21 Series Mill Controller

Operation Manual

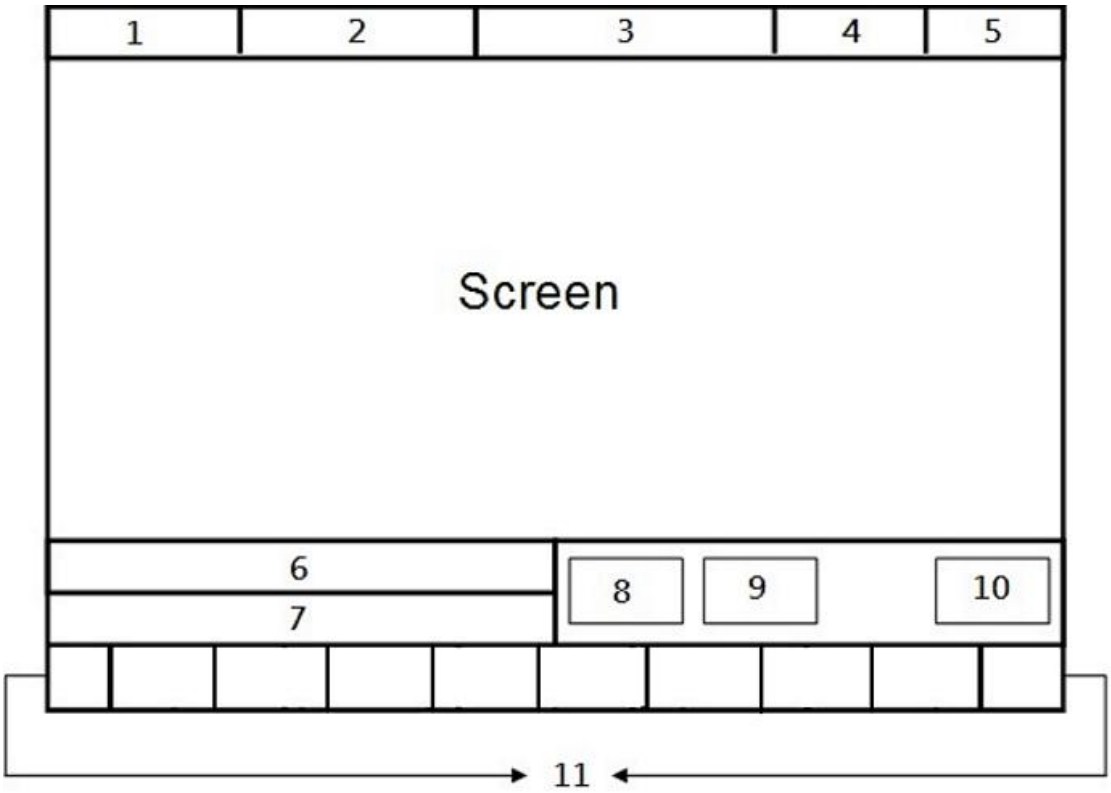
Date: 2013/11/08

Version: 1.0

Contents

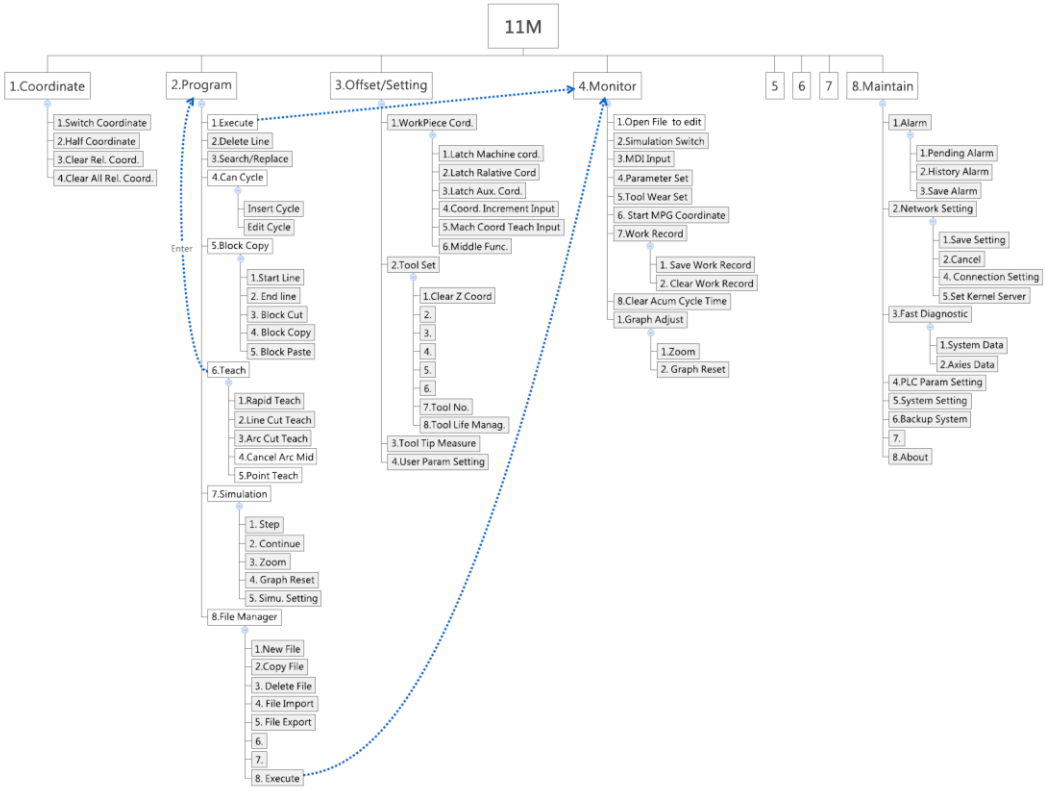
1 Function Key and System Configuration

1.1 Main Screen Sections

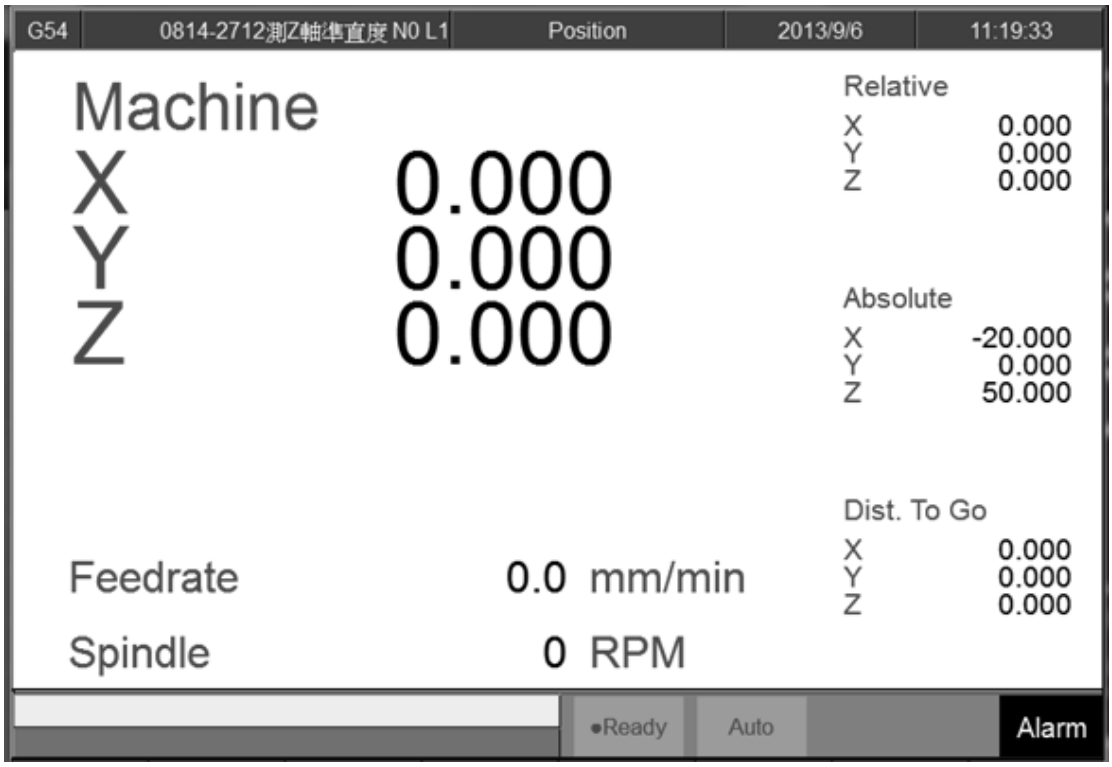


- Meanings for Sections on the Display:
    1. Present coordinate system
    2. Working file name and machining command line
    3. Title of interface(Current interface)
    4. Date
    5. Time
    6. Data input
    7. Display(Hint)
    8. Status
    9. Mode
    10. Alarm
    11. Function Key

1.2 CNC System Configuration



1.3 Coordinate



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

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

1.3.1 Explanation of Function

1.3.1.1 Coordinate Display

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

1.3.1.2 F(Feedrate)

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

1.3.1.3 S(Rotating Speed of Spindle)

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

1.3.1.4 Run Time

- ✧ Machining Duration.

1.3.1.5 Part counter

- ✧ Number of parts that had been finished.

1.3.1.6 T(Tool No.)

- ✧ Current Tool no. and Tool compensation no.

1.3.2 Switch Coordinate

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

1.3.3 Half Coordinate

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

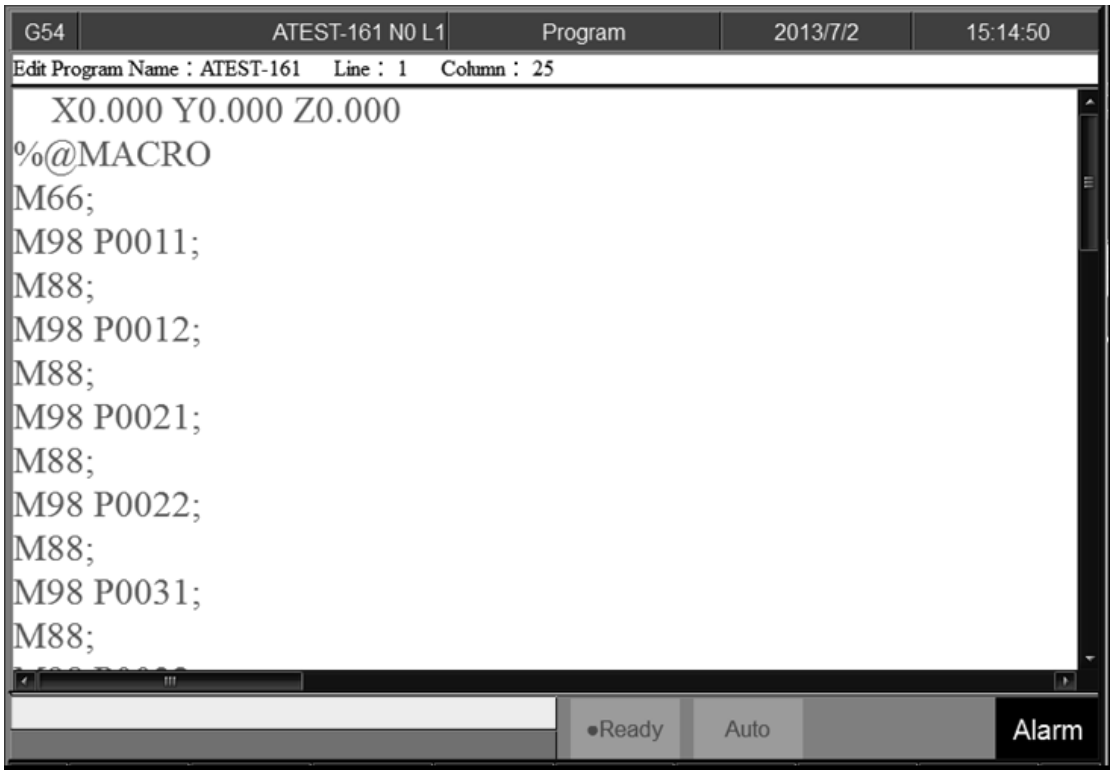
1.3.4 Clear Rel. Coord.

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

1.3.5 Clear All Rel. Coord.

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

1.4 Program



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

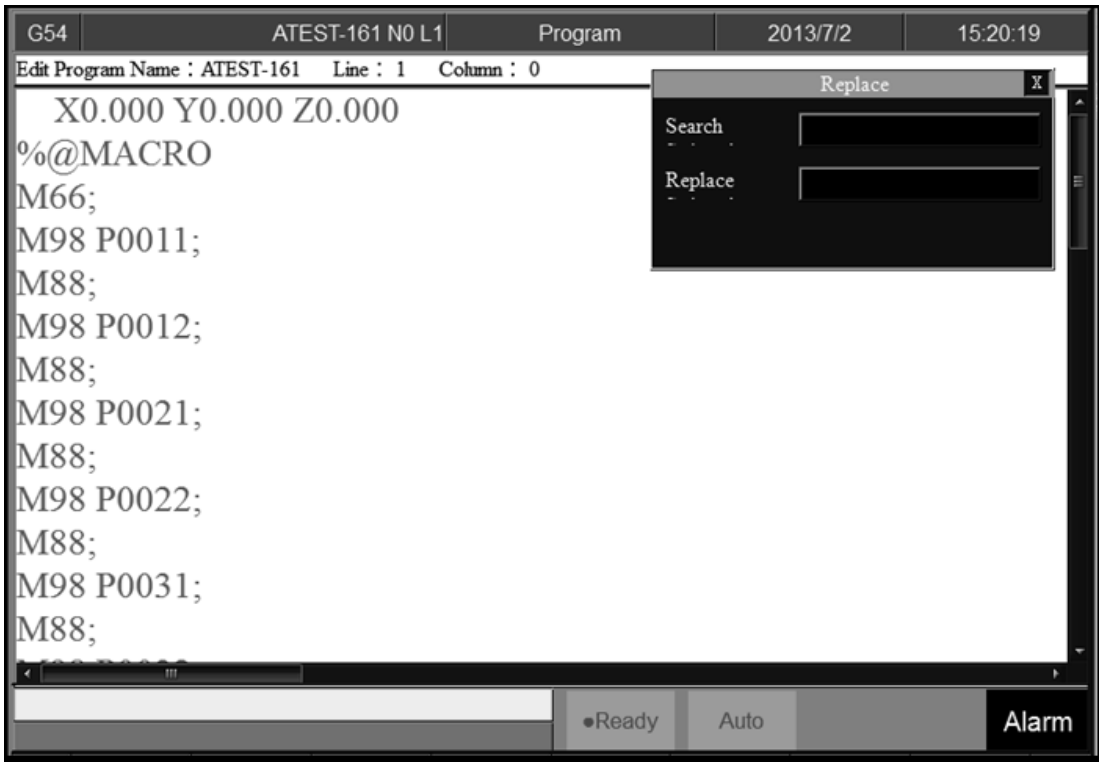
1.4.1 Execute

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

1.4.2 Delete Line

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

1.4.3 Search/Replace



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

1.4.3.1 Find Next

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

1.4.3.2 Replace

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

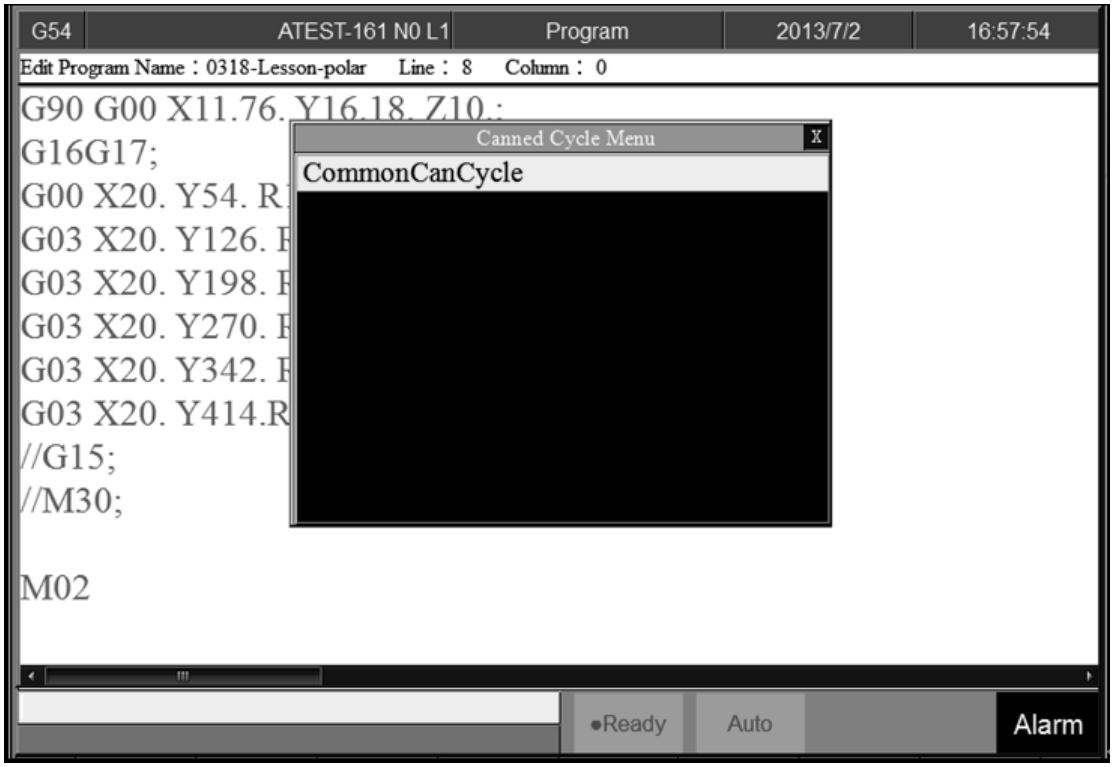
1.4.3.3 Replace All

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

1.4.3.4 Modify Setting

- Command
  - ✧ F2 Program→F3 Search/Replace→F4 Modify Setting
- Function
  - ✧ Reset 「Search/Replace」 string.
- Operation Method
  - ✧ Press F4「Modify Setting」to reset the「Search/Replace」content.

1.4.4 Can Cycle



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

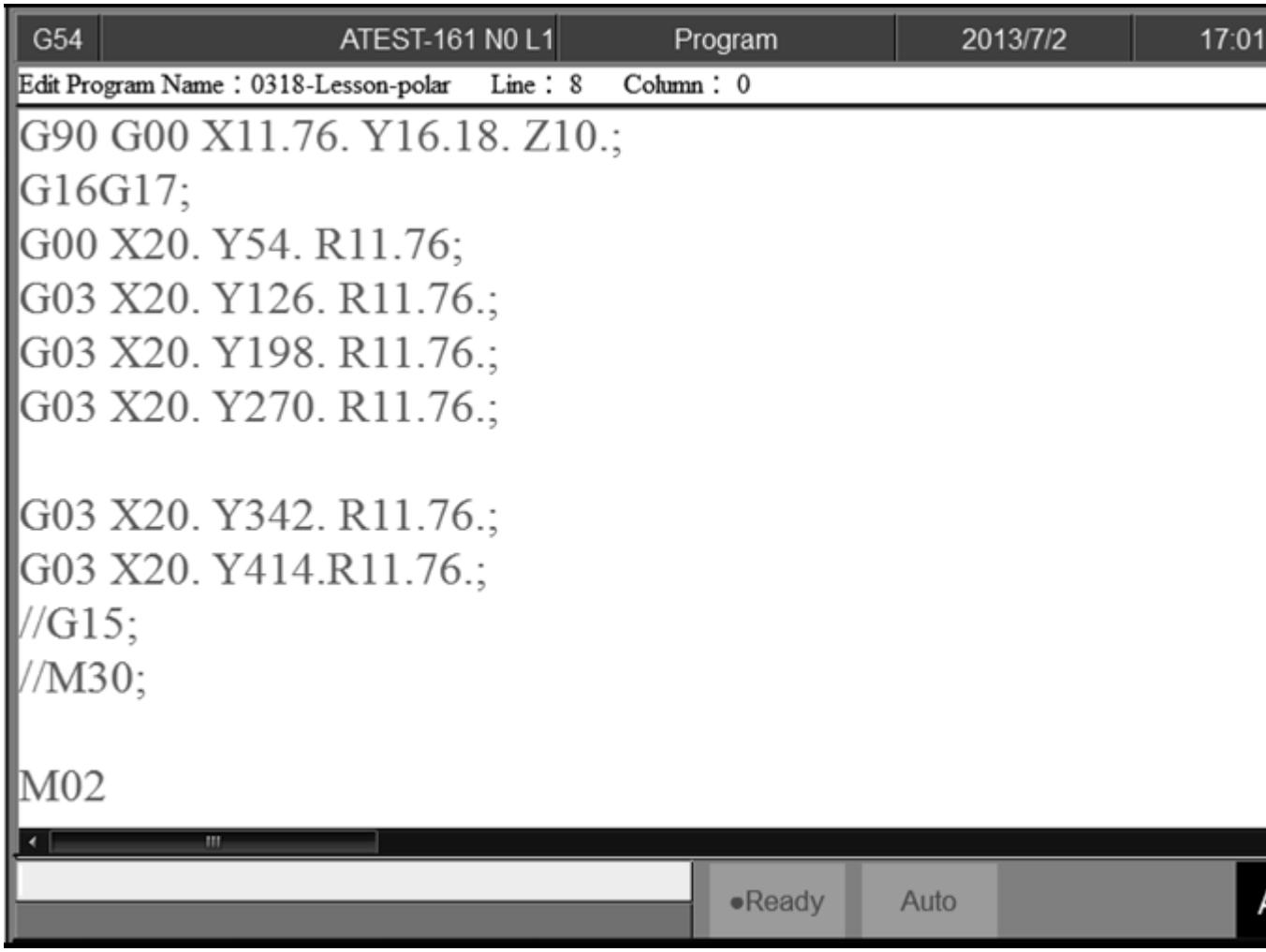
1.4.4.1 Insert Cycle

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

1.4.4.2 Edit Cycle

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

1.4.5 Block Copy



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

1.4.5.1 Start Line

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

1.4.5.2 End Line

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

1.4.5.3 Block Cut

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

1.4.5.4 Block Copy

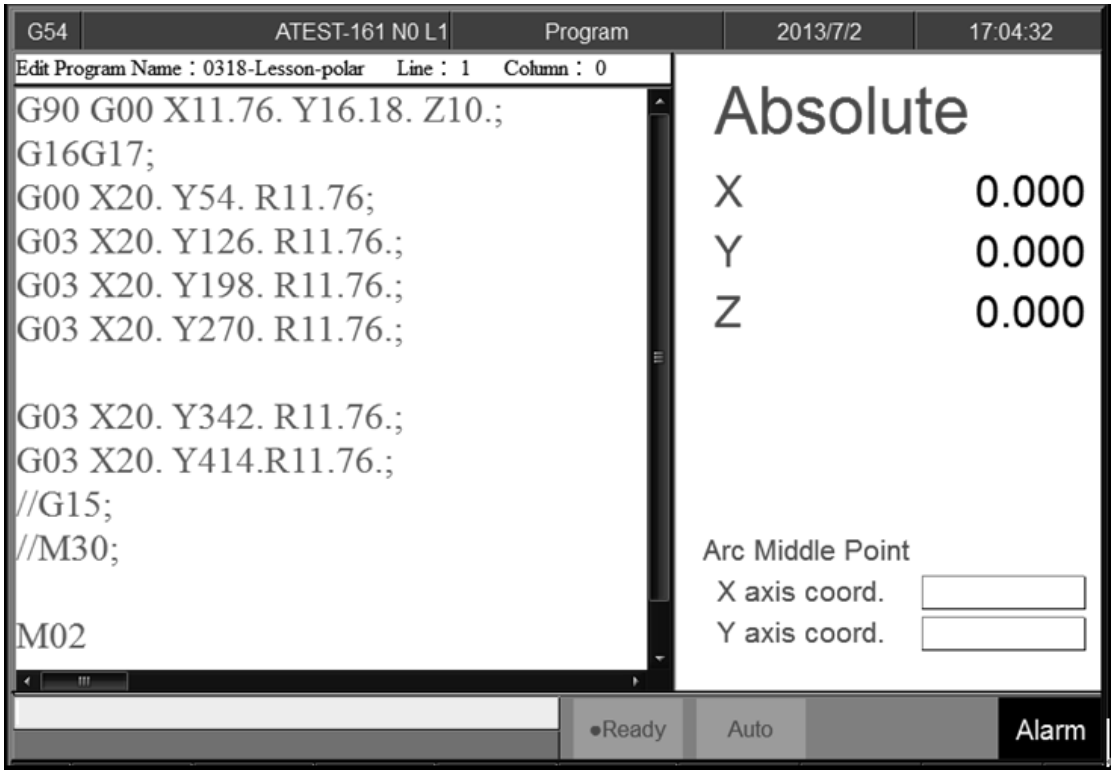
- Command
  - ✧ F2 Program→F5 Block Copy→F4 Block Copy
- Function
  - ✧ Copy the block that had been selected.

