br>SIAM RADERMEN Co., Ltd.<br>TEL：+66-27553536<br>FAX：+66-27575476<br>E-Mail：<br>lee_cheahow@yahoo.com |

| | |
|---|---|
| Room 706, Hsin Toun Building, Toun Chung Road, Ai Lian Hsin Toun Village, Loggan | Address：<br>52/4,MOO5,THEPARAK RD.(KM11.5)BANGPLEEYAI,BANGPLEE, SAMUTPRAKRAN 10540 |
| Malaysia<br>SURE FIRST ENTERPRISE (M) SDN BHD.<br>TEL：+603-61577632<br>FAX：+603-61574632<br>Address：<br>47000 NO 19.JALAN TSB 6,TAMAN INDUSTRI SUNGAI BULOH.47000 SELANGOR DARUL.EHSAN | Lebanon / Damascus<br>Kesmeyan Group Co.<br>Lebanon Beirut, Jdeideh - Ammaieh - Kessmeyan Group Build.<br>Damascus<br>TEL: +963 11 6731602<br>Mob.: +963 988 211407<br>E-Mail : info@rockwell.tw<br>TEL：+961-76-883778<br>FAX：+961-1-901807<br>E-Mail：<br>kesmeyan@rockwell.tw |
| France<br>INTEGRATION CNC<br>TEL：+33 2 35 06 07 83<br>FAX：+33 2 35 06 07 83<br>http://www.integrationcnc.fr/<br>E-Mail：<br>vaque.j@online.fr<br>Address：<br>6, Rue l'arché de Copigny, 76630 BAILLY, FRANCE | Turkey<br>KASIKCIOGLU ELEKTROMEKANIK OTOMASYON<br>TEL：+90-224-4434684<br>FAX：+90-224-4434685<br>E-Mail：<br>murat@kasikcioglu.com<br>Address：<br>Besevler K.S.S. 17 BLOK NO:72 Nilufer/BURSA/TURKYIE |
| ISTANBUL/TURKEY<br>AKSIS MAKINA<br>TEL：+90 212 613 87 83<br>FAX：+90 212 613 87 85<br>http://www.aksismakina.com<br>E-Mail：<br>info@aksismakina.com<br>Address：<br>Yeni doğan mahallesi karakaş Sk. Emintaş Erciyes Sanayi Sitesi | |

| | |
|---|---|
| no:12/206 | |