1.4.5.5 Block Paste

- Command
  - ✧ F2 Program→F5 Block Copy→F5 Block Paste
- Function
  - ✧ Paste the block that had been 「Block Cut」and「Block Copy」.

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

1.4.6 Teach



- Command
  - ✧ F2 Program→F6 Teach
- Function
  - ✧ Move the machine table with 『MPG』/『JOG』/『INJOG』 to the destination and use 『Teach』function, input the current absolute coordinate value to NC program.
  - ✧ Omit the manual input problem.

1.4.6.1 Rapid Teach

- Command
  - ✧ F2 Program→F6 Teach→F1 Rapid Teach
- Function
  - ✧ Add the current absolute coordinate as the value of 「G00 Rapid Traverse」function in current program.

1.4.6.2 Line Cut Teach

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

1.4.6.3 Arc Cut Teach

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

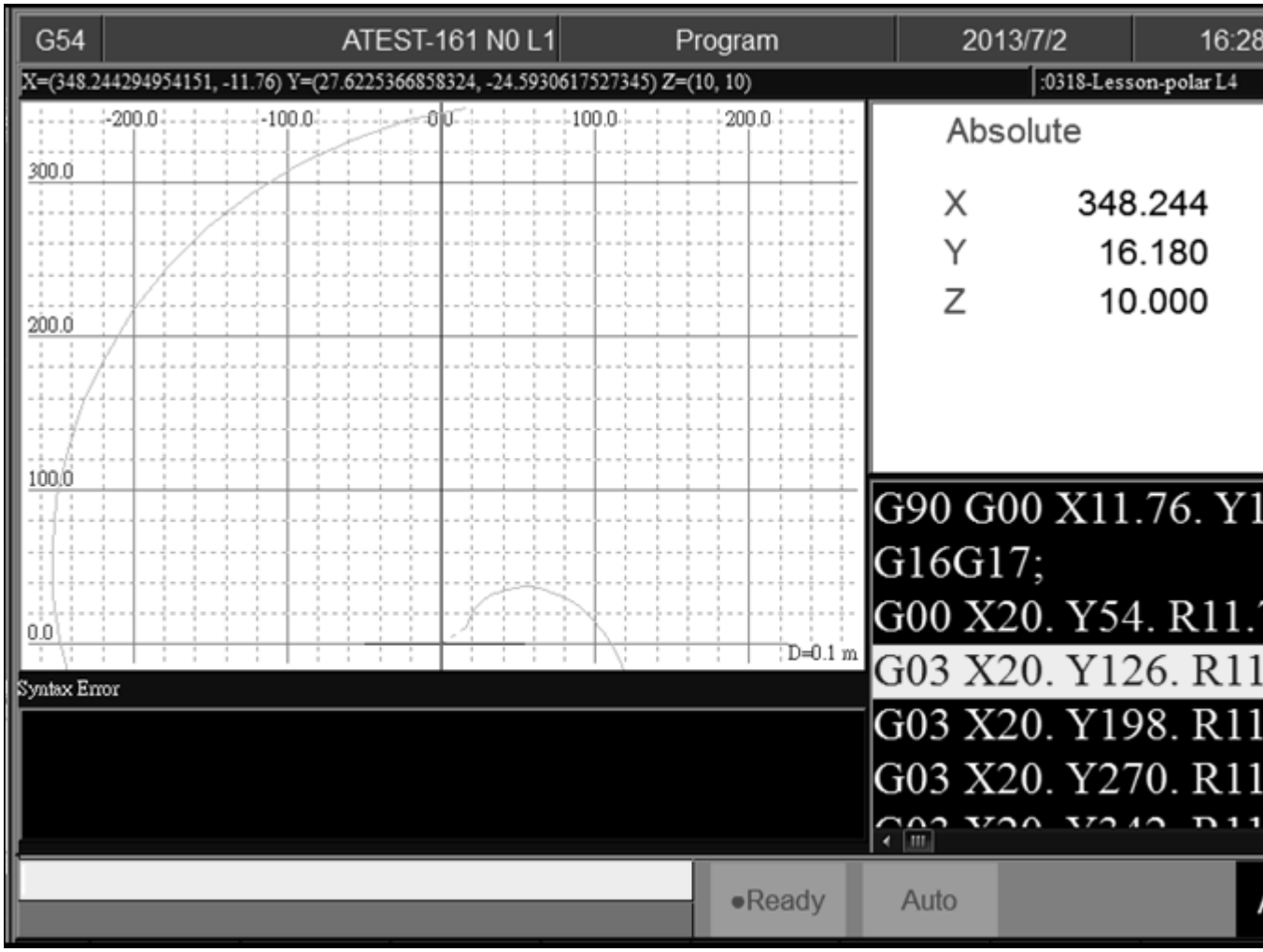
1.4.6.4 Cancel Arc Middle

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

1.4.6.5 Point Teach

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

1.4.7 Simulation



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

1.4.7.1 Step

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

1.4.7.2 Continue

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

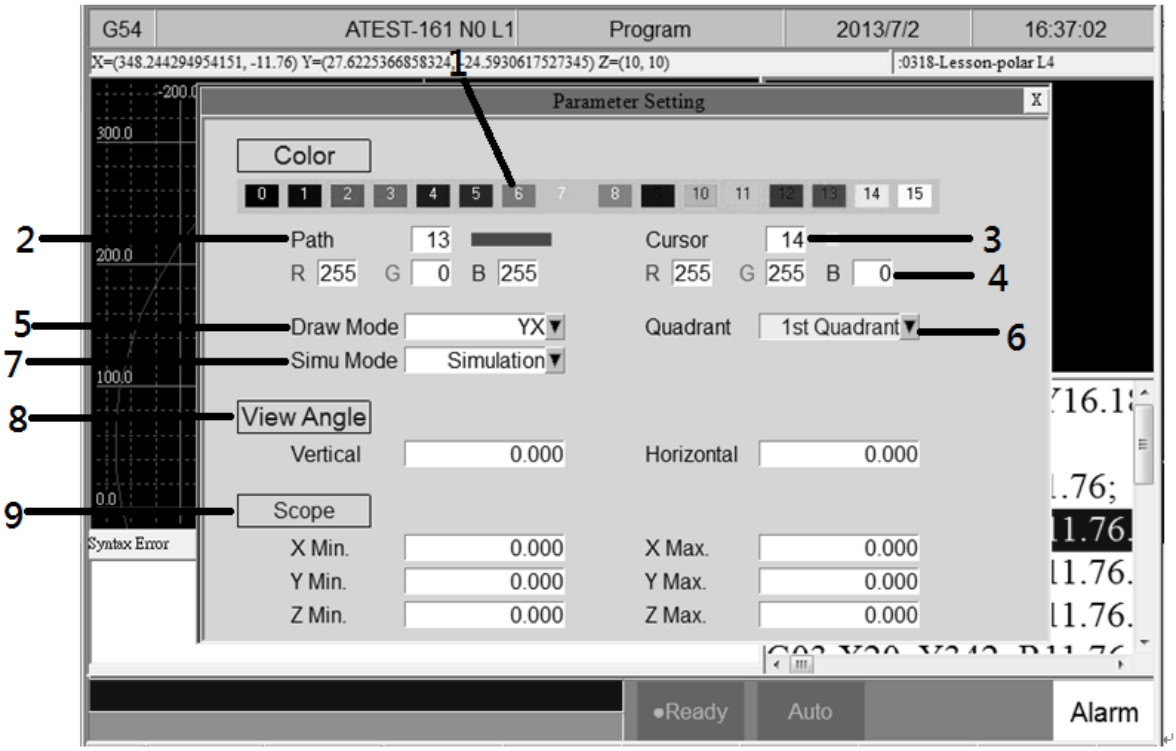
1.4.7.3 Zoom

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

1.4.7.4 Graph Reset

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

1.4.7.5 Simu. Setting



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

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

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

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

7. Simulate Mode

   - ◆ Setting profile simulate method.

   - Simulation
     - ◆ When user go to the 『Monitor』 page, simulation will show up automatically.
     - ◆ System will scan the whole program and detect the simulation boundary, and then simulation will execute. No need to define the simulate boundary.
   - Direct Draw
     - ◆ When user go to the 『Monitor』 page, cursor will show up but simulation will not execute automatically.
     - ◆ User need to define the simulation boundary first.
     - ◆ When the machining starts, cursor will follow up as well.
   - Not Simulation
     - ◆ Close the simulation function.

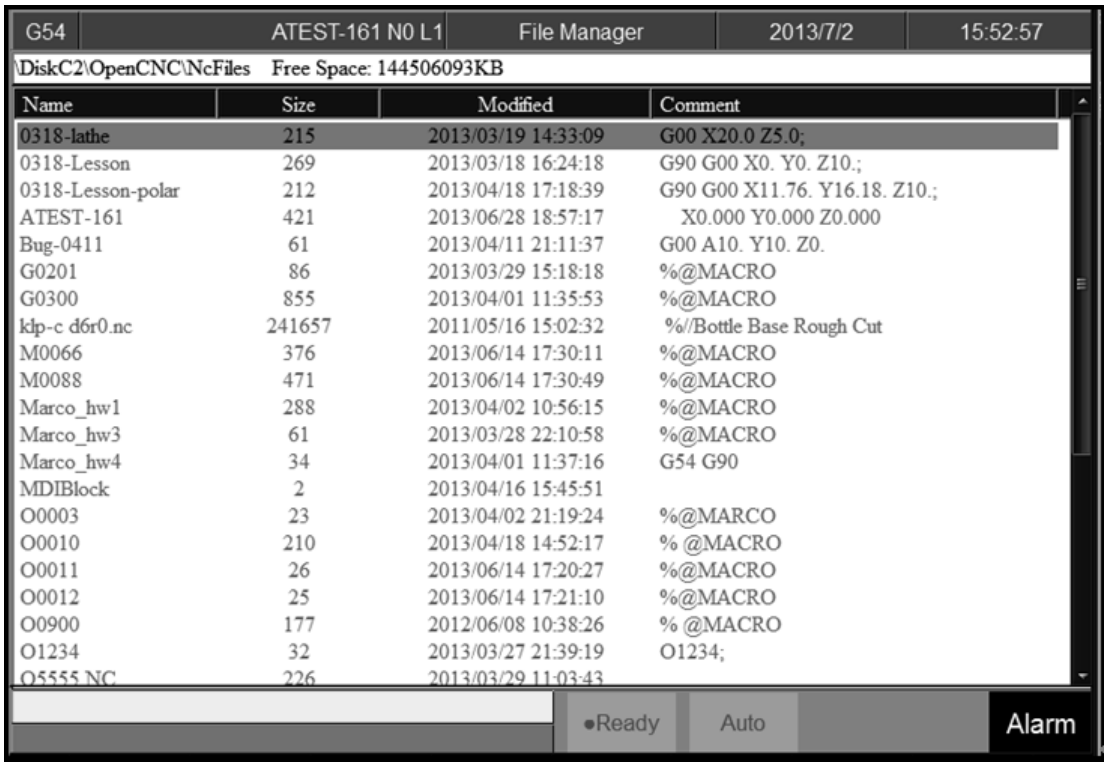
8. View Angle Setting

   - ◆ Under XYZ draw mode, by setting this parameter, simulation will show up with 3D simulation.
   - ◆ View angle can define are as below.
     - ◆ Vertical
     - ◆ Horizontal

9. Scope

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

1.4.8 File Manager



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

1.4.8.1 New File

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

1.4.8.2 Copy File

- Command
  - ✧ F2 Program→F8 File Manager→F2 Copy File
- Function
  - ✧ Copy the file that remarked by cursor.
- Operation Method
  - ✧ Use【↑】【↓】to move the cursor to the file that want to copy.
  - ✧ Press「Copy File」function key.
  - ✧ A dialog box will appear, enter the new file name.
- Note
  - ✧ Default file name has no file extension. If user want to create a new file with file extension such as *.NC, just enter the extension (*.NC) as well.

- ✧ The length of file name cannot be longer than 32 characters (include file extension)

1.4.8.3 Delete File

- Command
  - ✧ F2 Program→F8 File Manager→F3 Delete File
- Function
  - ✧ Delete file that remarked by cursor.
- Operation Method
  - ✧ Press F3「Delete File」, check box will show up in front of the NC file within the「File Manager」monitor page. Use【↑】【↓】 to select the delete file.
- Sub-function Key
  - ✧ Select: Select file, can select more than one file and also can cancel the selection of one file.
  - ✧ Select All: Select all file.
  - ✧ Cancel Select: Deselect all files.
  - ✧ Delete File: Delete all of the selected files.
  - ✧ Delete All: Delete all file within data storage device.
- Note
  - ✧ Current Programming and machining file cannot be deleted.

1.4.8.4 File Import

- Command
  - ✧ F2 Program→F8 File Manager→F4 File Import
- Function
  - ✧ Import outer file into controller
- Function Page Explanation
  - ✧ The upper block shows the outer devices selection with the following choice.
    - ■ USBDisk
    - ■ DiskA
    - ■ Network
    - ■ USBDisk2
  - ✧ Left column shows data structure of the outer device.
  - ✧ Right column shows data structure of the inner storage of the controller.
- Sub-function Explanation
  - ✧ Copy: Copy the remarked file from the outer device to the controller.
  - ✧ Select: Select or deselect each file. (Not available for folder)
  - ✧ Select All: Select all files.
  - ✧ Cancel Select: Deselect all files.
  - ✧ Device Change: Change outer device selection.
- Operation Method
  - ✧ Press F1「File Import」, a dialog box will appear.
  - ✧ Default outer device is USB Disk.
  - ✧ If you want to change the outer device, press F5「Device Change」,switch the cursor to the desire device and press【Enter】,

then the left column data structure will change and show the data structure of selected device.

✧ Use【↑】【↓】to select file.

✧ Move the cursor to the import file and press F2「Select」or 【Space】to remark file.

✧ After remarked all of the import file, press F1「Copy」then all of the remarked file will import into controller.

1.4.8.5 File Export

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

then the below left column data structure will change, according to the selected device .

✧ Use【↑】【↓】to select file.

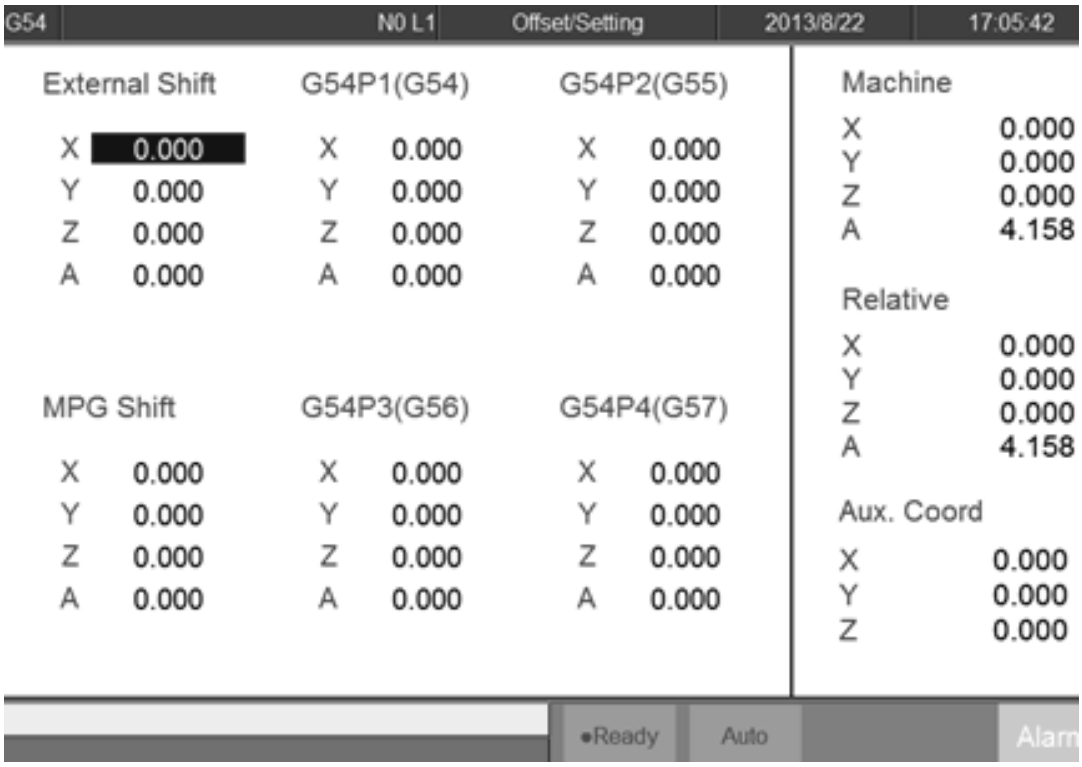
✧ Move the cursor to the export file and press F2 「Select」or 【Space】to remark file.

✧ After remarked all of the export file, press F1「Copy」 then all of the remarked file will export from controller to outer device.

1.4.8.6 Execute

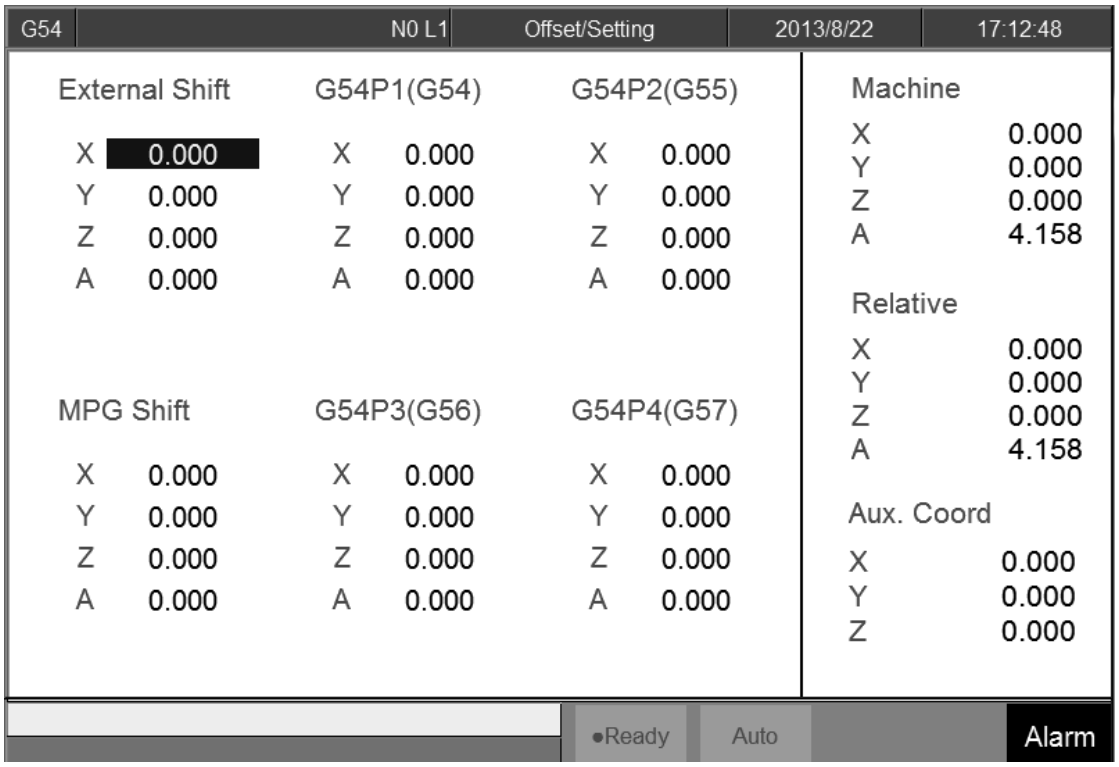
- Command
  - ✧ F2 Program→F8 File Manager→F8 Execute
- Function
  - ✧ Execute current program. and also change the screen to the 「monitor」 page.
- Operation Method
  - ✧ Use【↑】【↓】to select file, and then press「Execute」the selected file will be the executive file.
- Note
  - ✧ This function is invalid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

1.5 Offset/Setting



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

1.5.1 Workpiece Cord.



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

1.5.1.1 Latch Machine Cord.

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

1.5.1.2 Latch Relative Cord.

- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F2 Latch Ralative Cord.
- Function
  - ✧ Set current relative coordinate value into work piece coordinate.
- Operation Method
  - ✧ Move the worktable to the target location.

- ✧ Move the cursor to the relate work piece coordinate and press 「Latch Relative Cord.」
- ✧ Original value of workpiece coordinate will be replaced by the corresponding relative coordinate.

- Example
  - ✧ Current relative coordinate of X axis is 5.000
  - ✧ Current value of X axis of G54 is 0.000
  - ✧ Move the cursor to G54 X axis.
  - ✧ Press 「Latch Relative Cord.」,the coordinate of G54 X axis become 5.000

1.5.1.3 Latch Aux. Cord.

- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F3 Latch Aux. Cord.
- Function
  - ✧ Set the current cursor located work piece coordinate value as the aux. coordinate value.
  - ✧ Aux. value appears after using middle function.
- Operation Method
  - ✧ Using middle function to calculate the aux. coordinate value.
  - ✧ Move the cursor to the relate work piece coordinate and press 「Latch Aux. Cord.」
  - ✧ Original value of workpiece coordinate will be replaced by the corresponding Aux. coordinate.
- Example
  - ✧ Current mechanical coordinate of X axis is 5.000
  - ✧ Current value of X axis of G54 is 0.000
  - ✧ Move the cursor to G54 X axis.

✧ Press 「Latch Aux. Cord.」, the value of X axis of G54 become 5.000

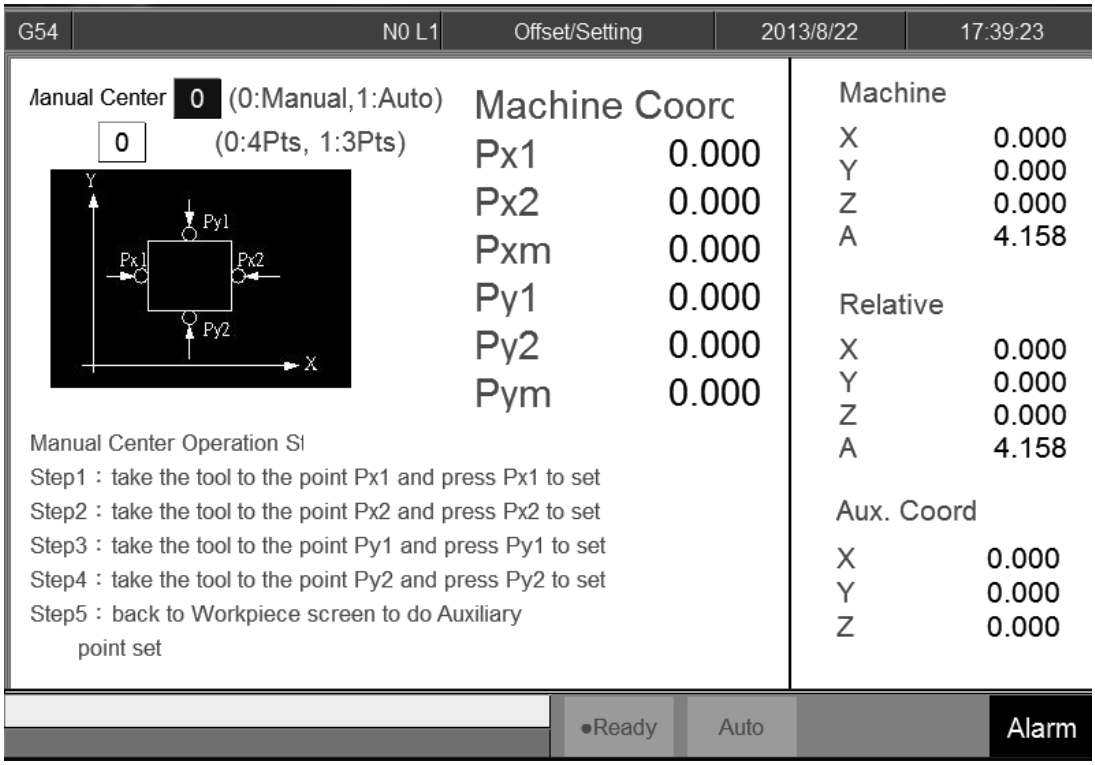
1.5.1.4 Coord. Increment Input

- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F4 Coord. Increment Input
- Function
  - ✧ Add the work piece coordinate value with the manual input value and restore into work piece coordinate again.
- Operation Method
  - ✧ Move worktable to the target location.
  - ✧ Input increment value.
  - ✧ Move the cursor to the work piece coordinate and press 「Coord. Increment Input.」
  - ✧ Work piece coordinate will be replaced by the cursor located coordinate +input value.
- Example
  - ✧ Current mechanical coordinate of X axis is 5.000
  - ✧ Move cursor to the workpiece coordinate G54 of X axis
  - ✧ Key in 10.000
  - ✧ Press 「Coord. Increment Input.」
  - ✧ The coordinate of G54 X axis become 15.000

1.5.1.5 Mach. Coord Teach Input

- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F5 Mach. Coord Teach Input
- Function
  - ✧ By the Mach. Coord Teach Input, set the current cursor located work piece coordinate value as the new start coordinate value.
- Operation Method
  - ✧ Move the worktable to the destination.
  - ✧ Move the cursor to the relate work piece coordinate and press 「Mach. Coord Teach Input.」
  - ✧ Current cursor located work piece coordinate will be replaced by the cursor coordinate+input number.
- Example
  - ✧ Current mechanical coordinate of X axis is 5.000
  - ✧ Current workpiece coordinate G54 of X axis
  - ✧ Key in 10,000
  - ✧ Press 「Mach. Coord Teach Input.」
  - ✧ the coordinate of G54 X axis become 15.000

1.5.1.6 Middle Func.



- Command
  - ✧ F3 Offset/Setting→F1 Work Piece Cord.→F6 Middle Func.
- Function
  - ✧ This function can help to correlate the middle point of the work piece.
  - ✧ Input the number that Middle Func caculate result to G54~G59 work piece coordinate.
  - ✧ Including manual and auto setting method.

1.5.1.6.1 Manual

- Function
  - ✧ Set the Middle Func. as 0
  - ✧ User controls the machine by MPG, and then moves the tip of 3D machine to the X&Y end point of the workpiece. This system will calculate the center point of the workpiece automatically.
- Operation method

- ✧ Move the machine by MPG, touching Px1 point in tis figure and then press PX1 Set. The system will record the current mechanical coord. to Px1. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm and Aux. X position.
- ✧ Move the machine by MPG, touching Px2 point in this figure and then press PX2 Set. The system will record the current mechanical coord. to Px2. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm and Aux. X position.
- ✧ Move the machine by MPG, touching Py1 point in tis figure and then press PY1 Set. The system will record the current mechanical coord. to Py1. It will also compute the middle point of Py1 and Py2 and puts the result on Pym and Aux. Y position.
- ✧ Move the machine by MPG, touching Py2 point in this figure and then press PY2 Set. The system will record the current mechanical coord. to Py2. It will also compute the middle point of Py1 and Py2 and puts the result on Pym and Aux. Y position.
- ✧ Now, the values of Pxm and Pym are the middle point of the workpiece.
- ✧ In the WorkPiece Cord. Screen, move the cursor to the coord you want to set, press F3 insert the Aux. Coord value then this system will set the value according to the Aux Coord.

1.5.1.6.2 Auto

- Function
  - ✧ Set the Middle Func. as 1
  - ✧ Auto middle func. is different form Manual func. User only needs to enter the dimension of the workpiece and enter the boundary

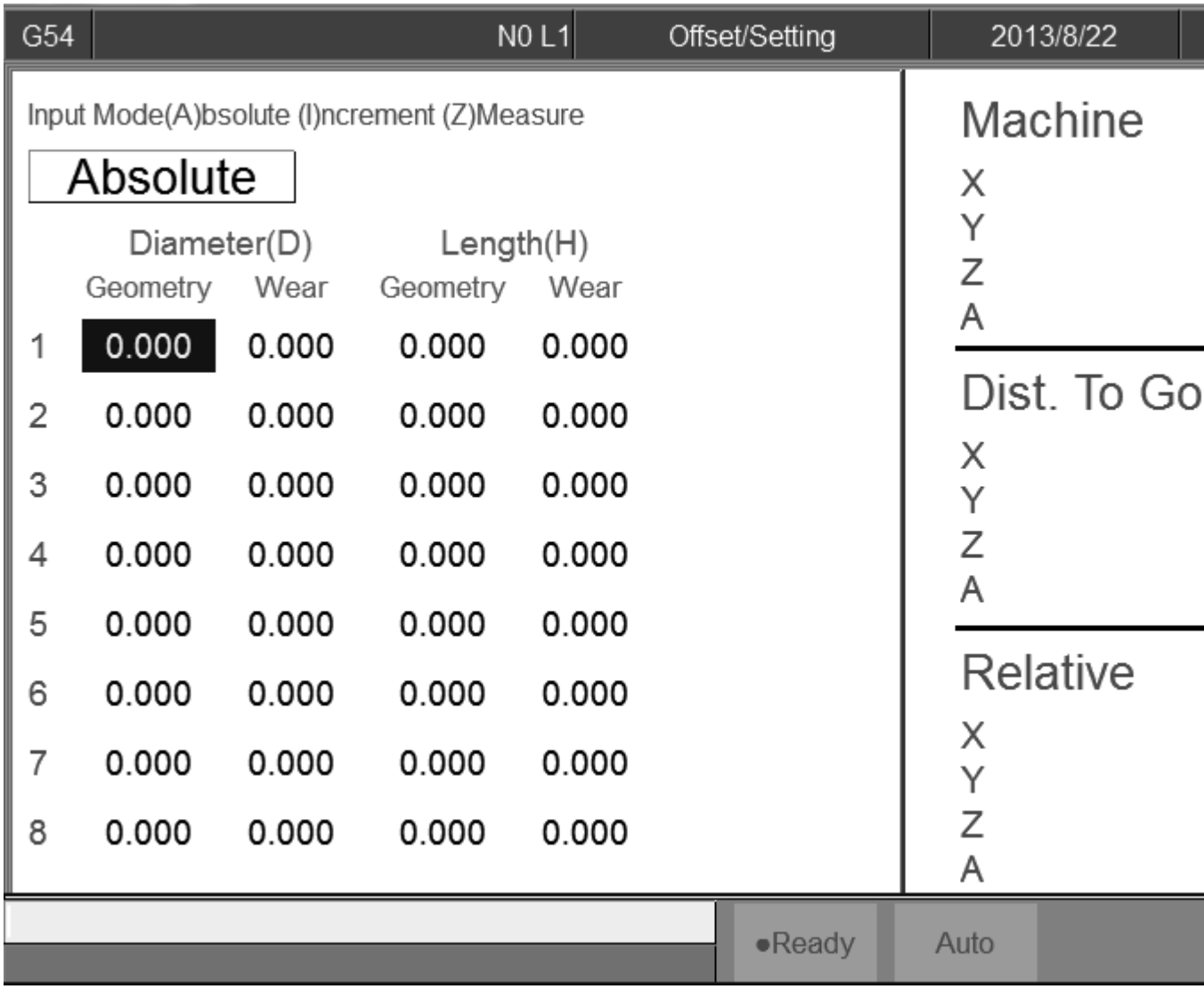
coordinate. Move the machine to the start point, system will find out the middle point automatically.

- ✧ Auto middle func. is different form Manual func. User only needs to enter the dimension of the workpiece and enter the boundary coordinate. Move the machine to the start point, system will find out the middle point automatically.

- Parameter description
  - ✧ Length I: Workpiece X dimension length
  - ✧ Width J: Workpiece Y dimension length
  - ✧ Safe Distance H: This is the distance between start point P2 and the workpiece no matter X or Y direction.
  - ✧ Feedrate F: Auto. center detect speed.
  - ✧ Z Coordinate: This is the safety distance prevent the contact between the tool and work piece while moving.
- Operation method
  - ✧ Move the tool to the Z height, the safe distance between tool and work piece. After that, press Z Coordinate Set, then system will record the present Z coordinate value as the safe distance.
  - ✧ Move the tool down to P2 under the work piece, the start point for auto center.
  - ✧ Press Auto center start, system will move the tool according to the setting data to contact with work piece and show the coordinate on the screen. It will also calculate the X Y center point of the work piece.

- ✧ Go back to WorkPiece Cord. Move the cursor the workpiece coordinate position, press F3 insert the Aux. Cord value then this system will set the value according to the Aux Cord.

1.5.2 Tool Set



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

- ✧ Wear of geometry: Tiny geometry modification of tool.
- ✧ Length: G43/G44 tool lengths Hn compensation.
- ✧ Length wear: Tiny length modification of tool.

- Operation method
  - ✧ With 【↑】【↓】【←】【→】key to move the cursor.
  - ✧ 【PageUp】【PageDown】 to switch the page.
  - ✧ Key in method: Absolute, Increment, Measure method
- Absolute method:
  - ■ Press 「A」 and press 「Enter」
  - ■ The value where the cursor is will be set as the input value.
- Increment method
  - ■ Press 「I」 and then press 「Enter」.
  - ■ The value where the cursor is will be set as input value + cursor value.
- Measure method
  - ■ Press 「Z」 and then press 「Enter」.
  - ■ The value where the cursor is will be set as current Z coord relative coord value.
- Note
  - ✧ When the tool length had been set, related tool wear will become zero.
  - ✧ This Setting is disabled during machining condition.

1.5.2.1 Clean Z relative coordinate

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

1.5.2.2 Tool No.

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

1.5.2.3 Tool Life Manag.

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

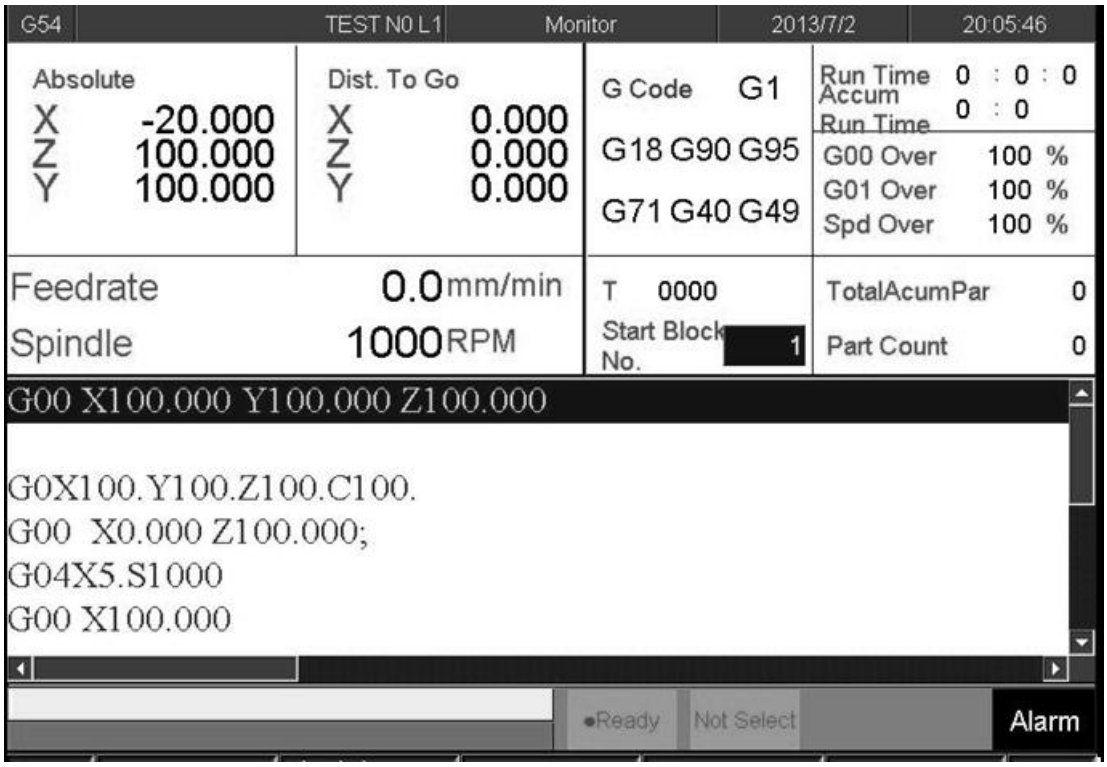
1.5.3 Tool Tip Measure

- ✧ F3 Offset/Setting→F3 Tool Tip Measure
- Function
  - ✧ When the machine is equip with tool detection equipment, via the parameter setting, can move the machine table to the detective location and execute the top tip measure.
  - ✧ According to different situation, here are three different method
    -Single tool, single workpiece
    -Single tool, multi workpiece
    -multi tool, multi workpiece
    -Refer the chapter 3.3.3.3.

1.5.4 User Parameter Setting

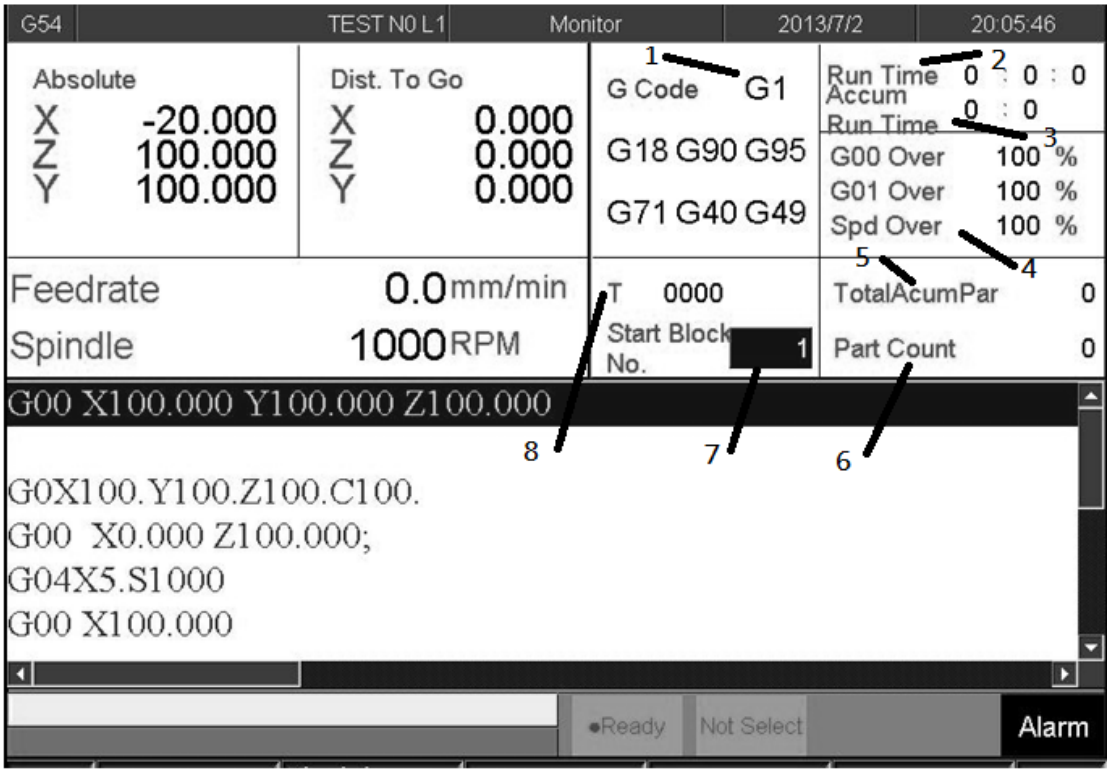
- Command
  - ✧ F3 Offset/Setting→F4 User Parameter Setting
- Function
  - ✧ SYNTEC controller provided user to set the related machining parameter by them self.
- Function of parameter
  - ✧ Detail explanation please reference「Mill Parameter Manual」.

1.6 Monitor



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

1.6.1 Monitor Area of Machining Information



1.6.1.1 Machine Status Monitor

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

1.6.1.2 Program Monitor Block

✧ This block will display current machining program

✧ Yellow bar indicate to the current running block.

1.6.1.3 Machining Information

- Function

  ✧ It is overlap with 「Process Setting」.

  ✧ Press 「Parameter Set」 can switch display information.

- Description

  1. G Code

     - It will show the G code under machining

2. Run Time Accum
    - Processing time for this workpiece now
3. Run Time
    - Total machining process time
4. Percentage ratio
    - G00 percentage
    - G01 percentage
    - Spindle speed percentage
5. Total Acum Par
    - Total work pieces that had been finished.
    - System won't do any initialized action automatically.
    - If you want to do the initialization by manual, press 「Parameter Set」 switch to 「Part count」, set the 「Total Acum Par」 as 0.Part Count
6. Part Count
    - Count no. will begin from zero when the program is running.
    - Total work pieces number machined by CNC
7. Start Block
    - We can set the start block of machining process.
    - n: Set the start line number as n. (Ex. 20)
    - L+n: Set the start line number as n. (Ex.L20)
    - N+n: Search N+n located line number and then assign that line as the start line (Ex. N3).
    - T+n: Search T+n located line number and assign it as the start line (Ex. T01).
    - If line number is out of max line number, then it will assign to the last line.
    - Please refer to 3.4.3.4 break point initialization about start block go back.

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

1.6.1.4 Display Area of Machining Setting

- Description
    - ✧ This area is overlap with 「Machining Information」, press F4 「Parameter Set」 to change the displays.
    - Explanation of Display:

1. Interrupt Line No.

- ✧ Display the last interrupted serial number (N)

2. Interrupt Colum No.

- ✧ Display the last interrupted line number (L)

3. Spindle speed

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

- Feed rate
    - ✧ Set the speed of the feed rate.。
    - ✧ It is allow to setting when system is busy, but the value will be updated after completely executing processing block.
- Total AcumPar
    - ✧ Total work pieces number machined by CNC

      System cannot automatically reset this value to zero
- Part count
    - ✧ Setting current work pieces no.
    - ✧ Count no. will begin from zero when the running program is change.

- ✧ When CNC executes M code defined by parameter 3804, part count would be added 1 and run time will be reset to 0. When required part number is reached, system will change to halt status.

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

1.6.1.5 Simulation Area

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

1.6.2 Open File to Edit

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

1.6.3 Simulation Switch

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

1.6.4 MDI Input

- Command:
  - ✧ F4 Monitor→F3 MDI Input
- Description
  - ✧ Manual Data Input. Using for simple NC program or testing purpose.
- Operation:
  - ✧ Select MDI mode
  - ✧ MDI function is enabled after finishing HOME search action.
  - ✧ Press F3 「MDI」, edit the program.
  - ✧ Press F1 (OK) to confirm the input command.
  - ✧ The command line will show up on the right upper corner of screen.
  - ✧ Press 【CYCLE START】to execute the command.
- Note:

✧ This function is enables under MDI mode.

1.6.5 Parameter Set

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

1.6.6 Tool Wear Set

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

1.6.7 Start MPG Coordinate

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

1.6.8 Work Record

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

1.6.9 Clear Acum Cycle Time

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

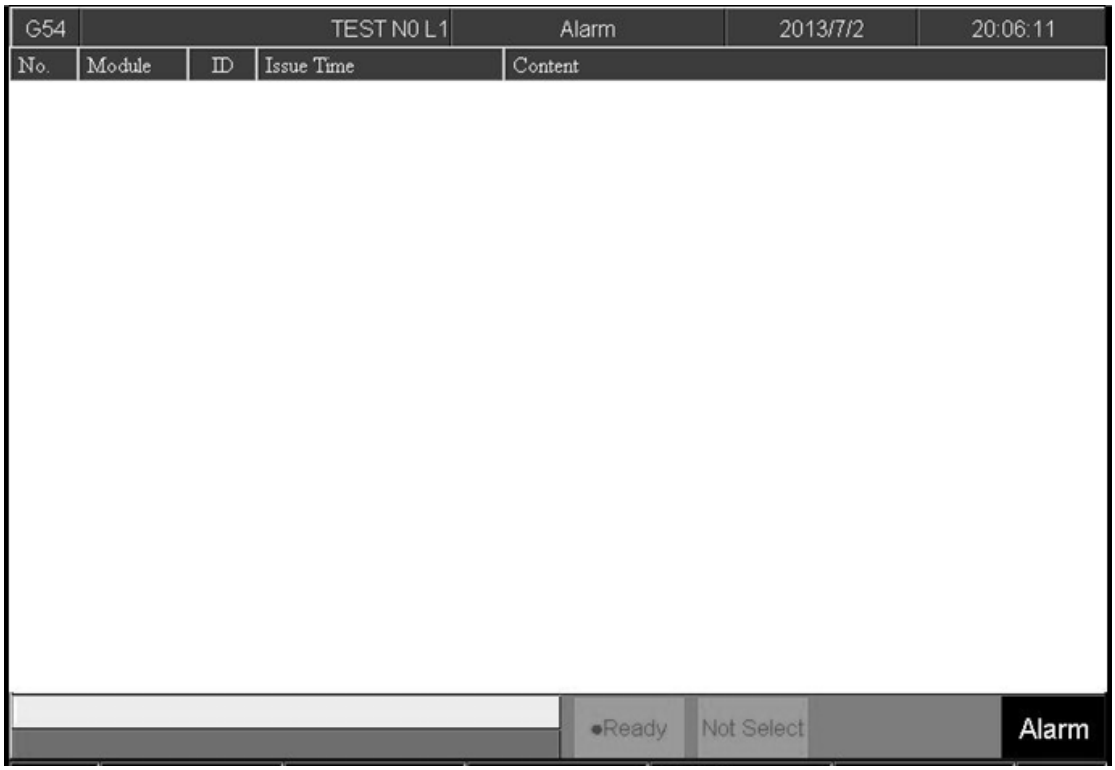
1.6.10 Graph Adjust

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

1.7 Maintain

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

1.7.1 Alarm



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

1.7.1.1 Pending Alarm

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

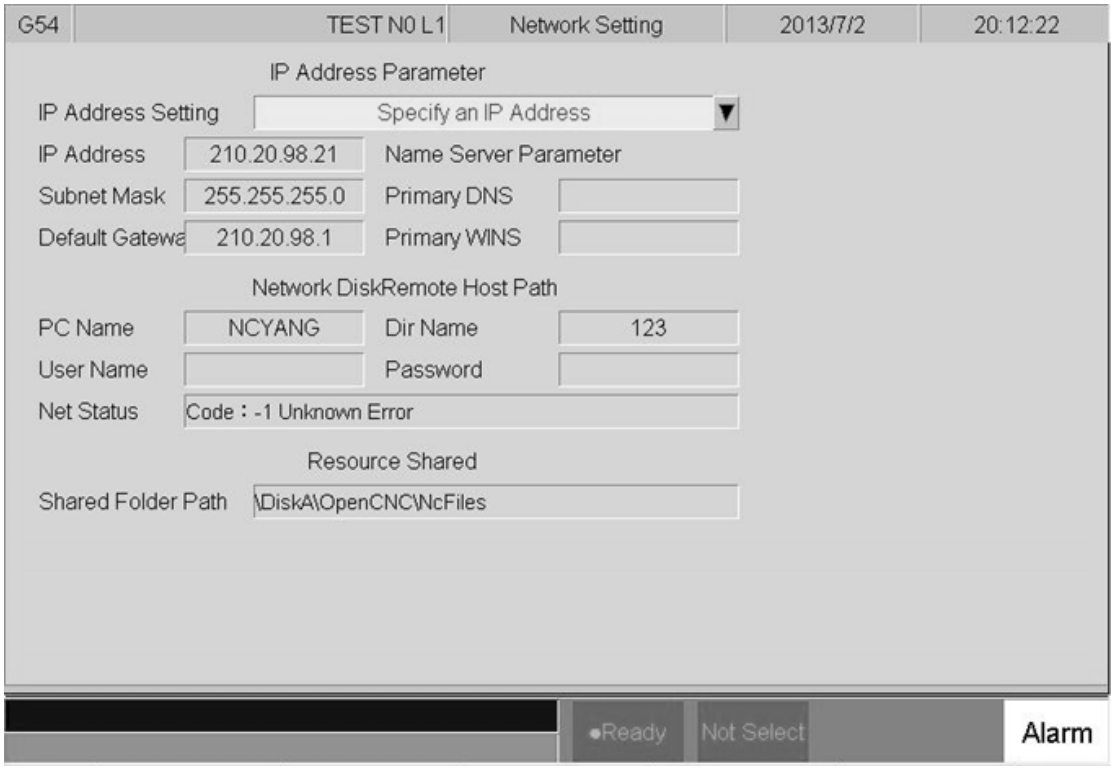
1.7.1.2 History Alarm

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

1.7.1.3 Save Alarm

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

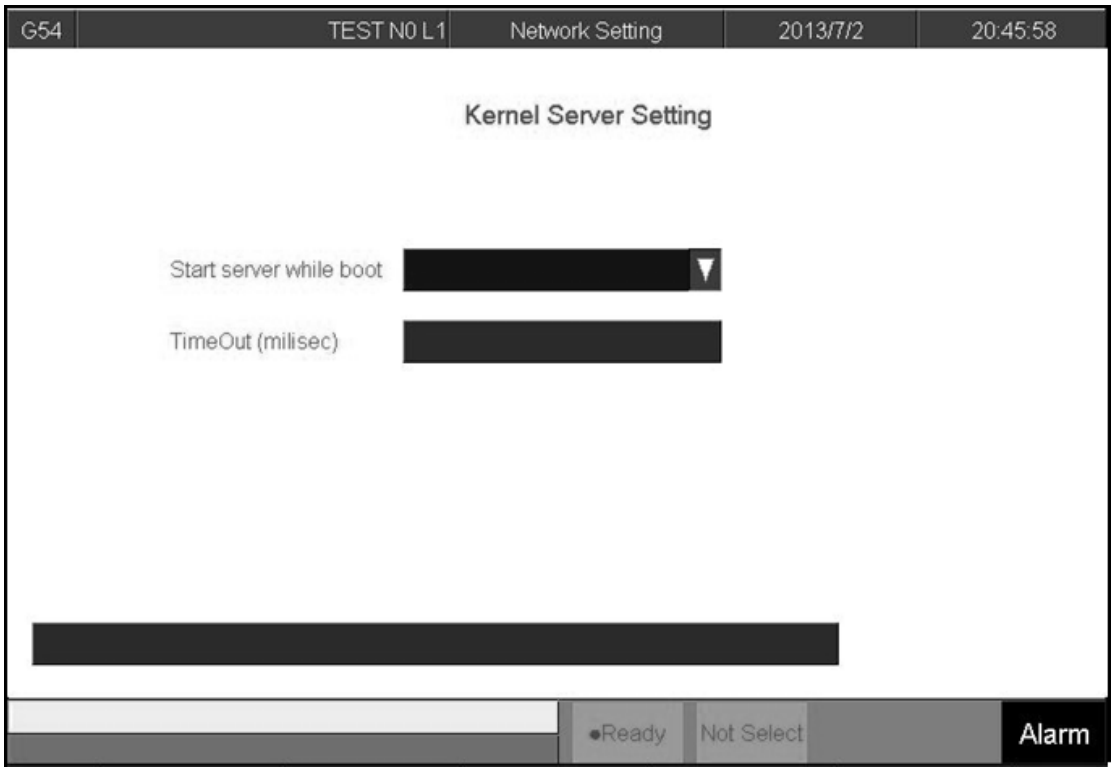
1.7.2 Network Setting



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

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

1.7.2.1 Set Kernel Server

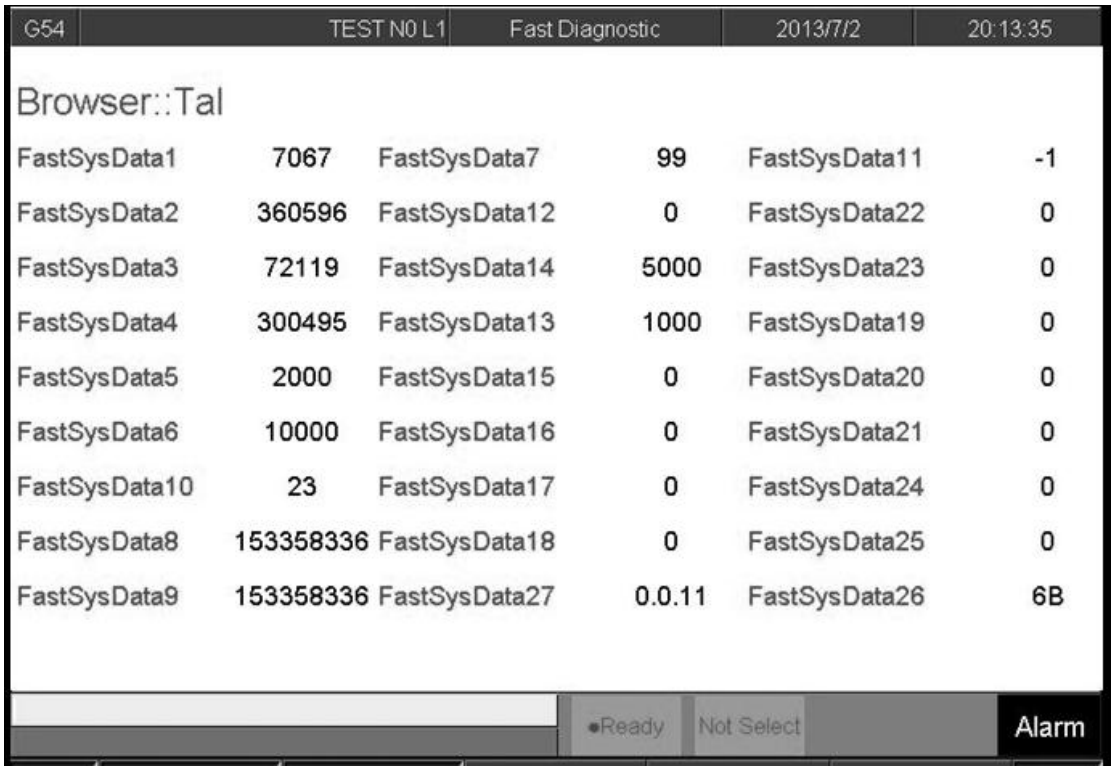


- Command:
  - ✧ F8 Maintain→F2 Network Setting→ F5 Set Kernel Server
- Description
  - ✧ Setting related function to kernel server
- Related infor.
  - ✧ Start server and kernel or not when power on.
  - ✧ Timeout(ms)
  - ✧ Set the acceptable time out when connecting to Kernel server unsuccessfully.

1.7.2.1.1 Start Server

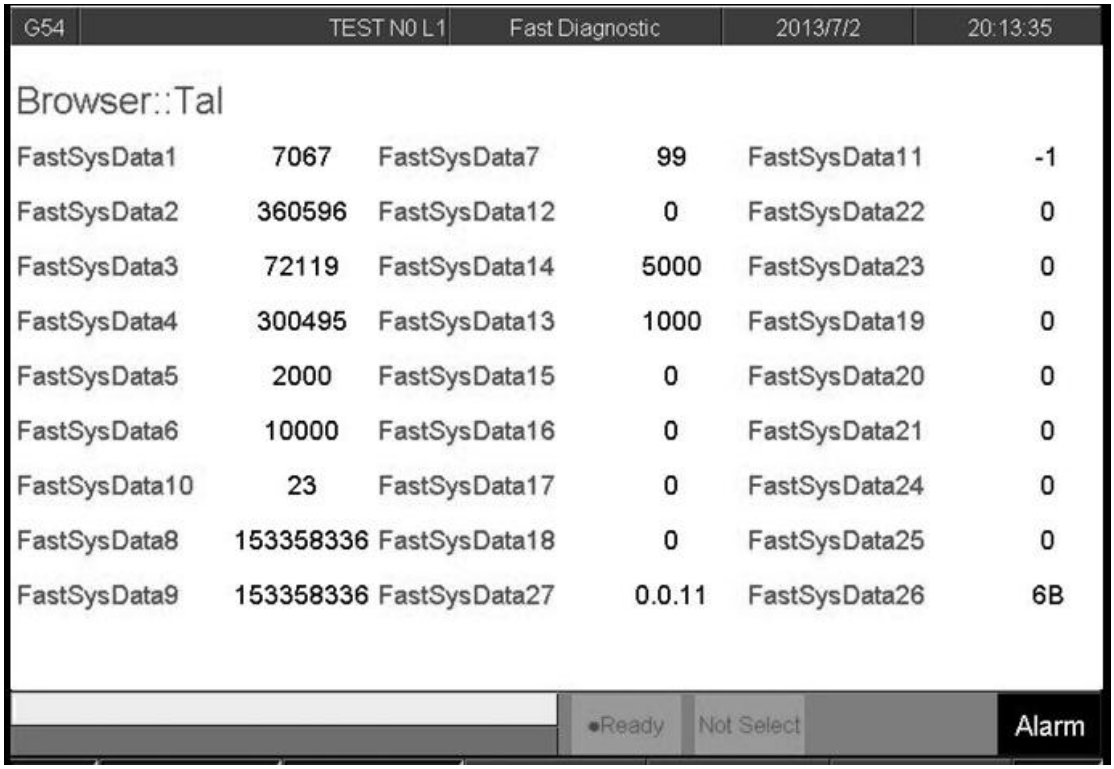
- Command:
  - ✧ F8 Maintain→F2 Network setting→ F5 Set Kernel Server→ F1 Start Server
- Description
  - ✧ Start server immediately

1.7.3 Fast Diagnostic



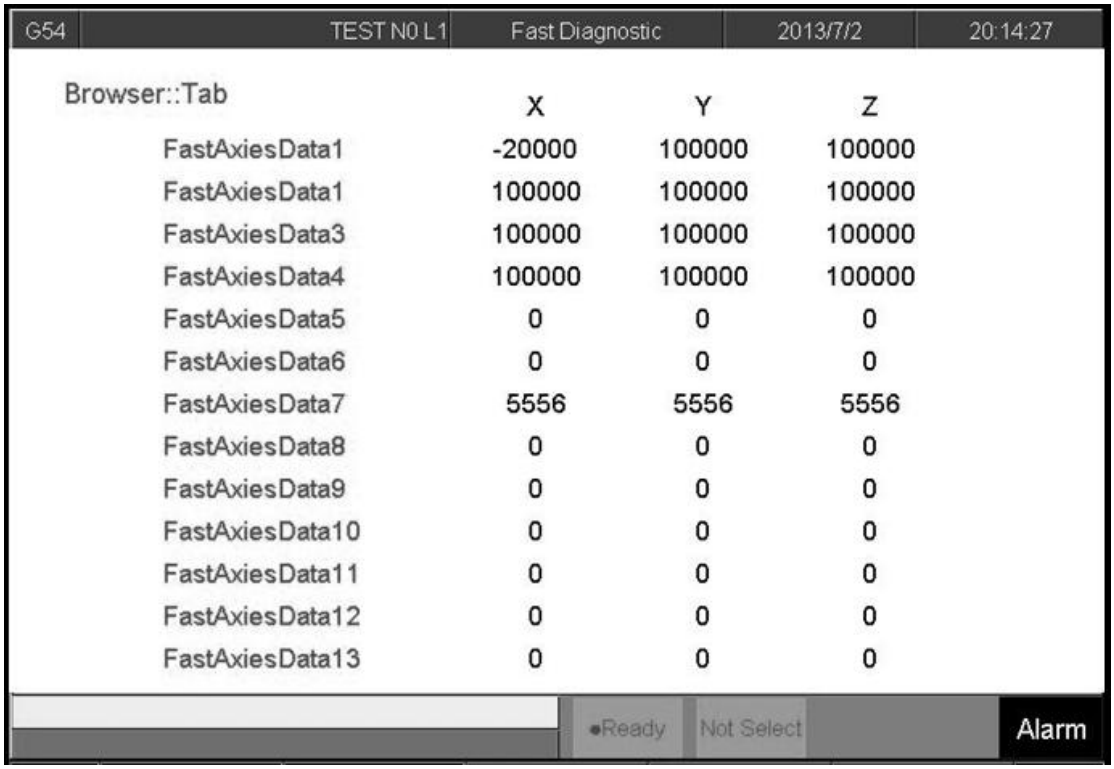
- Command:
  - ✧ F8 Maintain→F3 Fast Diagnostic
- Description
  - ✧ Display simple diagnostic information of system and axes

1.7.3.1 System Data



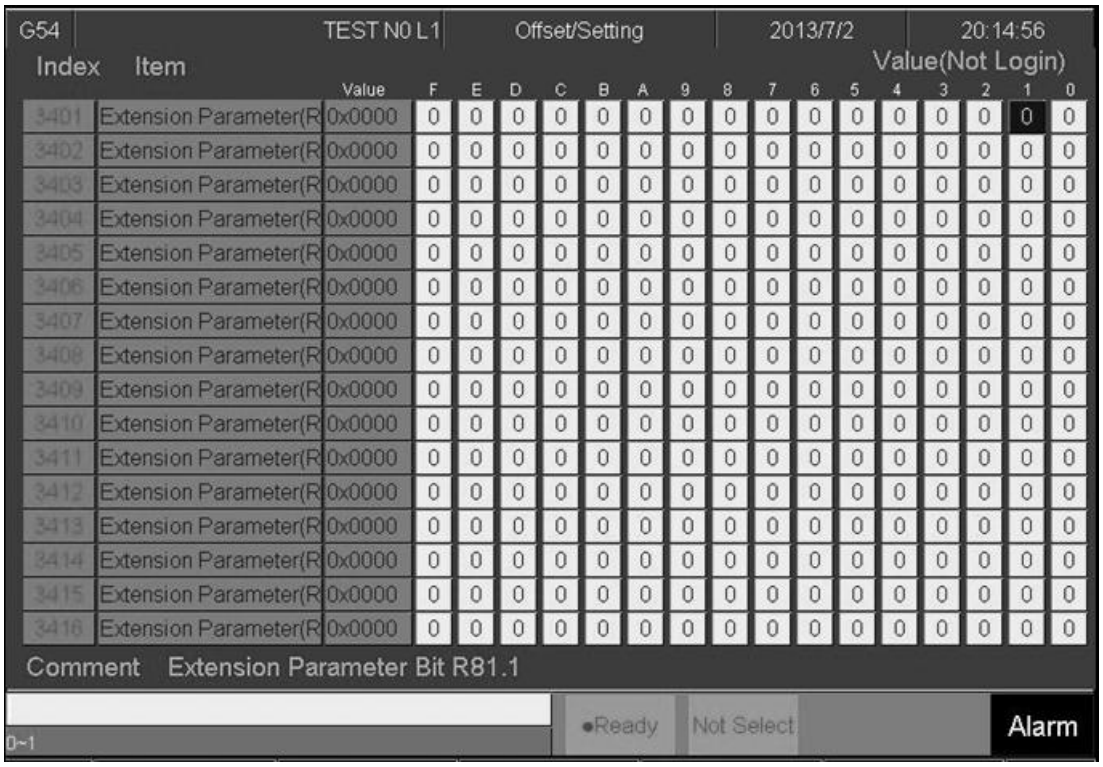
- Command:
  - ✧ F8 Maintain→F3 Fast Diagnostic→ F1 System Data
- Description
  - ✧ Display simple diagnostic information of system

1.7.3.2 Axes Data



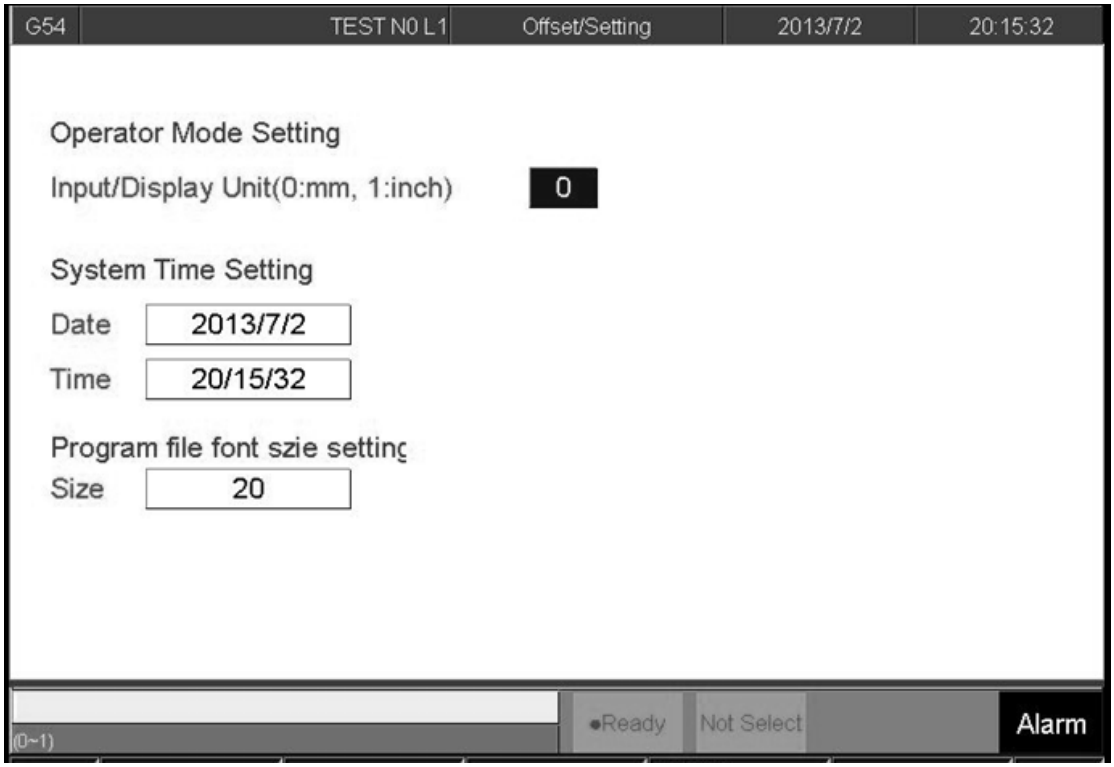
- Command:
  - ✧ F8 Maintain→F3 Fast Diagnostic→ F2 Axes Data
- Function
  - ✧ Display simple diagnostic information of Axes

1.7.4 PLC Param Setting



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

1.7.5 System Setting



- Command:
  - ✧ F8 Maintain → F5 System Setting
- Description
  - ✧ This page is used to set system environment
- Operation
  - ✧ Press【↑】【↓】【←】【→】to move cursor and 【PageUp】【PageDown】 go to next/previous page

1.7.5.1 Operator Mode Setting

✧ Setting system unit

■ 0：mm (Metric unit)

■ 1：Inch (British unit)

✧ Note: reboot to enable setting

1.7.5.2 System Time Setting

✧ Date: input format YYYY/MM/DD

◆ YYYY: year

◆ MM: month

◆ DD: day

✧ Time: input format HH/MM/SS

◆ HH: hours

◆ MM: minute

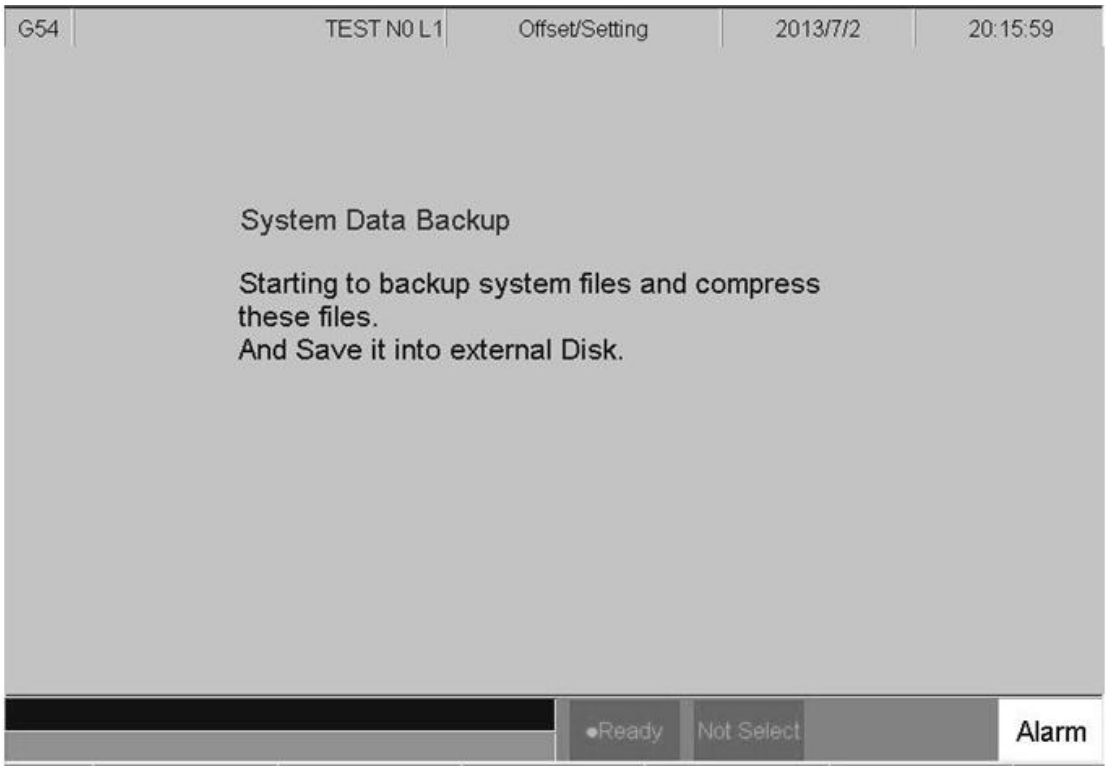
◆ SS: second

1.7.5.3 Program File Font Size Setting

✧ Setting display of font size in program

✧ Note: reboot to enable setting

1.7.6 Backup System



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

1.7.7 About

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

2 Machine Operation Panel

2.1 Operation Panel

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

■ **Home Mode**

- When CNC power is on, please implement home search

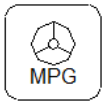
■ **JOG Mode (Rapid JOG)**

- Operators can use JOG to control axis movement with 「X+/X-/Y+/Y-/Z+/Z-」 on panel.

■ **Incremental JOG (IN JOG)**

- Operators can use JOG to control axis movement.

■ **MPG Mode**



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

■ **AUTO Mode**

- Users use this function to execute machining NC file

■ **MDI Mode**

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

■ **MPG Simulation**

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

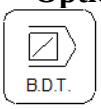
■ **Single Block**

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

■ **Optional Stop**

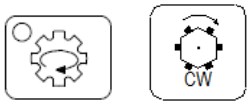
- Users can use this function to decide whether CNC is stop or not when encounter with 「M01」within NC (Numerical Control) program, while executing NC program.

■ **Optional Skip**

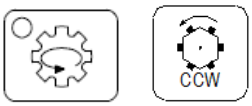


- Users can use this function to decide whether program skips or not when program is encountered with 「/」 sign in NC file.

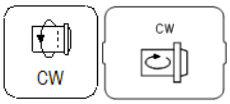
■ **Auto Tool Change in clock wise direction**



■ Auto Tool Change ATC in counter clock wise direction



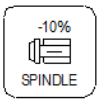
■ Spindle Control



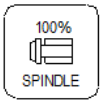
Spindle CW rotation

Spindle stop

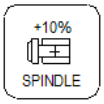
Spindle CCW rotation



Spindle rate deceleration: Spindle speed will decelerate 10%.



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

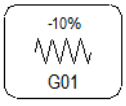


Spindle rate acceleration: Spindle speed will accelerate 10%.

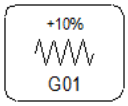
■ Working LED

- Turn ON/OFF working LED

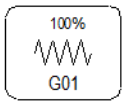
■ G01 Rank



Decrease G01 rate 10%

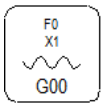


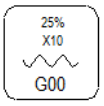
Increase G01rate10%

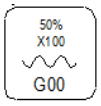


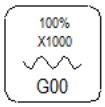
G01 rate100%

- **G00 Rank**









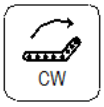
G00 rate x1 G00 rate x10 G00 rate x100 G00 rate x1000

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

- ON/OFF blowing air.
- **Cutting coolant system**

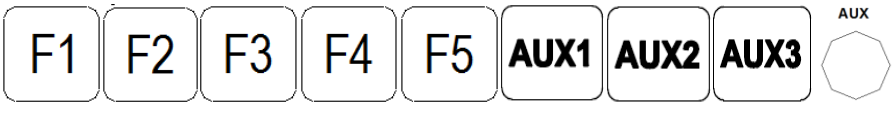
- Turn ON/OFF working liquid.
- **Auto cutting coolant system**

- This function will be enable, when encounter with relate coolant on/off M code in NC program.
- **Chip conveyor machine move in clockwise direction**

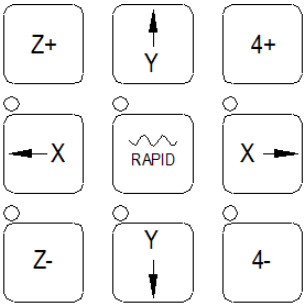
- **Chip conveyor machine move in counter clockwise direction**


- **Augment key(User define)**



- Machine builder augment key, combine with PLC ladder.

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



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

2.2 Text Key Description

A ~ Z : 26 English character keys

0 ~ 9 : Numerical keys

DEL : Delete key

INS : Insert /replace key

SHIFT : Switch keyboard to use subscript key on keyboard

SPACE : Space key

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

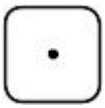
RESET : ”RESET “ abort the CNC status, only use this key if necessary

: To input current data to input box

: Enable Help function/message on screen

: For optional skip key input

: End of block



: Decimal fraction

: Change to program edit page.

: Change to monitor page.

: Change to customize teach page. (Disable function key with no customize action)

: Insert word when edit program.

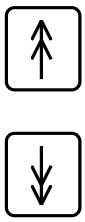
: Delete word when edit program.

: PAGE UP

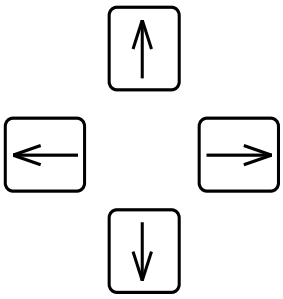
: PAGE DOWN

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

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



：Switch cursor to Page Up /Page Down



：Edit cursor control key

3 How to Operate 21 Series Controller

3.1 System Status

3.1.1 Not Ready

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

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

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

3.1.2 Ready

On “Ready” status, system can implement all operations
Conditions:

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

3.1.3 Busy

System is operating program
Conditions:

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

3.1.4 Pause

In operational process, system pause the axis movement
Conditions:

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

Note: On “Pause” status, Spindle still rotates

3.1.5 B-stop

Conditions:

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

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

3.2 Machine Preparation

3.2.1 Manual Function

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

3.2.1.1 JOG

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

3.2.1.2 Rapid JOG

- Function:
  - ✧ Control axis movement X, Y, Z according to selected direction with G00 speed
  - ✧ Can control more than one axis at the same time
- Condition:
  - ✧ System is on “Ready” status
  - ✧ JOG mode is selected
- Operation:
  - ✧ Press axis direction key (X+,X-,Y+,Y-,Z+…) and rapid key “～“at the same time ,machine will move with G00- rapid speed

- ✧ Hold the axis direction key to keep the axis moving uninterrupted
- ✧ Rapid JOG speed can be adjusted by G00%

- Note:
  - ✧ Rapid JOG is usually much faster than JOG, so when operating, please confirm the axis position to ensure human and machine safety.

1. INC JOG (incremental JOG)

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

2. MPG

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

3.2.2 Machining Process

3.2.2.1 AUTO

- Condition:
  - ✧ System is on “Ready” status
  - ✧ AUTO mode is selected
- Operation:
  - ✧ Press 「CYCLE START」 button
  - ✧ System will machine the current machining program
  - ✧ System status will be switched from “Ready” to “busy” and backs to “Ready” when machining is finished

3.2.2.2 Single Block

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

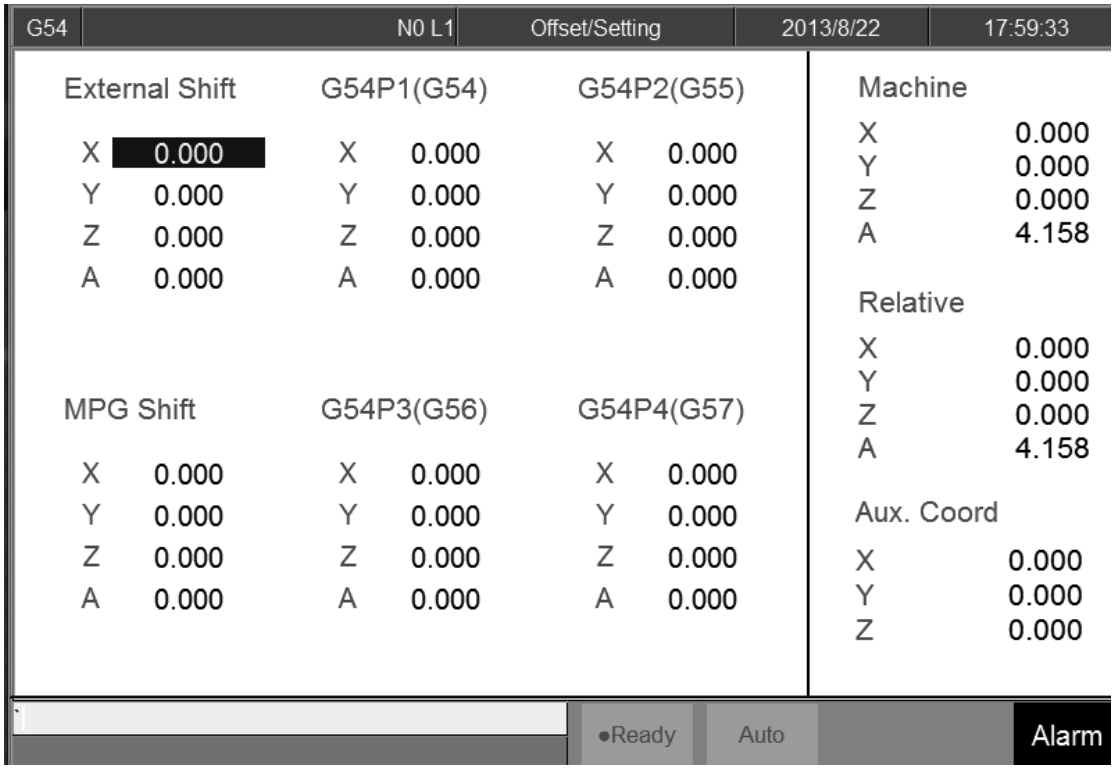
3.2.2.3 Home

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

- Operation:
  - ✧ Release emergency stop button, CNC status will change “NOT READY” to “ READY ”
  - ✧ Select HOME mode
  - ✧ Press axis direction key(X+,X-,Y+,Y-,Z+…) ,each axis would start HOMING
  - ✧ Home direction is defaulted in the CNC parameter
  - ✧ Home function can run 3 axes at the same time
  - ✧ After HOMING, all machine coordinates will be zero.
  - ✧ After HOMING, software stroke limit of each axis just is enable, so before HOMING, please do not run machine too fast

3.3 Workpiece Preparation

3.3.1 Workpiece Cord set



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

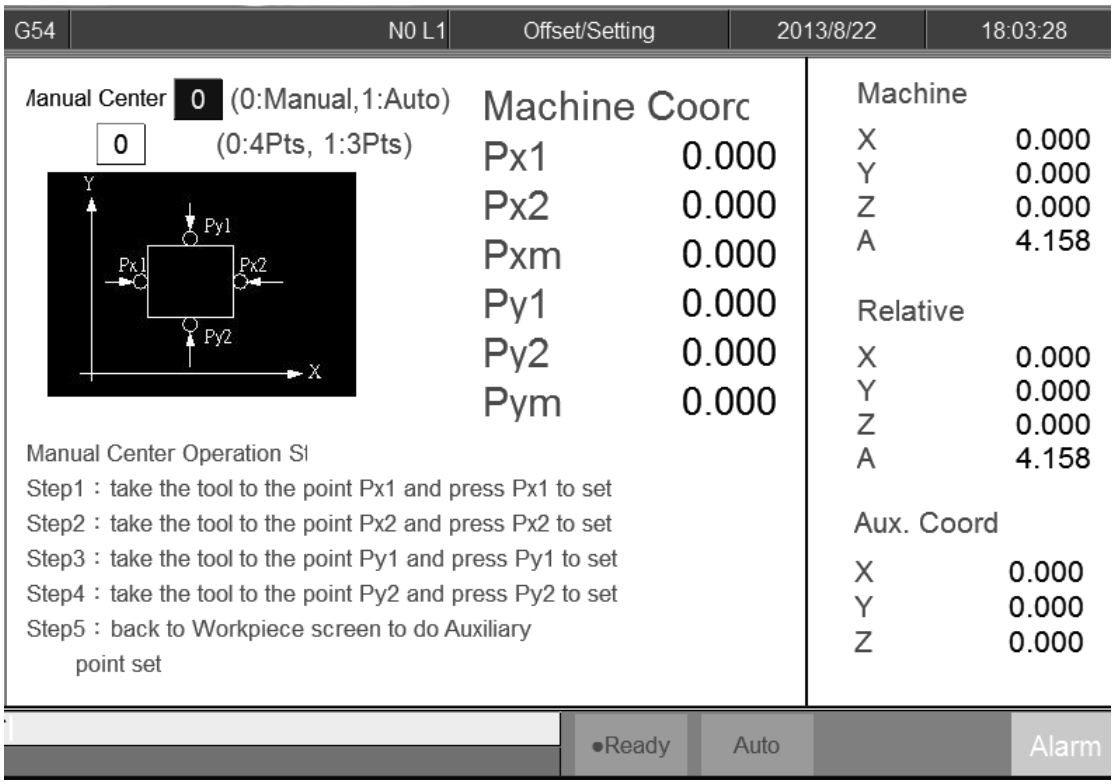
3.3.2 Middle Func.

- Function:

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

- Operation:

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



.

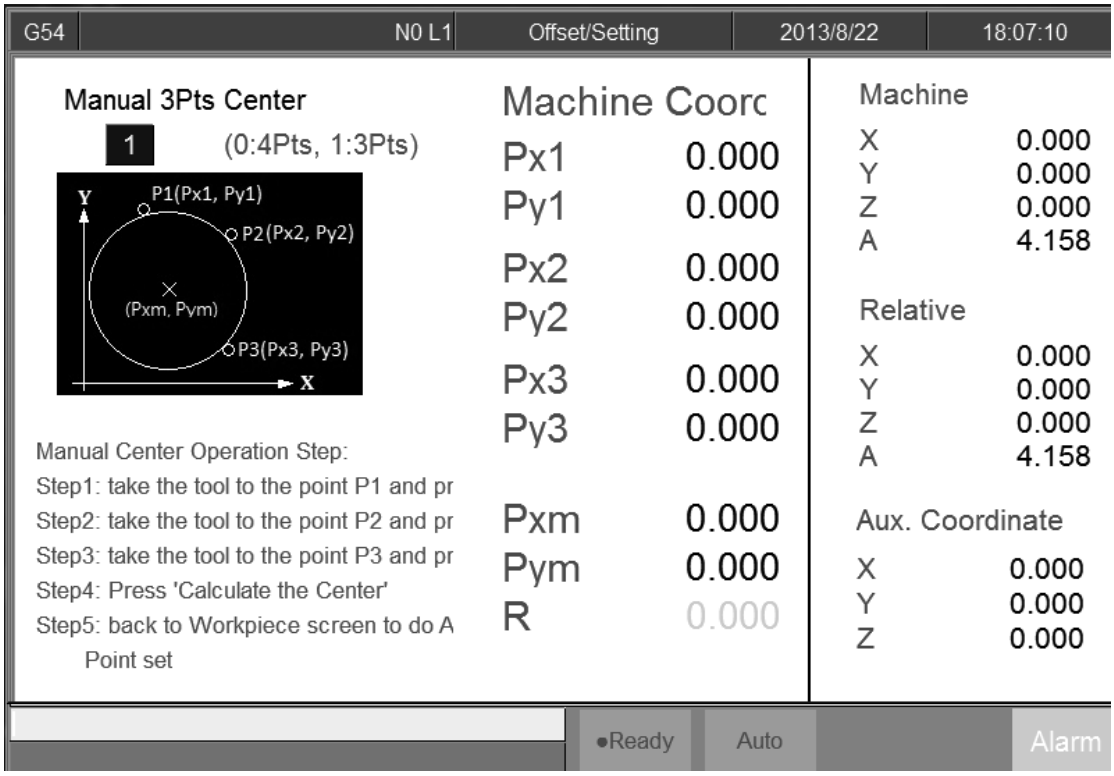
3.3.2.1 Manual 4Pts Center

- Function

  - ✧ Set the Middle Func. as 0
  - ✧ User controls the machine by MPG, and then moves the tip of 3D machine to the X&Y end point of the workpiece. This system will calculate the center point of the workpiece automatically.

- Operation method
  - ✧ Move the maching by MPG, touching Px1 point in this figure and then press PX1 Set. The system will record the current machnical coord. to Px1. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm and Aux. X position.
  - ✧ Move the maching by MPG, touching Px2 point in this figure and then press PX2 Set. The system will record the current machnical coord. to Px2. It will also compute the middle point of Px1 and Px2 and puts the result on Pxm and Aux. X position.
  - ✧ Move the maching by MPG, touching Py1 point in this figure and then press PY1 Set. The system will record the current machnical coord. to Py1. It will also compute the middle point of Py1 and Py2 and puts the result on Pym and Aux. Y position.
  - ✧ Move the maching by MPG, touching Py2 point in this figure and then press PY2 Set. The system will record the current machnical coord. to Py2. It will also compute the middle point of Py1 and Py2 and puts the result on Pym and Aux. Y position.
  - ✧ Now, the values of Pxm and Pym are the middle point of the workpiece.
  - ✧ In the WorklPiece Cord. Screen, move the cursor to the coord you want to set, press F3 insert the Aux. Coord value then this system will set the value accordind to the Aux Coord.

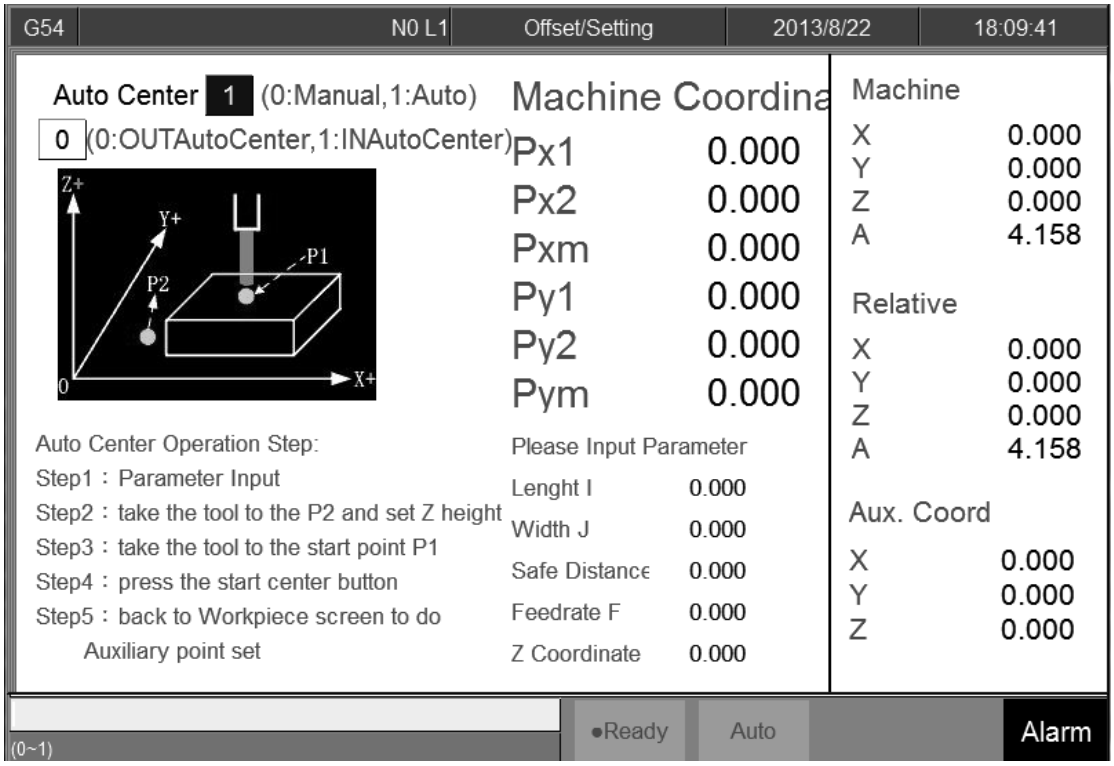
3.3.2.2 Manual 3Pts Center



- Operation:
  - ✧ Set the Middle func. as 0, the left upper side will show Manual Center
  - ✧ Set the second line, the middle method, as 1. You can see the screen as this figure.
  - ✧ Move the tool or detector to the round workpiece edge.
  - ✧ Press P1 set, system will record the current X and Y coordinates to Px1 and Py1.
  - ✧ Move to another edge point, press P2 set. System will record the current X and Y coordinates to Px2 and Py2.
  - ✧ Move to another edge point, press P3 set. System will record the current X and Y coordinates to Px3 and Py3.
  - ✧ After setting these, press Calculate the Center. System will calculate the center point Pxm, Pym and R.

- ✧ The Aux. Coordinate will become Pxm and Pym.
- ✧ If you set the same point or this three point is inline, it will have a warning window pop out.
- ✧ Now you can press  go back to previous page to reset the coordinate.
- ✧ In the WorklPiece Cord. Screen, move the cursor to the coord you want to set, press F3 insert the Aux. Coord value then this system will set the value accordind to the Aux Coord.

3.3.2.3 OUTAutoCenter

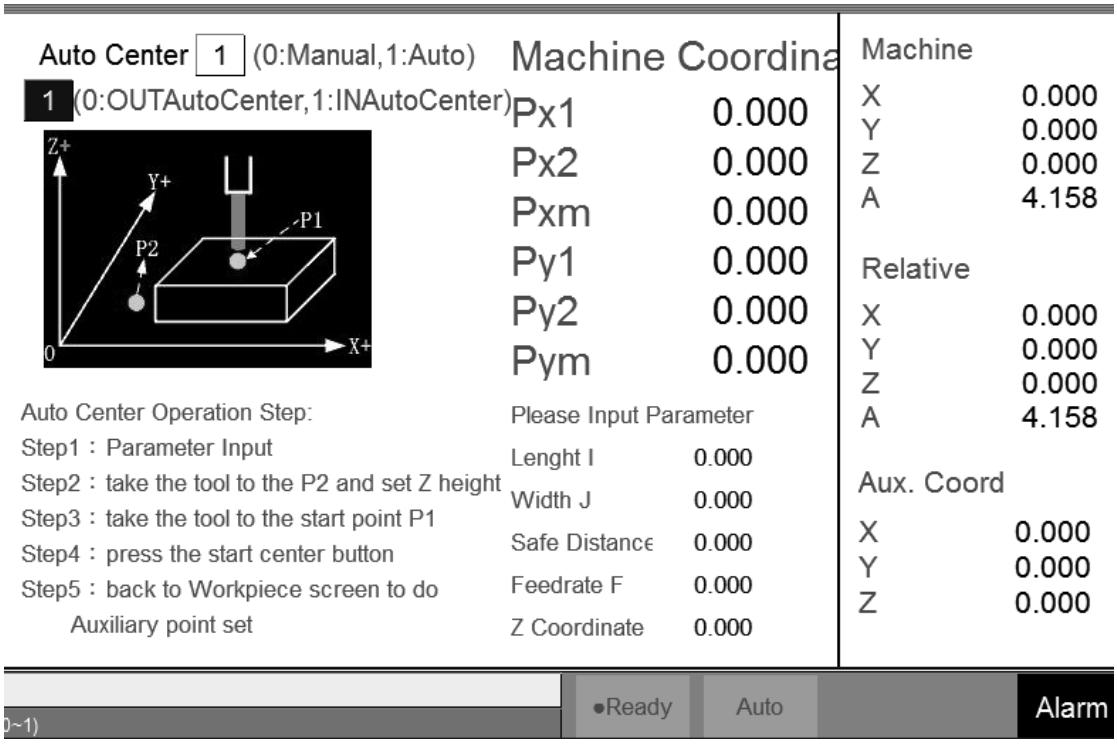


- Operation:
  - ✧ Set the Middle func. as 1, the left upper side will show Manual Center.
  - ✧ Set the second line, the middle method, as 0. You can see the screen as this figure.
  - ✧ Key in Length I and Width J for workpiece real length and width.
  - ✧ Set Z coordinate, this value is for the the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

on the screen. It will also calculate the X Y center point of the workpiece

- ✧ The center point will be saved at Pxm and Pym.
- ✧ After setting these, press Calculate the Center. System will calculate the center point Pxm, Pym and R.
- ✧ The Aux. Coordinate will become Pxm and Pym.
- ✧ Now you can press  go back to previous page to set the coordinate.
- ✧ In the WorkPiece Cord. Screen, move the cursor to the coord you want to set, press F3 insert the Aux. Coord value then this system will set the value according to the Aux Coord.

3.3.2.4 INAutoCenter



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

3.3.3 Tool Prepare

3.3.3.1 Tool Set

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

3.3.3.2 Manual Measurement of Tool Length

- Purpose
  - ✧ Describe how to use manual measurement to set tool length and work piece coordinate in order to set the machining tool.
- Function
  - ✧ We usually set the tool length and offset in the External Shift if we only use one tool in this machine.
  - ✧ If we have more than one tool, we need to set every tool length we can use relative length or absolute length.
- Operation condition

✧ Manual mode.

- Manual Operation method

-Relative method.

✧ Use manual function moving the reference tool to the Z-coordinate code 0 position of the workpiece (Ex. surface), set the value at G54.

✧ Clean all the relative coordinate value.

✧ Install the measured tool, move the tool to the same position as before (reference surface).

✧ The tool length is the different between measured tool and reference tool.

✧ Key in this number to the corresponding tool No.

✧ If tool break during machining and replace a new tool, we need to change to a new tool. If the Z-cord 0 position already disappear, we can use the standard tool to touch anywhere of the machine and then use new tool to touch the same position. The relative cord. is tool length.

-Absolute method

✧ Use manual function moving the measuring tool to a reference point (Ex. Work piece table).

✧ Measure every tool repeatedly.

✧ Move the machining tool to the reference position, clean all the relative coordinate.

✧ Move the tool to the Z-coord code zero position, set it to G54.

✧ If tool break during machining so we need change a new tool, we must touch the reference position first, and then the Z coordinate value is tool length.

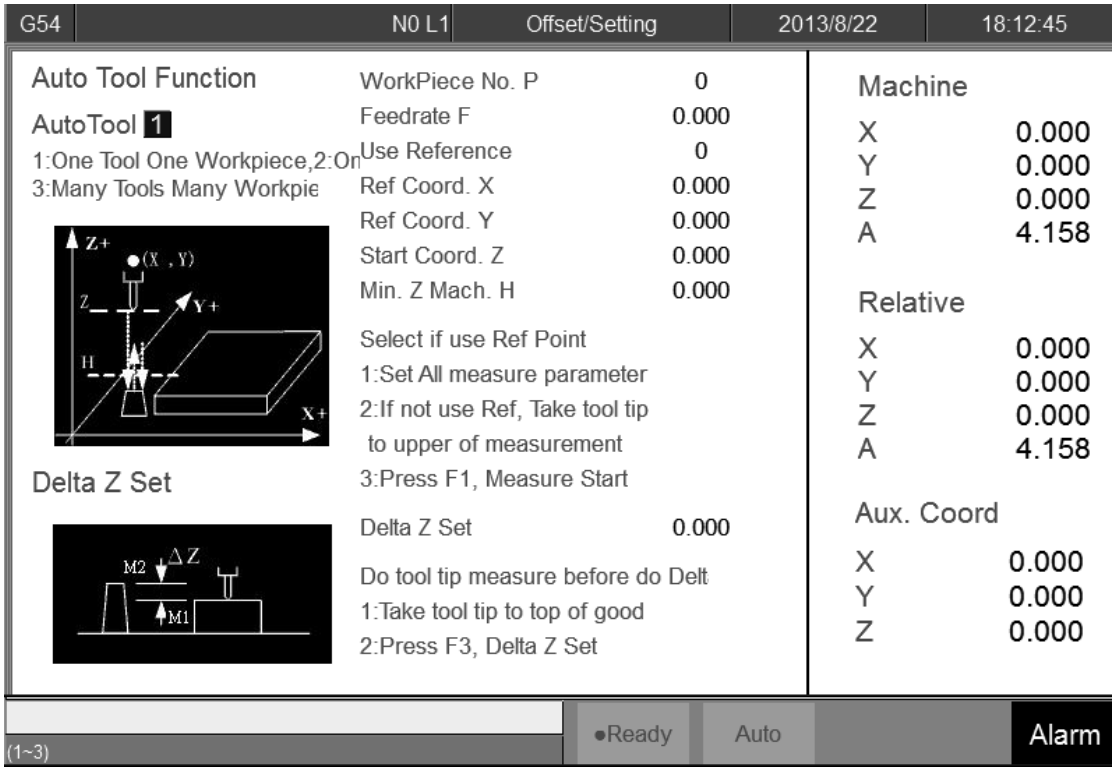
3.3.3.3 Tool Tip Measure

- Function

✧ Measuring different tool No. tip position. Because the trigger position of tool alignment equipment is fixed, user can use different tool tip position key in the workpiece coordinate system for the reference of tool length offset. You must check this machine has tool alignment equipment.

- Operation Method

✧ Offset/Setting→Tool Tip Measure



3.3.3.4 Auto Tool Function

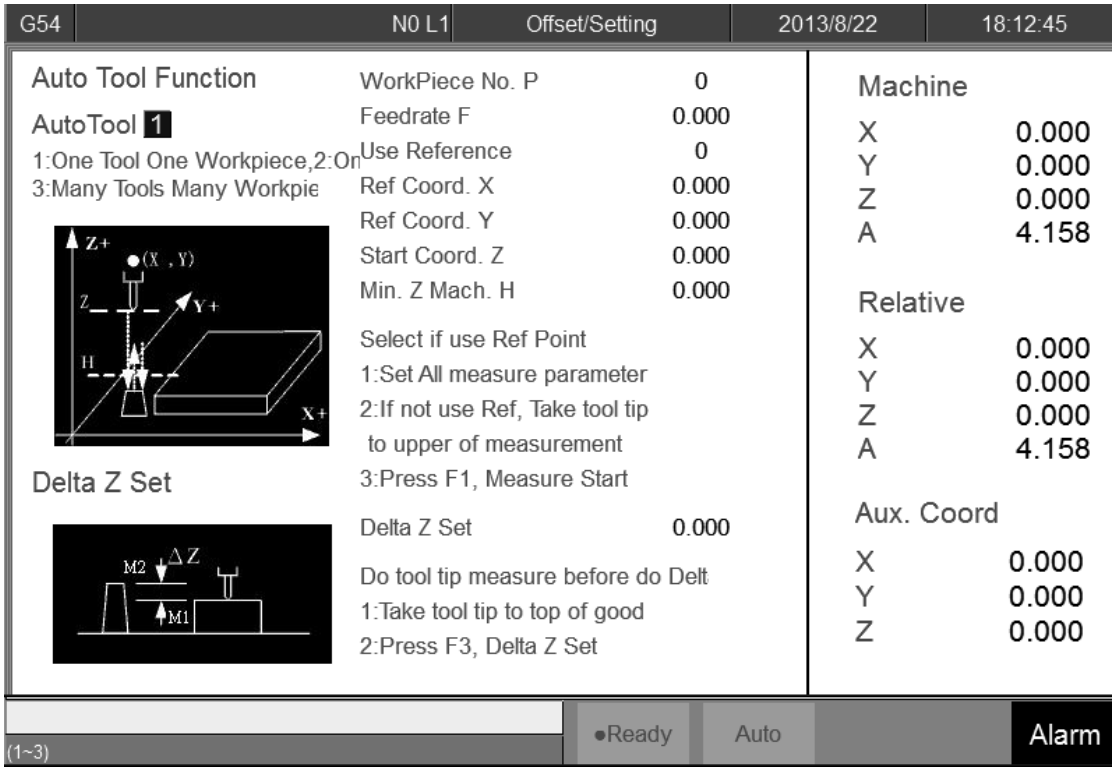
- Function

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

- Operation description

  Offset/Setting→Tool Tip Measure

- One Tool One Workpiece



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

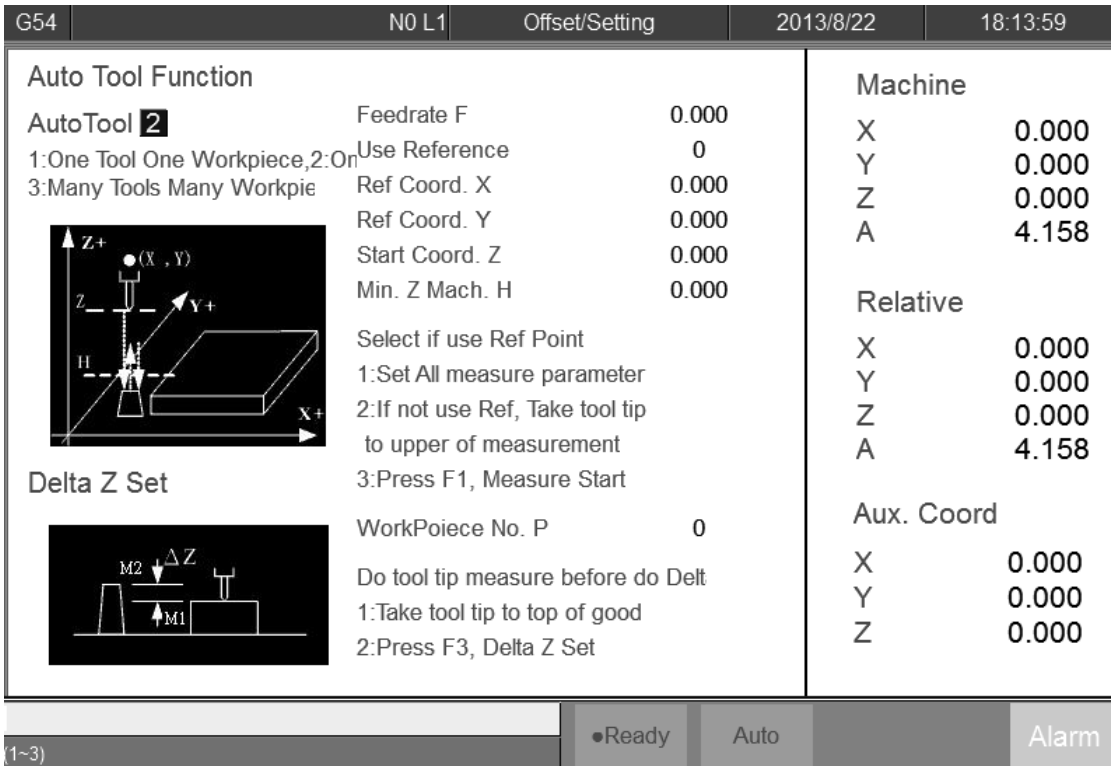
  ✧ Set the WorkPiece No. P as the workpiece coordinate.

- Table X- Coordinate table

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

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

- One Tool Many Workpieces



✧ Set the Auto Tool number as 2.(left-upper corner)

✧ Set the feed rate F for the first time alignment and pull back every time.

✧ Please skip to next step setting XY Ref. Coord. Teach if tool alignment equipment already has fixed mechanical coordinate. If current position is the same with tool alignment equipment, please set Use Reference as 0.

✧ While setting the XY Ref. Coord. Teach, you will need code "520", enter it and choose yes. These values will be entered in to Ref coord.X and Ref coord.Y.

✧ Move the cursor to Start Coord. Z, setting the start point for the auto alignment. Press F10 and then press Z Mach. Coord. Teach, you can insert the current Z coord. value into Start Coord. Z.

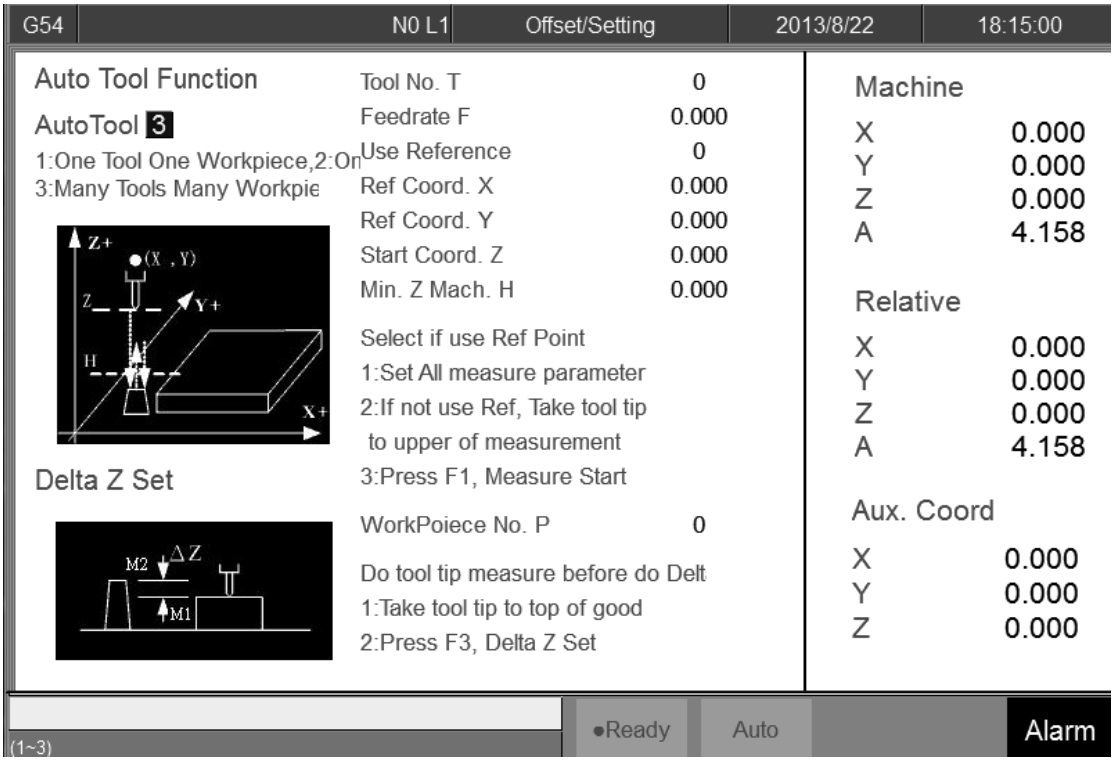
- ✧ Move the cursor to the Min. Z Mach. H, setting the minimum height for the alignment. System will pop out warning message and stop if the tool is lower than it.
- ✧ Please switch to auto mode after setting these and then press Start.
- ✧ After finishing alignment, this tool height value will be saved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

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

Z Delta set table

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

- Many Tool Many Workpieces



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

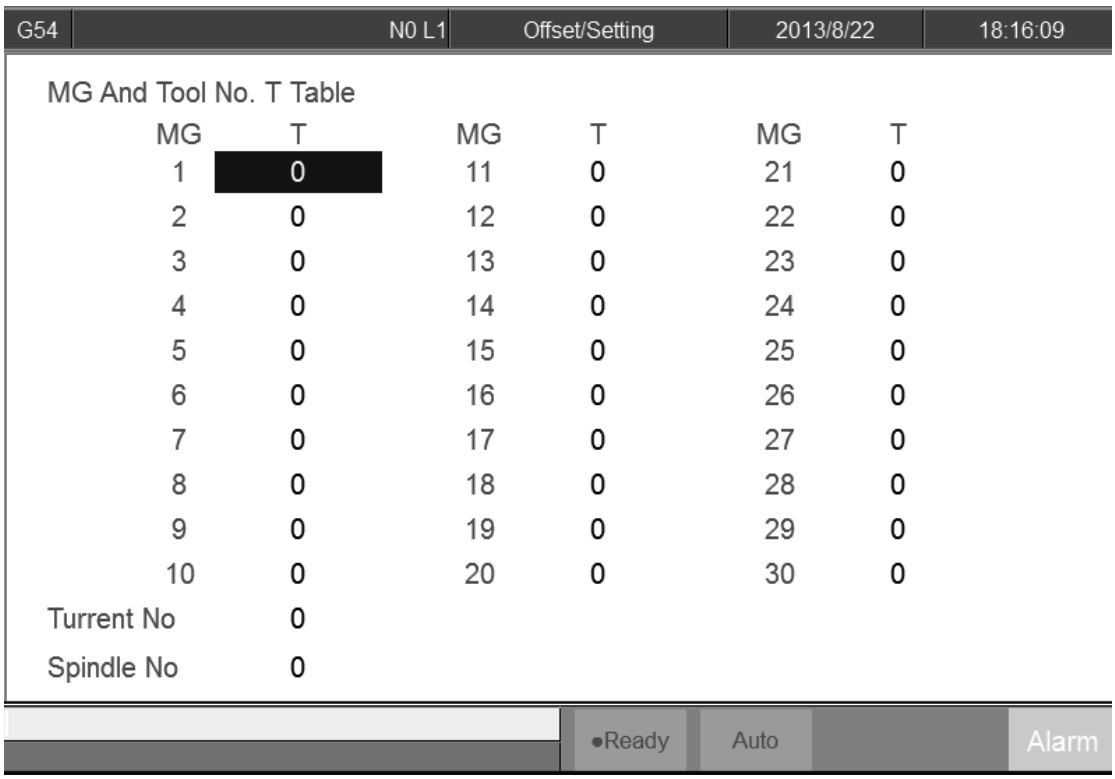
- ✧ Move the cursor to the Min. Z Mach. H, setting the minimum height for the alignment. System will pop out warning message and stop if the tool is lower than it.
- ✧ Please switch to auto mode after setting these and then press Start.
- ✧ After finishing alignment, this tool height value will be saved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

3.3.3.5 Tool No. Setting

This Chapter will describe how to set the tool No.

- Purpose:
  - ✧ We need to confirm the relation between Tool No. and real tool in order to change tool correctly.
- Condition:
  - ✧ Both Auto mode or Manual mode are OK..
- Operation Method
  - ✧ Tool Set→Tool No.
  - ✧ You will see the table about Tool No. and Tool MG.

✧ MG5—T = 7 means we install tool No.7 at Tool case No.5



✧ This table will renew during change a new tool.

- Time to Modify

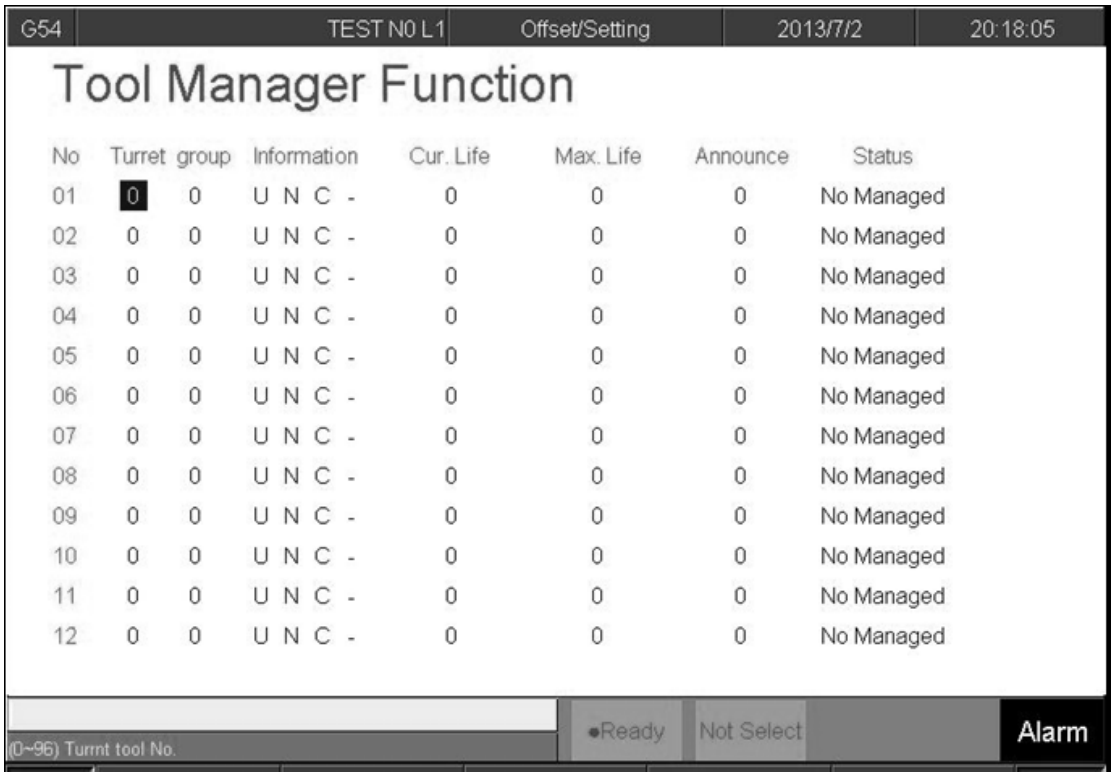
✧ First initialized when it is made from factory.

✧ Need to confirm the MG No. and T No. Manually when it is disordered.

3.3.3.6 Tool Manager Function

- Purpose:

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



- Condition
  - ✧ Both auto and manual can be used.
- Operation
  - ✧ Pr.3228 is the on/off control of 「Tool management」
- Description
    - ◆ Turret
  - ✧ Current tool case no. that tool located.
    - ◆ Group
  - ✧ Same kind of tool within in one group, if the first tool of that group is on lock state or 「Tool life」is end, whenever user use T

code to change the tool, system will skip the first tool and use the second one, when the second one is lock or「Tool life」is end, will use the third one, and so on.

◆ Tool information (Status)

✧ L—Lock / U—Unlock

✧ If the status of tool is lock, that tool cann't be use and when T code is use to change the tool, system will skip that tool.

✧ B—Large diameter Tool/ N—Normal Diameter Tool

✧ Adjacent side of large diameter tool set is empty(for display)

✧ T—working time T / C—Number of working times

✧ Decide the current life time, the maximum life time, life time prediction, unit of timing and number of time.

✧ R—effective value / - —non effective value.

✧ Current tool are using tool management or not.

◆ Current Life time

✧ Current Tool Using Condition

◆ Maximum Life Time

✧ Maximum lifetime of tool.

◆ Lifetime prediction

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

◆ Current Status of Tool

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

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

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

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

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

(5)ware of tool

3.4 Program Preparation and Execute Machining

3.4.1 Specifying Machining Program

- Condition
  - ✧ Except single block mode
- Operation
  - ✧ Specify current edit program as machining program
  - ✧ Switch to edit page
  - ✧ Select F1-Excute, and the program will be designated as the machining program
  - ✧ Specify machining program in file manager.
  - ✧ Switch to the "File Management" page
  - ✧ Move the cursor to the expected program and press Enter
  - ✧ Select F1-Excute, and the program will be designated as the machining program
- Confirmation:
  - ✧ There are two ways to confirm whether machining program is specified successfully.
  - ✧ The screen displays the correct machining program name
  - ✧ The content of machining program is displayed when pressing F4-Monitor

3.4.2 Simu. Setting

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

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

  -Simulation screen
  - ✧ The solid line represents the cutting path
  - ✧ The dashed line represents the moving path
  - ✧ In the scanning process, if there is any syntax or content error, they will be displayed on the screen with corresponding error line number.
  - ✧ F1-step: To simulate tool path corresponding to single block in NC files. It is used for coordinate checking purpose.
  - ✧ F2-Continue: System scans the whole program first before executing simulation.
  - ✧ F3-zoom: To zoom in/out the workpiece graph. Users can use the arrow key "←", "↑", "→", "↓" to move the frame to the

determined area, use “PageUp” “PageDn” to zoom in/out this area. After selecting zoom scales, press “enter” to finish.

- ✧ F5- simu. Setting: To set simulation parameter

3.4.3 Machining Test

3.4.3.1 MPG Simulation

- Condition
  - ✧ Only for single block and auto mode
- Operation
  - ✧ Select Auto mode
  - ✧ Press MPG simulation button on operation panel
  - ✧ Turn MPG to execute machining
  - ✧ If MPG is turned in CW direction, Program will be run from current NC line down to below NC line
  - ✧ If MPG is turned in CCW direction, Program will be run from current NC line up to above NC line
- Confirm
  - ✧ We can confirm MPG simulation successful or not by these two method.
  - ✧ Not machining, execute MPG function, and then in the monitor page try to rotate the MPG. If you can see G01 is zero before you rotate and has value after you rotate MPG.
  - ✧ While machining, execute MPG function, and then machine will stop to 0 immediately, until you rotate MPG or cancel MPG function.

3.4.3.2 Single Block

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

3.4.3.3 Start MPG Coordinate

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

- Condition
  - ✧ MDI or Auto mode
- Operation Method
  - ✧ F4 Monitor→F6 Start MPG Coordinate
  - ✧ You will see the MPG coordinate system window.
  - ✧ Switch to the MPG mode.
  - ✧ Rotate the MPG, you can see the value at this window.
- Confirmation
  - ✧ Switch to workpiece coordinate page, confirm the value in MPG Shift is correct.
- Relative parameter
  - ✧ .Pr 3201 Set the Lathe Rule, you must set 0 if you want to use Start MPG Coordinate function

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

  -Keep the MPG coordinate while starting the machine.

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

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

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

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

3.4.3.4 Break Point

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

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

✧ Execute.

3.4.4 Machining Monitor

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

3.4.4.1 Part Count Manager

- Description

1. Total accum part

✧ The total accumulative part machined by CNC

2. Required part count

✧ Once machining program specifies the needed workpiece number, and CNC continues machining when it meets M99, if the demand workpiece number is reached, machining process will be paused and notification message will be displayed.

3. Part count

✧ Once CNC continues machining when it meets M99, this number will be accumulated until reset

✧ Part count reset (clear to 0) condition

✧ Required part count is reached

✧ Change machining files

✧ Modify the required part count, and the required part count is smaller than part count.

3.4.4.2 Work Record Function

- Description

✧ Once CNC continues machining when it meets M99, work record function will automatically record status

- Work record condition

✧ Required part count is reached

✧ Change machining files

- ✧ Modify the required part count, and the required part count is smaller than part count.

3.4.5 Alarm Processing

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

3.4.5.1 Emergency Stop

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

3.4.5.2 Alarm Display

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

3.4.5.3 Pending Alarm

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

3.4.5.4 History Alarm

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

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

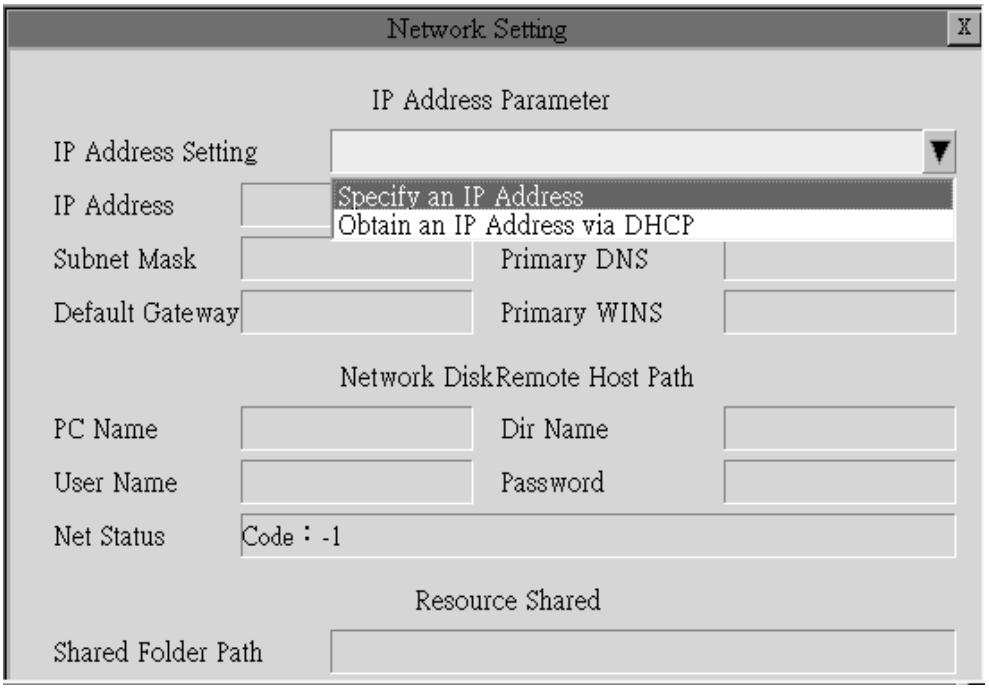
3.4.5.5 Save Alarm

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

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

3.4.6 Network Setting

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



4. **Subnet Mask**: Enter the IP address for subnet mask (the same with PC subnet mask).
5. **PC Name**: Enter the full computer name of your PC.
6. **Dir Name**: Enter the sharing folder name (the same name with PC sharing folder)
7. **User Name**: Enter GUEST
8. Press「**F1 OK**」, and then reboot controller to finish installation.

3.4.7 PC Setting

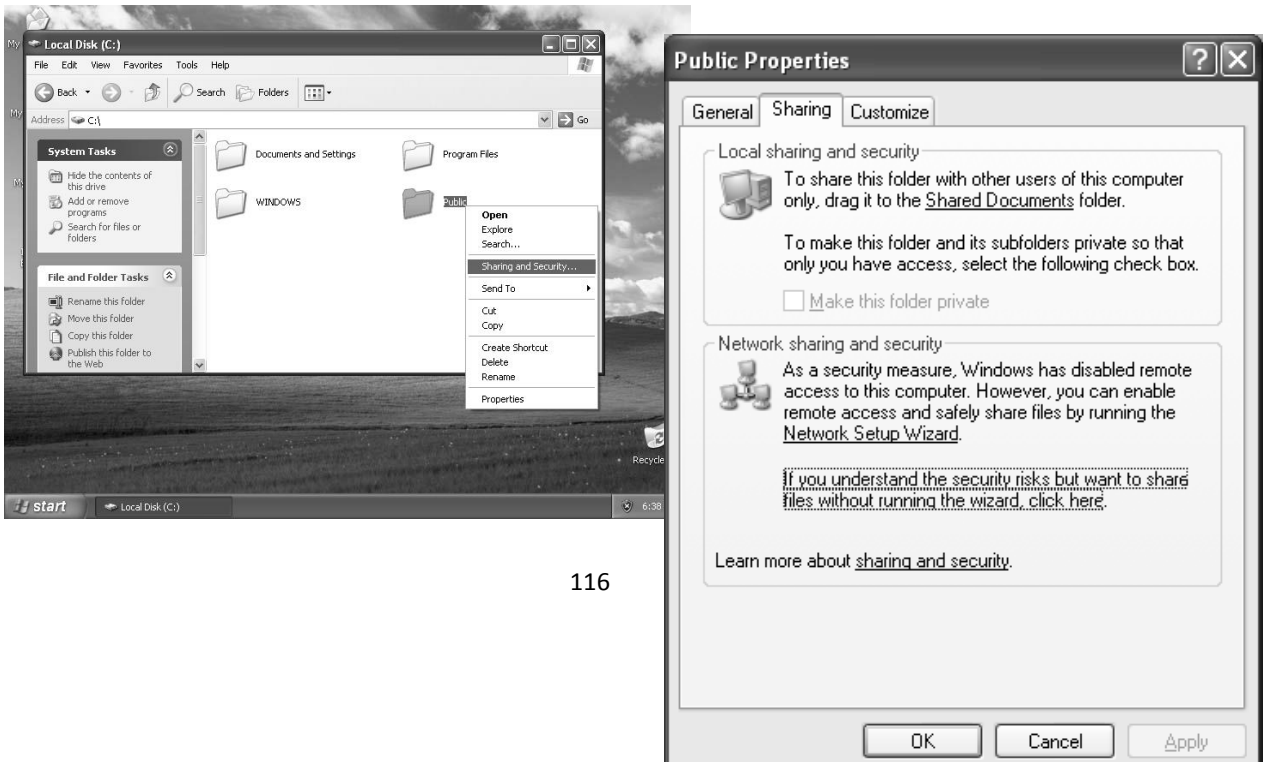
3.4.7.1 XP OS

1. Guest account setting

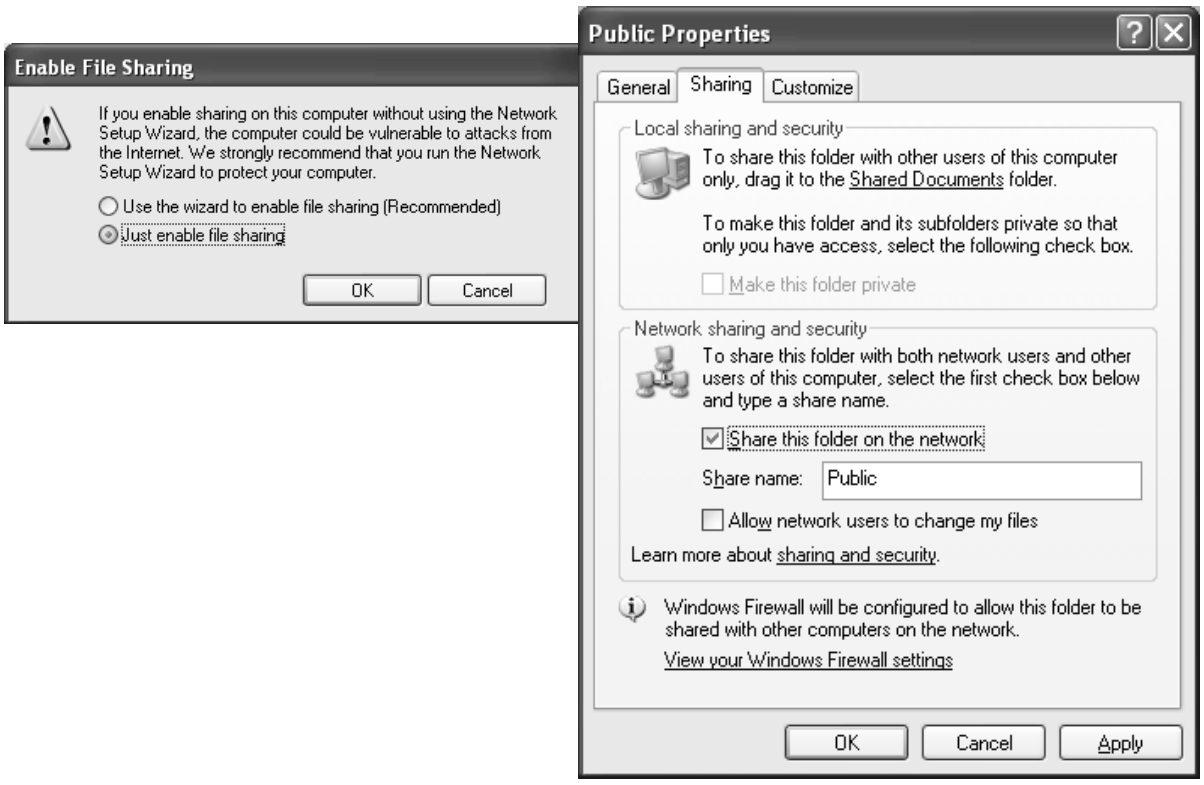
Log in as an Administrator and select “start”→ “control panel”→”user account”→ Guest

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”

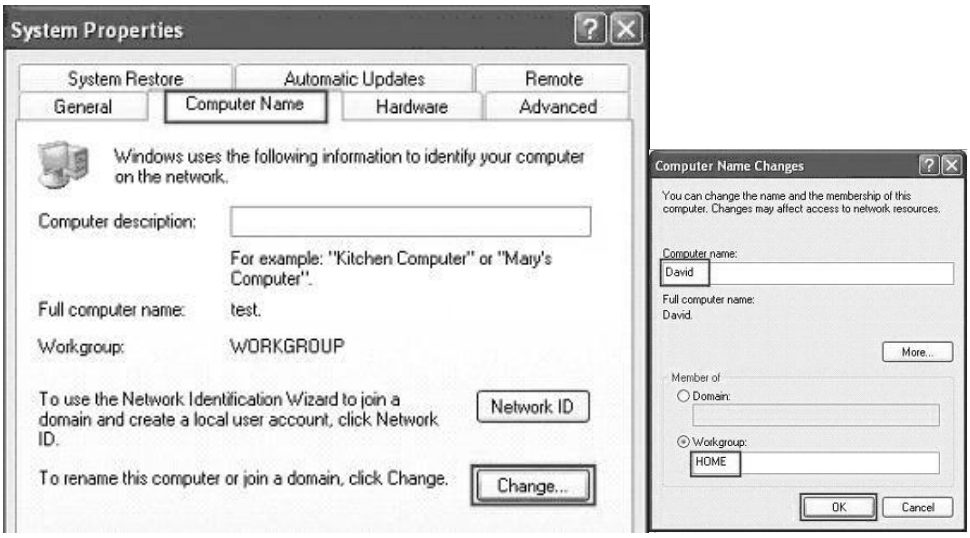


3. Click “OK” to confirm sharing setting; Select “Share this folder on the network”, and “Allow network users to change my files”.



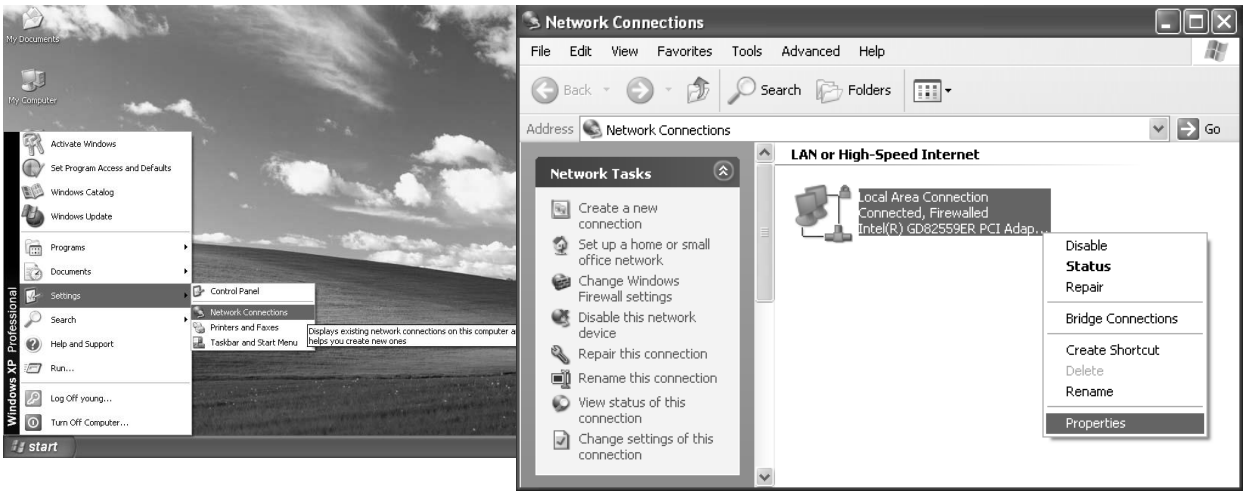
4. Setting PC name and workgroup

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



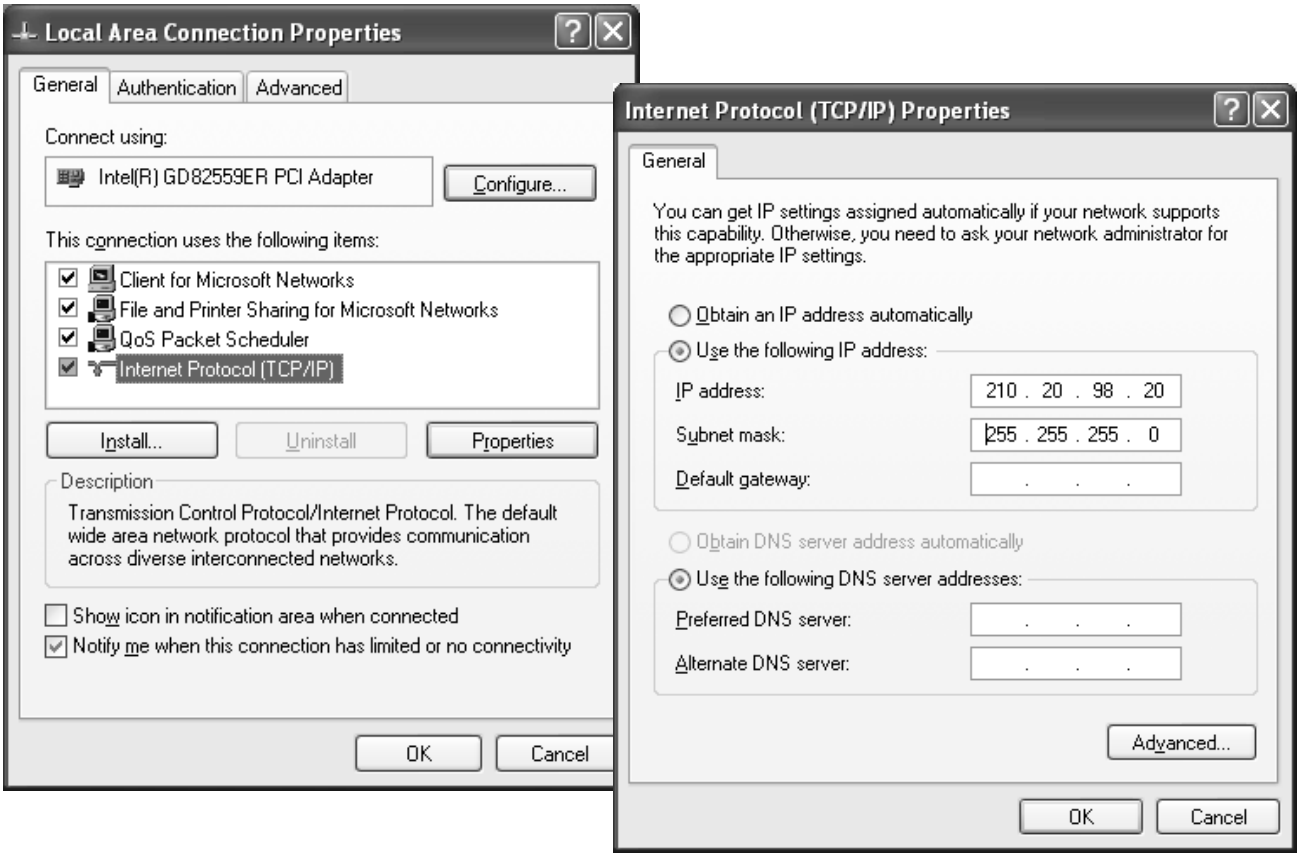
5. TCP/IP setting

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



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

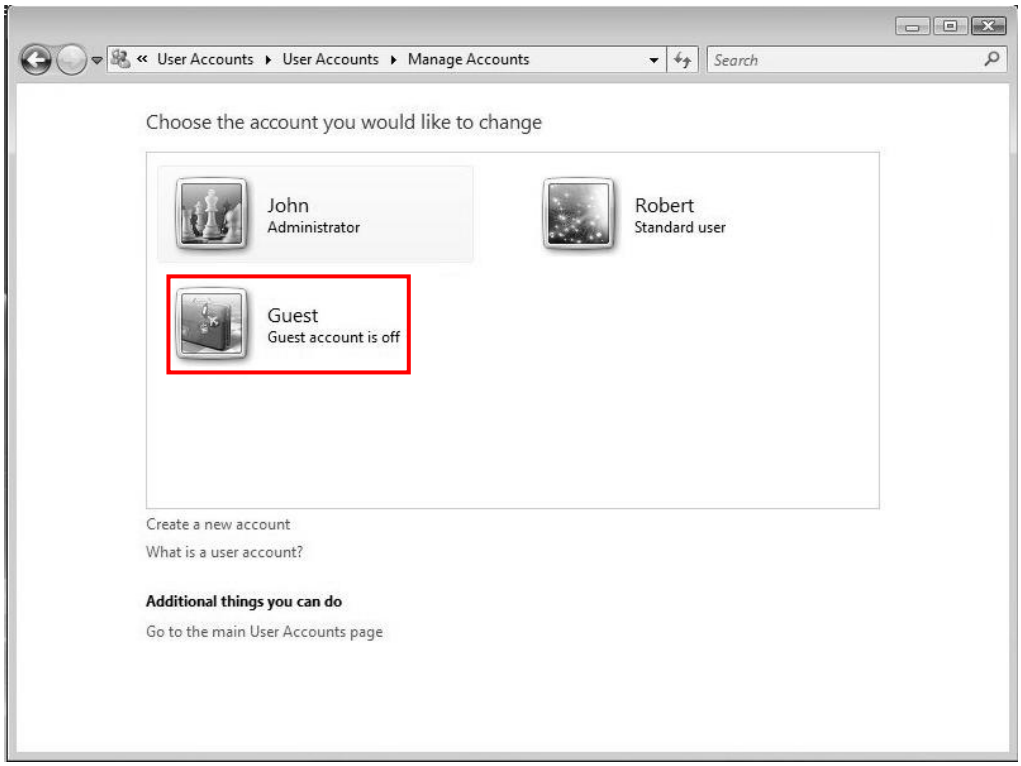
- ✧ Jumper cable (without HUB), select "use the following IP address" and enter IP address (the forth number is different from controller setting) and Subnet mask (same with controller setting)
- ✧ Network cable (with HUB), select "Obtain an IP address automatically"



3.4.7.2 VISTA OS

1. Guest account setting

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

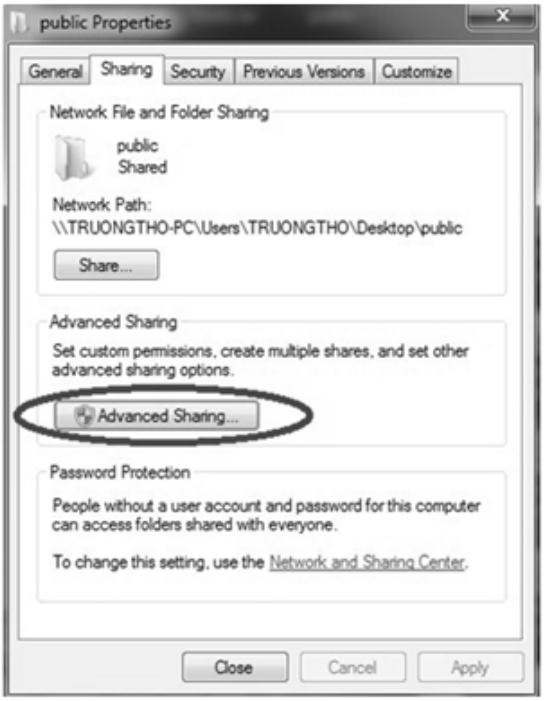


2. Sharing Resource Setting

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

A. Click on "advanced sharing"

B. Click on "share this folder"



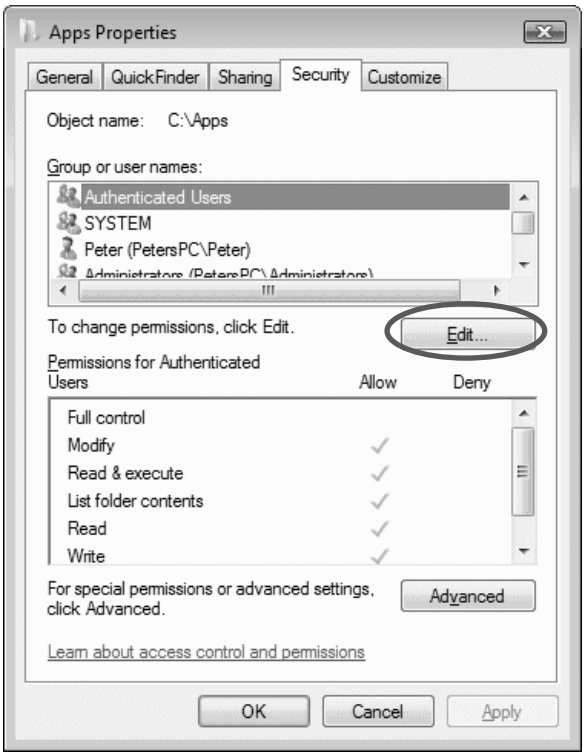
C. Click on "permission"

D. Click on "add"

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

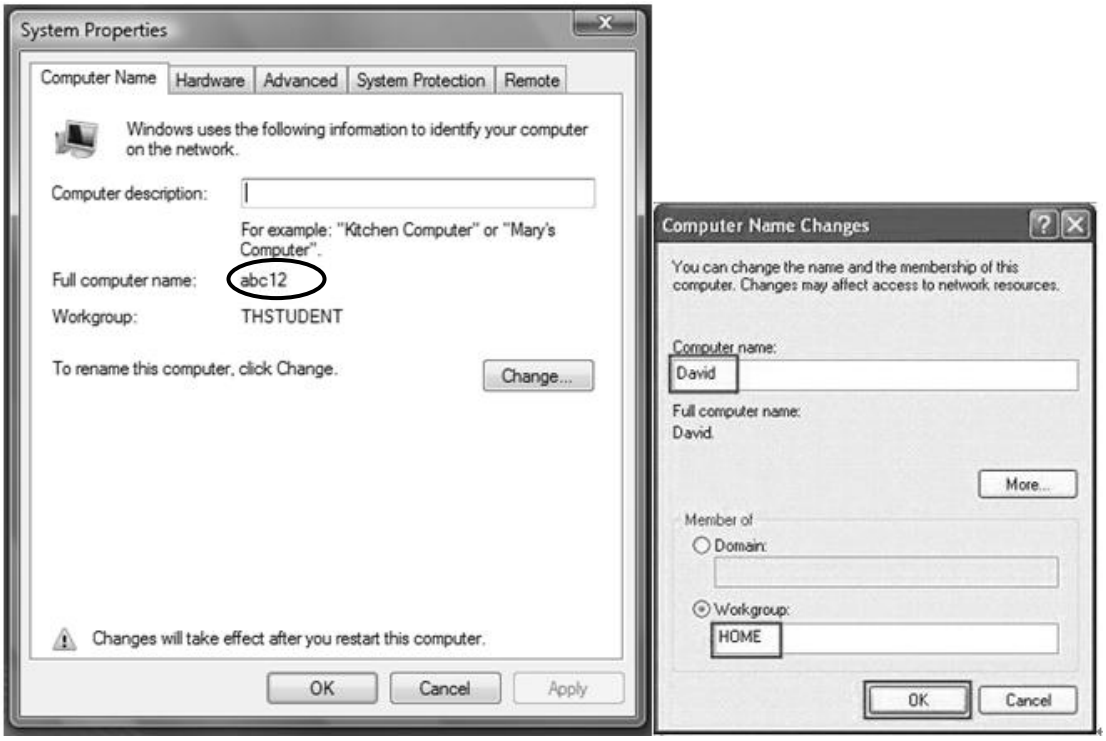
3. Security setting

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



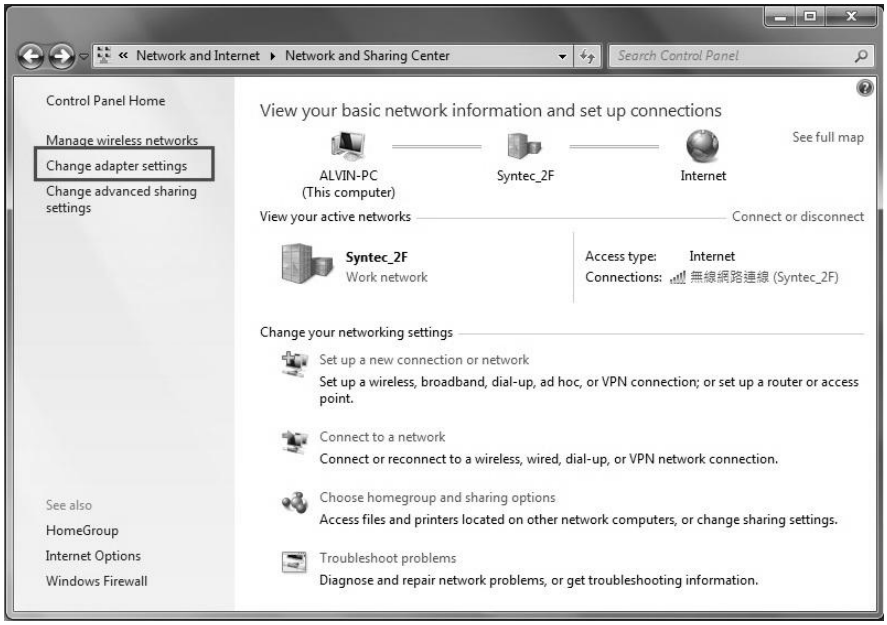
4. Setting PC name and workgroup

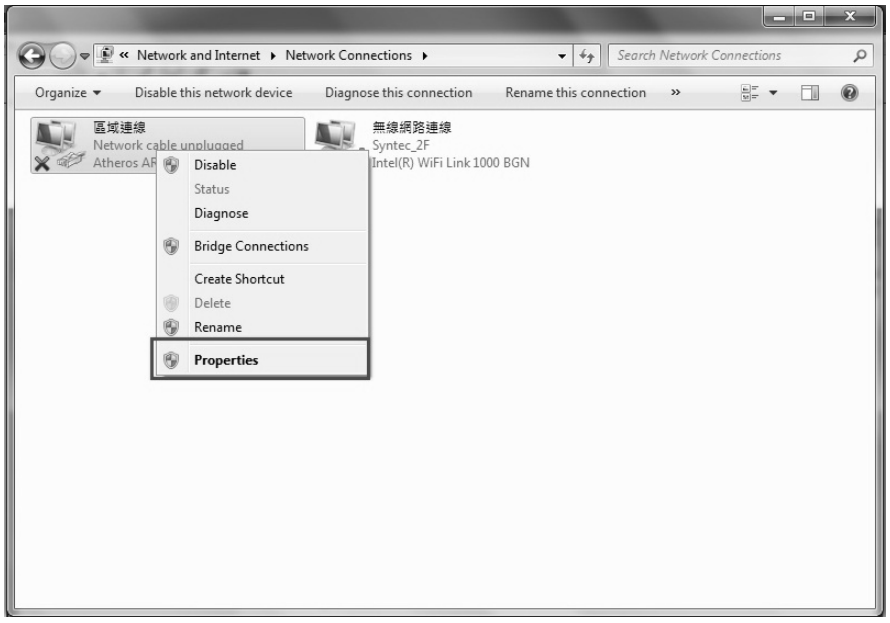
"Start"→ "control panel"→"System"→ "change" to set "Computer Name" and "Workgroup", and remember these setting contents to use later on when setting controller.



5. TCP/IP Setting

a. “Start” → “control panel” → “Network and Internet”→“Network and Sharing Center” → “Properties”

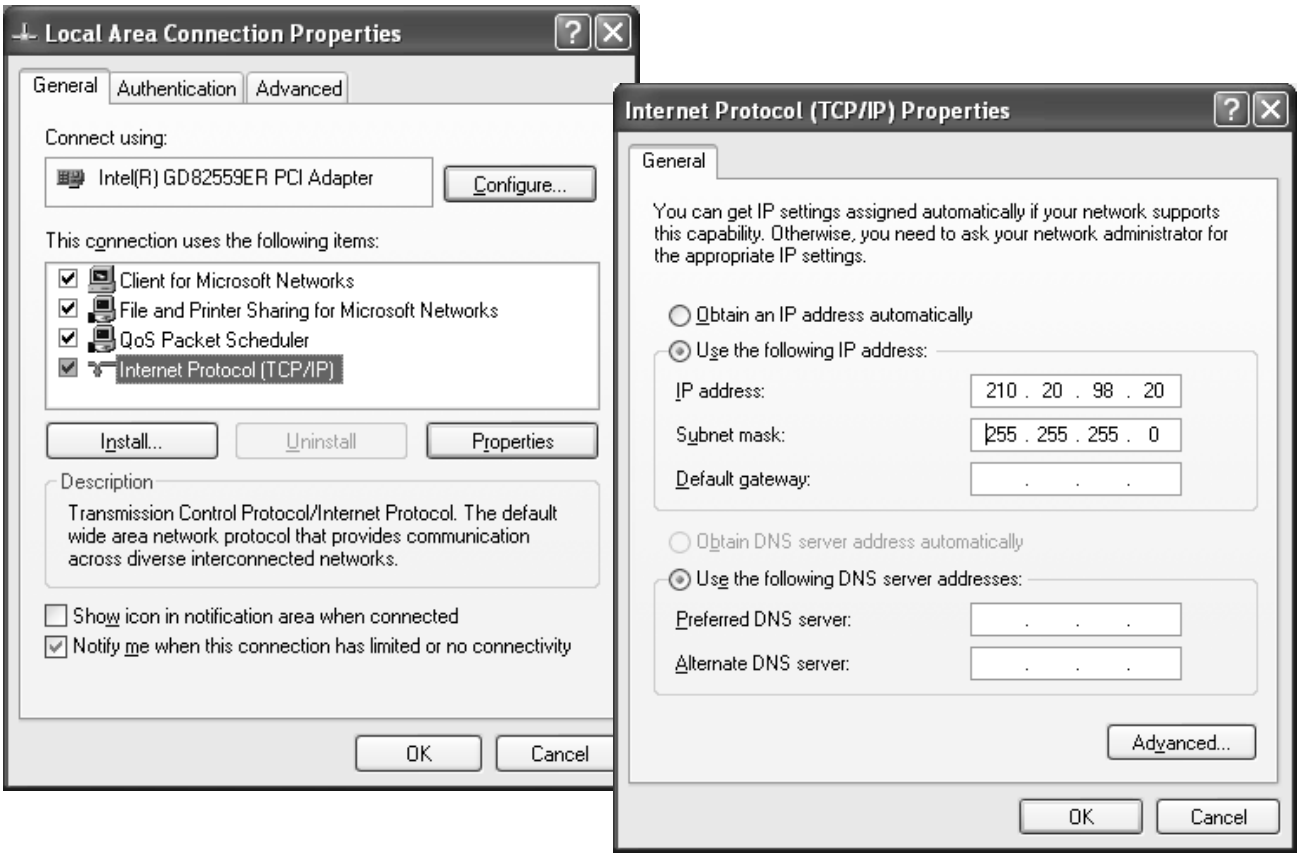




b. Select “internet protocol(TCP/IP)” as shown below:

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

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



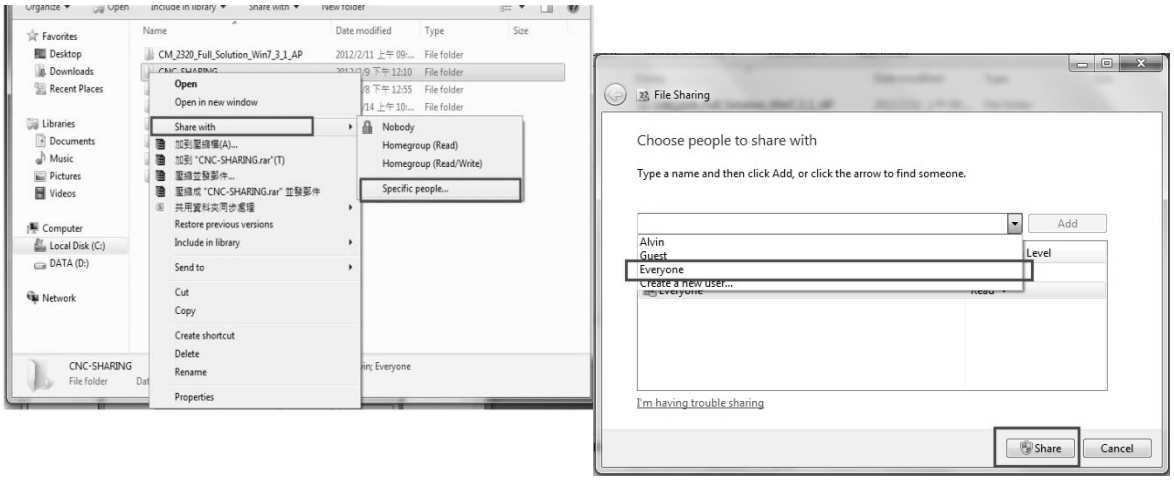
3.4.7.3 Win 7 OS

1. Sharing resource setting

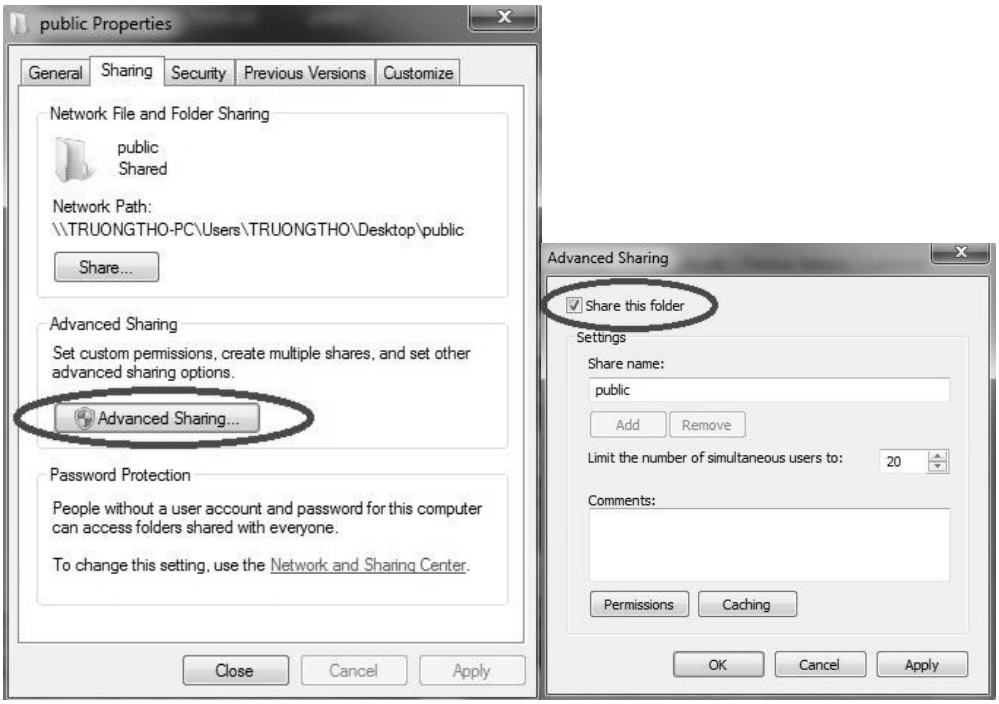
   ✧ Right-click on folder wants to share, select “share with” and “specific people”

   ✧ Share this folder to everyone, and then click “Share” as follows.

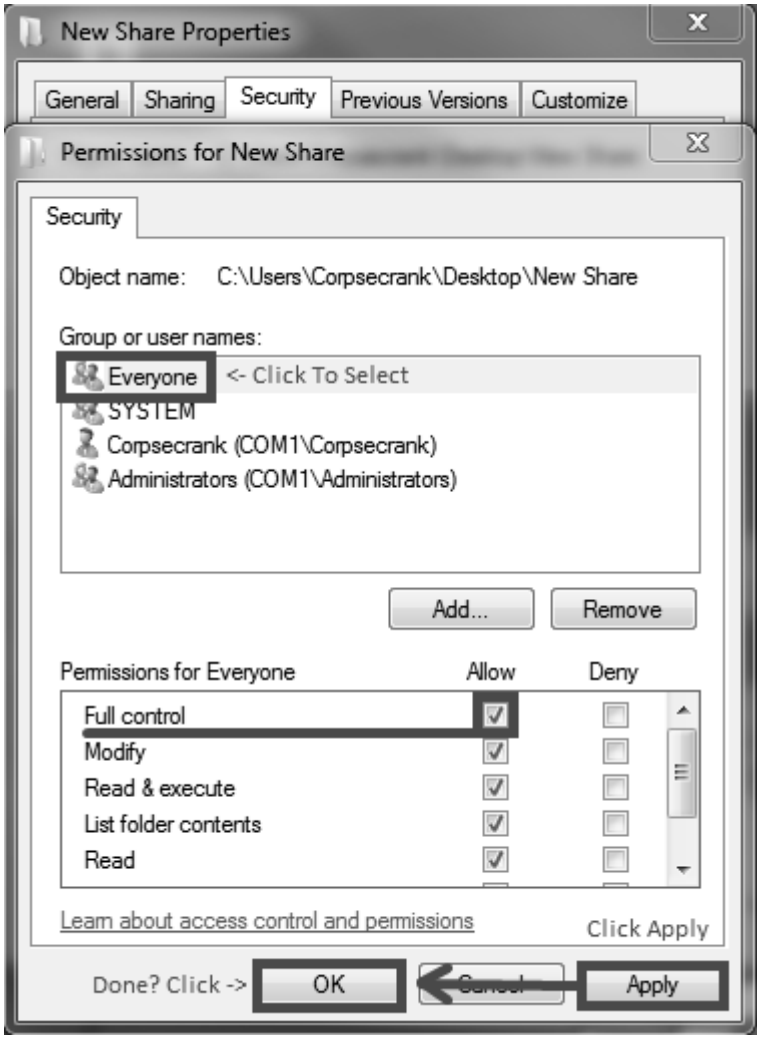
   ✧ Set permission as write/read



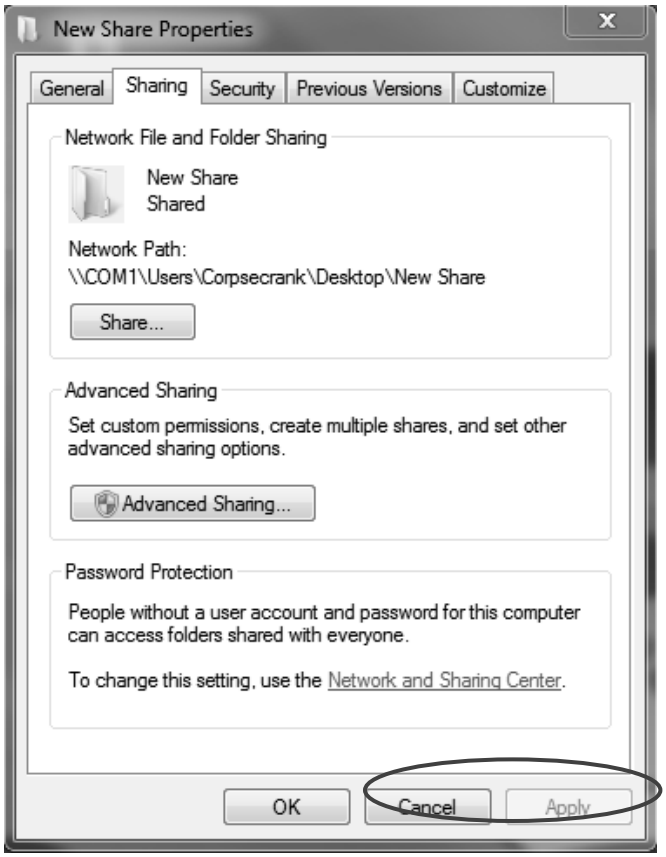
   ✧ Left-click on “advanced sharing” and select “share this folder” to everyone.

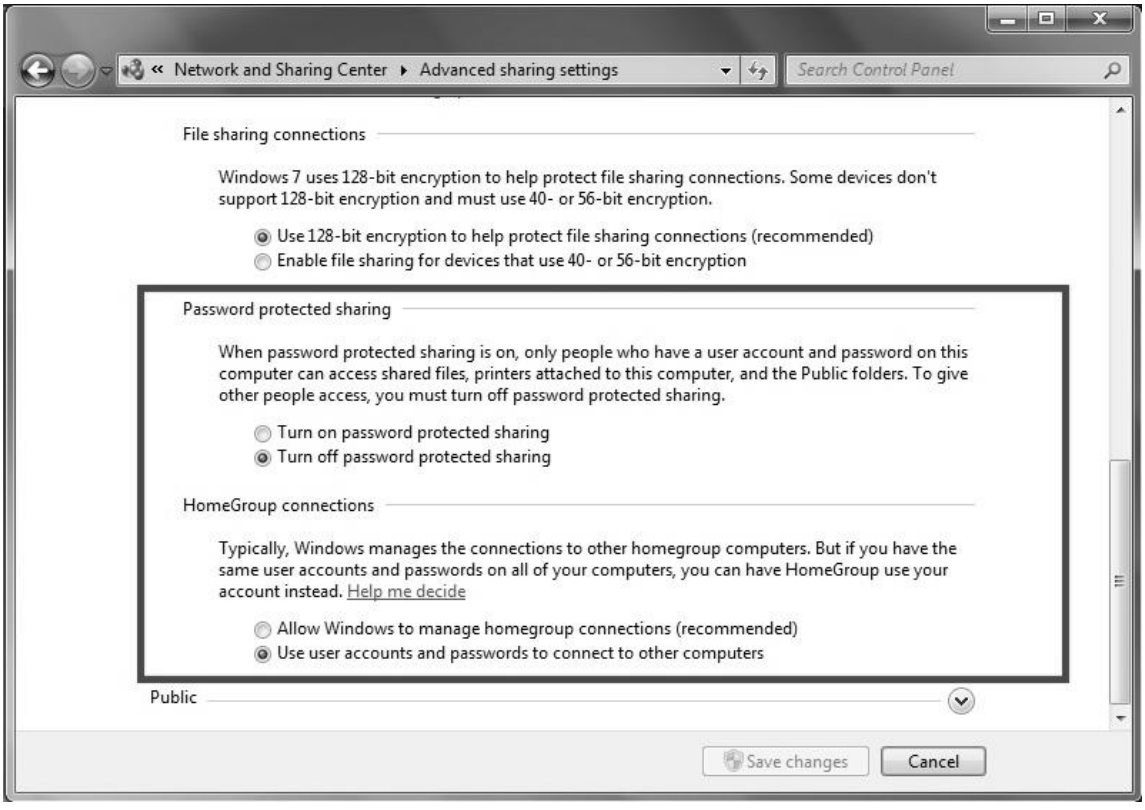


✧ Select “permission” and select “full control” “only read” and “change”



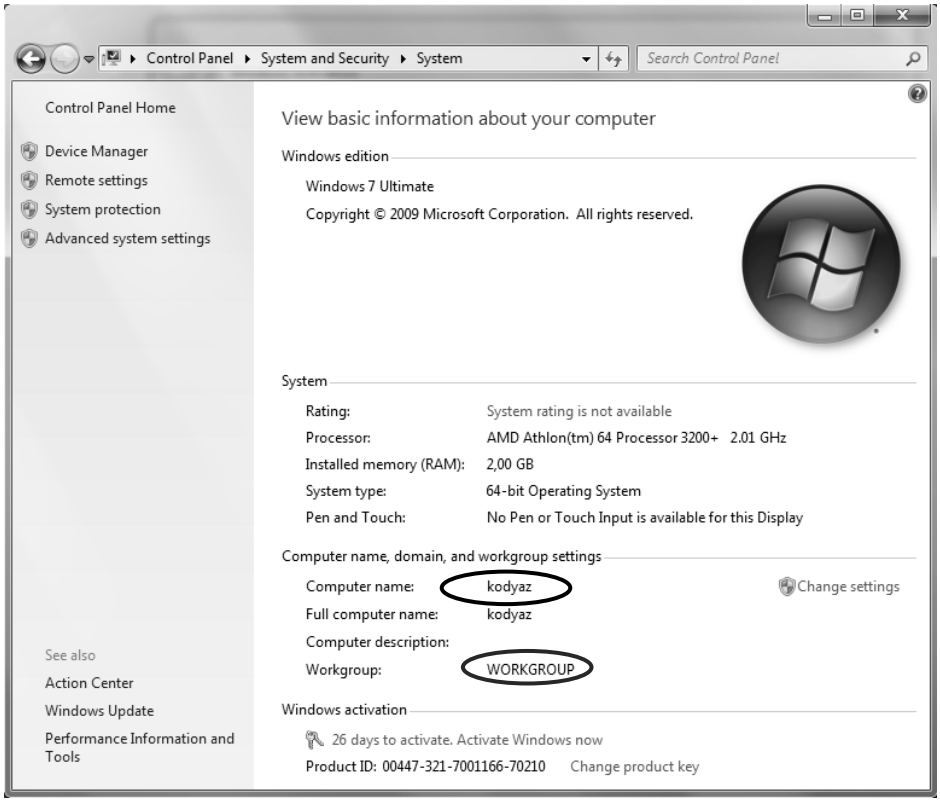
✧ Open “Network and sharing center”, select “turn off password protected sharing” and “Open sharing…..”



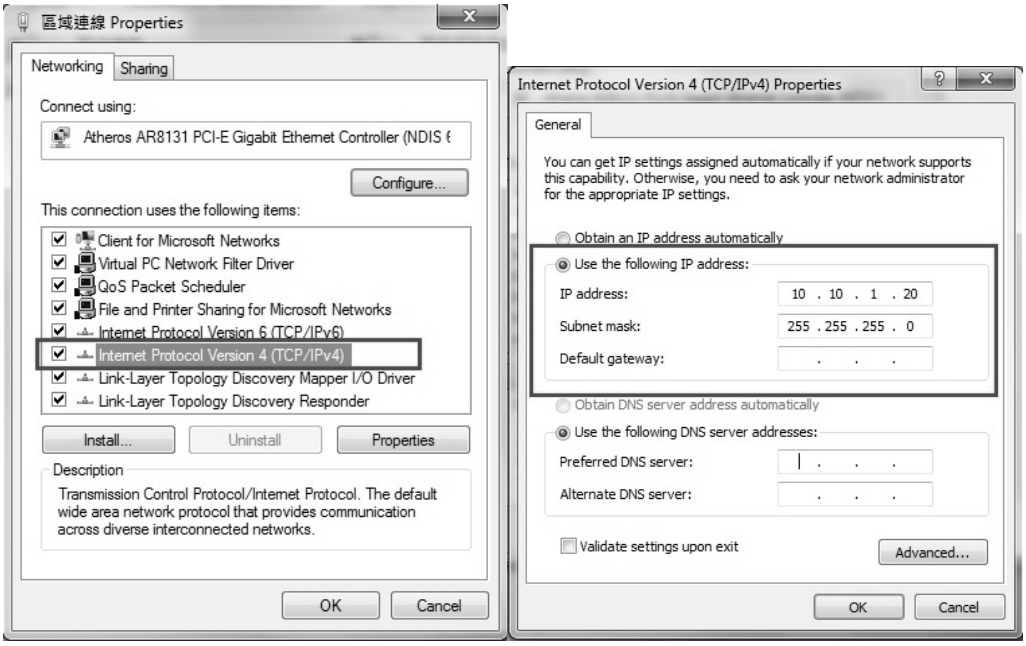


2. Setting PC name and workgroup

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



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

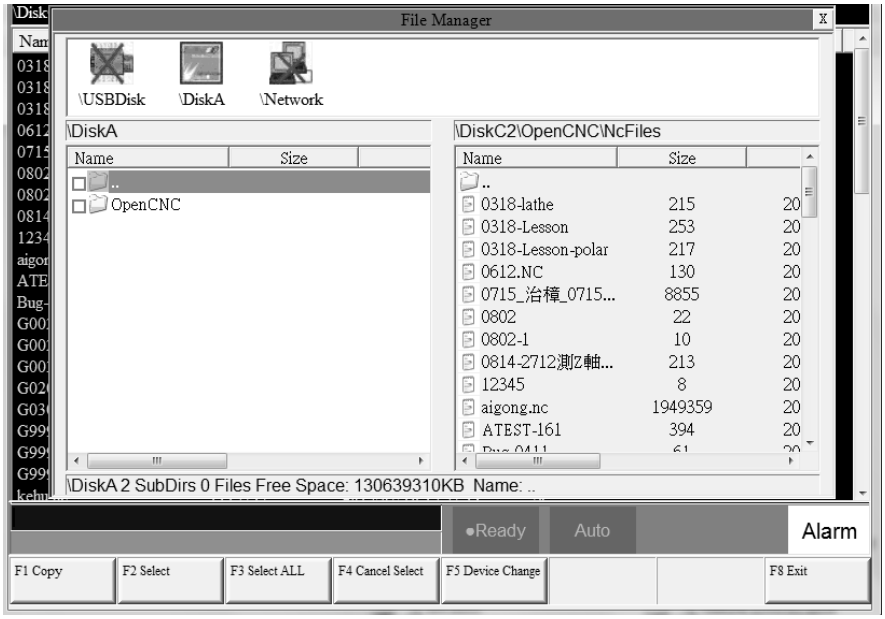


3.5 File Transfer

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

3.5.1 File Import

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



3.5.2 File Export

- Operation
  - ✧ Path: F2-program→ F8-file manager → F5 File export
  - ✧ Other interface will appear on screen, press F5-device change to move cursor to desired external device on the status bar
  - ✧ Press [Enter] key to access to inside device, if the device icon has red cross, which means that there is no connection to this device, Select the file wants to export and press [Copy] to complete export file
  - ✧ Press F4 cancel select to cancel the selected file
  - ✧ After complete file transfer, press [left] or [ESC] to leave this screen
- Note:
  - ✧ if destination of export file does not exist, below alarm will appear

4 Appendix

4.1 Release Note

Doc. Ver.	Content	Release Date	Author	Reviwer	CNC Ver.
1.0	1st Version	2013/11/08	Sandy.duan	Yulius.Duma	1.0.6
02					
03					
04					
05					

4.2 Contact Window

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

Room1202 Unit2 Buiding1, Edifice DiKai , Jincheng Road, Beigan Street,Xiaoshan District , Hangzhou City,Zhejiang Province, China TEL：+86-571-82751187 FAX：+-86-571-82751186	Room 12-07,No.262,416 Alley,Zhaohui Road,Jiangdong District, Ningbo City. TEL：+86-574-87750305 FAX：+86-574-87750306
Zhejiang-Wenling The Suzhou new generation of CNC Equipment Co., Ltd. Wenling Branch Room 1206, Department A, Zhenxing Plaza, Taiping Subdistrict, Wenling City TEL：0576- 86138372 FAX：0576-86119106	Guangdong-Guangzhou SuZhou SYNTEC Equipment CO.,Ltd. Guangzhou Branch Room 403 Stairs1 Youyi Building!ALuojia Village 8,Fuyi Road,ShijiTown,Panyu Distrct,Guangzhou City,Guangdong Province, China TEL：+86-20-34583040 FAX：+86-20-34583220
Guangdong-Dongguan SuZhou SYNTEC Equipment CO.,Ltd. Dongguan Branch Room 705 JinGuo Business affairs centre Xin an District Chang an TEL：+86-769-81660318 FAX：+86-769-81660328	Shaanxi-Xi’an The Suzhou new generation of CNC Equipment Co., Ltd. Xi'an Office Room 2503,Unit 1,Rancho Santa Fe,No.36, Dianzi Third Road, Yanta District, Xi’an, Shaanxi TEL：029-88287423 FAX：029-88287423
Shaanxi-Baoji The Suzhou new generation CNC Equipment Co., Ltd. Baoji offices Room 2401, Building 1, Xinjian Road, Weibin District, Baoji TEL：18700712118	Shandong-Jinan SuZhou SYNTEC Equipment CO.,Ltd. Jinan Branch Room 1112, Unit A, Jiahui Global Plaza, No.548, Beiyuan Street, Tianqiao District, Jinan, China TEL：+86-53185907208 FAX：+86-53185905708
Fujian-Xiamen SuZhou SYNTEC Equipment CO.,Ltd. Xiamen Branch No. 100,B Chuang Chang Fang 3F East,Jin Fu Road,Tong An District,Xiamen City, Fujian Province, China TEL：+86-592-7191901 FAX：+86-592-7220536	Tianjin The Suzhou new generation of CNC Equipment Co., Ltd. Tianjin Branch Haiyi International 3-1416, No.2, Rongyuan Road, Huayuan Industries Park, Binhai Hi-and-New Tech Park, Tianjin TEL：+86-22-87134111 FAX：022-87134111

Jiangsu-Nanjing The Suzhou new generation of CNC Equipment Co., Ltd. Chongqing Branch Room 505, Unit 3, Building 12,Wuyi Luzhou Guan Chu Yuan, No.99, Tianyuan Middle Road, Jiangning District, Nanjing TEL：0512-69008860-300	Hubei-Wuhan Suzhou new generation of CNC Equipment Co., Ltd. Wuhan Office Room2003-2006,A#Guangguguoji,456 #Luoyu Road,Donghukaifaqu,Wuhan City, China TEL：027-87638876 FAX：027-87204137
Liaoning-Shenyang The Suzhou new generation CNC Equipment Co., Ltd. Shenyang Office No.3 A 1-12-3, South Wenhui Street, Shenhe District, Shenyang, Liaoning TEL：15002491395 FAX：18698806526	Anhui-Hefei The new generation of CNC Equipment Co., Ltd. in Suzhou, Hefei Office Room 704, Building 13,Wanzhen Xiaoyao Garden Ⅳ, Taihu East Road, Hefei TEL：15951989576
Thailand SIAM RADERMEN Co., Ltd. 52/4,MOO 5,THEPARAK RD.(KM11.5)BANGPLEEYAI,BANGPLEE, SAMUTPRAKRAN 10540 TEL：+66-27553536 FAX：+66-27575476 E-Mail：lee_cheahow@yahoo.com	Malaysia SURE FIRST ENTERPRISE (M) SDN BHD. 47000 NO 19.JALAN TSB 6,TAMAN INDUSTRI SUNGAI BULOH.47000 SELANGOR DARUL.EHSAN TEL：+603-61577632 FAX：+603-61574632
Lebanon / Damascus Kesmeyan Group Co. Lebanon Beirut, Jdeideh - Ammaieh - Kessmeyan Group Build. Damascus TEL: +963 11 6731602 Mob.: +963 988 211407 E-Mail : info@rockwell.tw TEL：+961-76-883778 FAX：+961-1-901807 E-Mail：kesmeyan@rockwell.tw	France INTEGRATION CNC 6, Rue l'archė de Copigny, 76630 BAILLY, FRANCE TEL：+33 2 35 06 07 83 FAX：+33 2 35 06 07 83 http://www.integrationcnc.fr/ E-Mail：vaque.j@online.fr
Turkey KASIKCIOGLU ELEKTROMEKANIK OTOMASYON Besevler K.S.S. 17 BLOK NO:72	ISTANBUL/TURKEY AKSIS MAKINA Yeni doğan mahallesi karakaş Sk. Emintaş Erciyes Sanayi Sitesi no:12/206

Nilufer/BURSA/TURKYIE TEL：+90-224-4434684 FAX：+90-224-4434685 E-Mail：murat@kasikcioglu.com	TEL：+90 212 613 87 83 FAX：+90 212 613 87 85 http://www.aksismakina.com E-Mail：info@aksismakina.com